

SimHydraulics<sup>®</sup>

User's Guide



MATLAB<sup>®</sup>&SIMULINK<sup>®</sup>

R2015b



## How to Contact MathWorks



Latest news: [www.mathworks.com](http://www.mathworks.com)  
Sales and services: [www.mathworks.com/sales\\_and\\_services](http://www.mathworks.com/sales_and_services)  
User community: [www.mathworks.com/matlabcentral](http://www.mathworks.com/matlabcentral)  
Technical support: [www.mathworks.com/support/contact\\_us](http://www.mathworks.com/support/contact_us)



Phone: 508-647-7000



The MathWorks, Inc.  
3 Apple Hill Drive  
Natick, MA 01760-2098

*SimHydraulics*<sup>®</sup> *User's Guide*

© COPYRIGHT 2006–2015 by The MathWorks, Inc.

The software described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from The MathWorks, Inc.

**FEDERAL ACQUISITION:** This provision applies to all acquisitions of the Program and Documentation by, for, or through the federal government of the United States. By accepting delivery of the Program or Documentation, the government hereby agrees that this software or documentation qualifies as commercial computer software or commercial computer software documentation as such terms are used or defined in FAR 12.212, DFARS Part 227.72, and DFARS 252.227-7014. Accordingly, the terms and conditions of this Agreement and only those rights specified in this Agreement, shall pertain to and govern the use, modification, reproduction, release, performance, display, and disclosure of the Program and Documentation by the federal government (or other entity acquiring for or through the federal government) and shall supersede any conflicting contractual terms or conditions. If this License fails to meet the government's needs or is inconsistent in any respect with federal procurement law, the government agrees to return the Program and Documentation, unused, to The MathWorks, Inc.

### **Trademarks**

MATLAB and Simulink are registered trademarks of The MathWorks, Inc. See [www.mathworks.com/trademarks](http://www.mathworks.com/trademarks) for a list of additional trademarks. Other product or brand names may be trademarks or registered trademarks of their respective holders.

### **Patents**

MathWorks products are protected by one or more U.S. patents. Please see [www.mathworks.com/patents](http://www.mathworks.com/patents) for more information.

## Revision History

March 2006	Online only	New for Version 1.0 (Release 2006a+)
September 2006	Online only	Revised for Version 1.1 (Release 2006b)
March 2007	Online only	Revised for Version 1.2 (Release 2007a)
September 2007	Online only	Revised for Version 1.2.1 (Release 2007b)
March 2008	Online only	Revised for Version 1.3 (Release 2008a)
October 2008	Online only	Revised for Version 1.4 (Release 2008b)
March 2009	Online only	Revised for Version 1.5 (Release 2009a)
September 2009	Online only	Revised for Version 1.6 (Release 2009b)
March 2010	Online only	Revised for Version 1.7 (Release 2010a)
September 2010	Online only	Revised for Version 1.8 (Release 2010b)
April 2011	Online only	Revised for Version 1.9 (Release 2011a)
September 2011	Online only	Revised for Version 1.10 (Release 2011b)
March 2012	Online only	Revised for Version 1.10.1 (Release 2012a)
September 2012	Online only	Revised for Version 1.11 (Release 2012b)
March 2013	Online only	Revised for Version 1.12 (Release 2013a)
September 2013	Online only	Revised for Version 1.13 (Release 2013b)
March 2014	Online only	Revised for Version 1.14 (Release 2014a)
October 2014	Online only	Revised for Version 1.15 (Release 2014b)
March 2015	Online only	Revised for Version 1.16 (Release 2015a)
September 2015	Online only	Revised for Version 1.17 (Release 2015b)



## Getting Started

1

<b>SimHydraulics Product Description</b> .....	1-2
Key Features .....	1-2
<b>SimHydraulics Assumptions and Limitations</b> .....	1-3
<b>Simulating Hydraulic Models</b> .....	1-4
Simulation Basics .....	1-4
Selecting a Solver .....	1-4
Specifying Simulation Accuracy/Speed Tradeoff .....	1-5
Nonphysical Values During Simulation .....	1-5
Troubleshooting Hydraulic Models .....	1-6
<b>Model Sharing</b> .....	1-8

## Modeling Hydraulic Systems

2

<b>SimHydraulics Block Libraries</b> .....	2-2
Library Structure Overview .....	2-2
Using the Simulink Library Browser to Access the Block Libraries .....	2-2
Using the Command Prompt to Access the Block Libraries ..	2-3
<b>Modeling Hydraulic Power and Control Systems</b> .....	2-5
<b>Essential Hydraulic Modeling Techniques</b> .....	2-6
Overview of Modeling Rules .....	2-6
Working with Fluids .....	2-7

<b>Creating and Simulating a Simple Hydraulic Model</b> . . . . .	<b>2-9</b>
Building a SimHydraulics Diagram . . . . .	2-9
Modifying Initial Settings . . . . .	2-16
Running the Simulation . . . . .	2-20
Adjusting the Parameters . . . . .	2-22
<b>Modeling Power Units</b> . . . . .	<b>2-27</b>
<b>Modeling Directional Valves</b> . . . . .	<b>2-31</b>
Types of Directional Valves . . . . .	2-31
4-Way Directional Valve Configurations . . . . .	2-32
Building Blocks and How to Use Them . . . . .	2-36
Build a Custom Directional Valve . . . . .	2-40
<b>Modeling Low-Pressure Fluid Transportation Systems</b> . . .	<b>2-44</b>
How Fluid Transportation Systems Differ from Power and Control Systems . . . . .	2-44
Available Blocks and How to Use Them . . . . .	2-46
Low-Pressure Fluid Transportation System . . . . .	2-48

# Getting Started

---

- “SimHydraulics Product Description” on page 1-2
- “SimHydraulics Assumptions and Limitations” on page 1-3
- “Simulating Hydraulic Models” on page 1-4
- “Model Sharing” on page 1-8

## **SimHydraulics Product Description**

### **Model and simulate hydraulic systems**

SimHydraulics provides component libraries for modeling and simulating hydraulic systems. It includes models of hydraulic components, such as pumps, valves, actuators, pipelines, and hydraulic resistances. You can use these components to model fluid power systems such as front-loader, power steering, and landing gear actuation systems. Fuel supply and water supply systems can also be modeled using SimHydraulics.

SimHydraulics models can be used to develop control systems and test system-level performance. You can parameterize your models using MATLAB® variables and expressions, and design control systems for your hydraulic system in Simulink®. You can add electrical, mechanical, pneumatic, and other components to your hydraulic model using Simscape™ and test them in a single simulation environment. To deploy models to other simulation environments, including hardware-in-the-loop (HIL) systems, SimHydraulics supports C-code generation.

### **Key Features**

- Pump models, including centrifugal, jet, and axial-piston pumps
- Directional valve models, including check valves and common configurations for 2-, 4-, and 6-way valves
- Flow and pressure control valve models, including ball, poppet, and pressure relief valves
- Translational and rotational actuator models, including optional friction and centrifugal forces
- Tank and pipe models with elevation effects for modeling fluid transportation systems
- Customizable library of common hydraulic fluids
- Ability to extend component libraries using the Simscape language
- Support for C-code generation



## SimHydraulics Assumptions and Limitations

SimHydraulics software performs transient analysis of hydro-mechanical systems. You may be able to use the higher-level library blocks, or you may need to build your actuators out of the lower-level library blocks. SimHydraulics software is specifically developed to cover modeling scenarios with hydraulic actuators as part of a control system. It is also appropriate for systems that allow consideration in lumped parameters.

SimHydraulics software is based on the assumption that fluid temperature remains constant during the simulated time interval, and this temperature must be set as a parameter together with the relative amount of entrapped air.

## Simulating Hydraulic Models

### In this section...

“Simulation Basics” on page 1-4

“Selecting a Solver” on page 1-4

“Specifying Simulation Accuracy/Speed Tradeoff” on page 1-5

“Nonphysical Values During Simulation” on page 1-5

“Troubleshooting Hydraulic Models” on page 1-6

### Simulation Basics

SimHydraulics software gives you multiple ways to simulate and analyze hydraulic power and control systems in the Simulink environment. Running a hydraulic simulation is similar to running a simulation of any other Simscape model. See “Desktop Simulation” in the Simscape documentation for a discussion of the following topics:

- Explanation of how SimHydraulics software validates and simulates a model
- Specifics of using Simulink linearization commands in SimHydraulics models
- Generating code for SimHydraulics models
- Restrictions and limitations on using Simulink tools in SimHydraulics models

All these aspects of simulating SimHydraulics models are exactly the same as for Simscape models.

### Selecting a Solver

SimHydraulics software supports all of the continuous-time solvers that Simscape supports. For more information, see “Setting Up Solvers for Physical Models” in the Simscape documentation.

You can select any of the supported solvers for running a SimHydraulics simulation. The variable-step solvers, `ode23t` and `ode15s`, are recommended for most applications because they run faster and work better for systems with a range of both fast and slow dynamics.

To use Simulink Coder™ software to generate standalone C or C++ code from your model, you must use the `ode14x` solver. For more information about code generation, see “Code Generation” in the Simscape documentation.

## Specifying Simulation Accuracy/Speed Tradeoff

To trade off accuracy and simulation time, adjust one or more of the following parameters:

- **Relative tolerance** (in the Configuration Parameters dialog box)
- **Absolute tolerance** (in the Configuration Parameters dialog box)
- **Max step size** (in the Configuration Parameters dialog box)
- **Constraint Residual Tolerance** (in the Solver Configuration block dialog box)

In most cases, the default tolerance values produce accurate results without sacrificing unnecessary simulation time. The parameter value that is most likely to be inappropriate for your simulation is **Max step size**, because the default value, `auto`, depends on the simulation start and stop times rather than on the amount by which the signals are changing during the simulation. If you are concerned about the solver missing significant behavior, change the parameter to prevent the solver from taking too large a step.

The Simulink documentation describes the following parameters in more detail and provides tips on how to adjust them:

- “Relative tolerance”
- “Absolute tolerance”
- “Max step size”

The `Solver Configuration` block reference page in the Simscape documentation explains when to adjust the **Constraint Residual Tolerance** parameter value.

## Nonphysical Values During Simulation

Sometimes SimHydraulics models may display physically unattainable values during simulation because of an irregularity within the model. In general, simulation does not stop if one or more variables assume nonphysical values, such as gauge pressures below  $-1$  bar, negative value of fluid volume in the reservoir, and so on. The rationale is that, at the end of simulation, the user knows the extent of the irregularity rather than just the fact that the reservoir is short of fluid or there is a pressure drop below vaporization level.

If you see that your model displays physically unattainable values during simulation, you must analyze your model and iteratively modify your design to smooth out the irregularities.

You can set your models to either warn or stop simulating with an error when absolute pressure in a hydraulic chamber falls below absolute zero. The default behavior is to stop simulation with an error. You can change this by using the **Hydraulic Fluid** block or the **Custom Hydraulic Fluid** block connected to the loop, and have the simulation continue with a warning. See the block reference pages for details.

## Troubleshooting Hydraulic Models

SimHydraulics simulations can stop before completion with one or more error messages. For a discussion of generic error types and error-fixing strategies, see “Troubleshooting Simulation Errors” in the Simscape documentation. The following troubleshooting techniques are specific to hydraulic models:

- Review the model configuration. If your error message contains a list of blocks, look at these blocks first. Also look for:
  - Wrong connections — Verify that the model makes sense as a hydraulic system. For example, look for accumulators connected to the pump outlet without check valves; cylinders connected against each other, so that they try to move in opposite directions; directional valves bypassed by a huge orifice, and so on.
  - Wrong use of hydraulic elements — SimHydraulics blocks model their respective hydraulic units within certain limits. For example, an Ideal Pressure Source block can simulate a pump only when the pressure remains constant (see “Modeling Power Units” on page 2-27). Similarly, the Pressure Relief Valve block is a steady-state representation of a real valve. A block may exhibit wrong behavior if it is placed in the wrong environment. Always check the validity of the model for a particular environment and the simulation objectives.
- Avoid portions of the system getting isolated from the main system. An isolated or "hanging" part of the system could affect computational efficiency and even cause failure of computation. Use the **Leakage Area** parameter, introduced specifically for this purpose, to maintain numerical integrity of the circuit. This parameter is present in all the directional valve blocks, pressure control and flow control valve blocks, and most of the variable orifices.
- Avoid "dry" nodes in a hydraulic system. By adding a hydraulic chamber to a node, you can considerably improve the convergence and computational efficiency of a model. Adding a chamber adds a degree of freedom. By adding a chamber, you replace a complex algebraic constraint (the dry node) by a dynamic constraint. In the mechanical domain, this is like replacing an ideal gearbox with a gearbox that includes compliance (spring and damper). The hydraulic chamber is represented

by the **Constant Volume Hydraulic Chamber** block in the Simscape Hydraulic Elements library.

MathWorks recommends that you build, simulate, and test your model incrementally. Start with an idealized, simplified model of your system, simulate it, verify that it works the way you expected. Then incrementally make your model more realistic, factoring in effects such as fluid compressibility, fluid inertia, and the other things that describe real-world phenomena. Simulate and test your model at every incremental step. Use subsystems to capture the model hierarchy, and simulate and test your subsystems separately before testing the whole model configuration. This approach helps you keep your models well organized and makes it easier to troubleshoot them.

## Model Sharing

The Simscape Editing Mode functionality lets you open, simulate, and save models that contain blocks from add-on products, including SimHydraulics blocks, in Restricted mode, without checking out add-on product licenses, as long as the products are installed on your machine.

This functionality allows a user, *model developer*, to build a model that uses Simscape and SimHydraulics blocks and share that model with other users, *model users*. When building the model in Full mode, the model developer must have both a Simscape license and a SimHydraulics license. Once the model is built, model users need only to check out a Simscape license to simulate the model and fine-tune its parameters in Restricted mode. As long as no structural changes are made to the model, model users can work in Restricted mode and do not need to check out SimHydraulics licenses.

Another workflow lets multiple users, who all have Simscape licenses, share a small number of SimHydraulics licenses by working mostly in Restricted mode, and temporarily switching models to Full mode only when they need to perform a specific design task that requires being in Full mode.

For a complete description of this functionality, see “Add-On Product License Management” in the Simscape documentation.

# Modeling Hydraulic Systems

---

- “SimHydraulics Block Libraries” on page 2-2
- “Modeling Hydraulic Power and Control Systems” on page 2-5
- “Essential Hydraulic Modeling Techniques” on page 2-6
- “Creating and Simulating a Simple Hydraulic Model” on page 2-9
- “Modeling Power Units” on page 2-27
- “Modeling Directional Valves” on page 2-31
- “Modeling Low-Pressure Fluid Transportation Systems” on page 2-44

# SimHydraulics Block Libraries

In this section...
“Library Structure Overview” on page 2-2
“Using the Simulink Library Browser to Access the Block Libraries” on page 2-2
“Using the Command Prompt to Access the Block Libraries” on page 2-3

## Library Structure Overview

SimHydraulics software uses the **Simscape** library as its main library. Simscape modeling environment provides the Physical Network approach for modeling and solving systems under design as one-dimensional networks. SimHydraulics software utilizes these basic modeling principles and contains a library of specialized hydraulic blocks that seamlessly interact with basic Simscape blocks. When modeling hydraulic power and control systems, you use the following Simscape libraries:

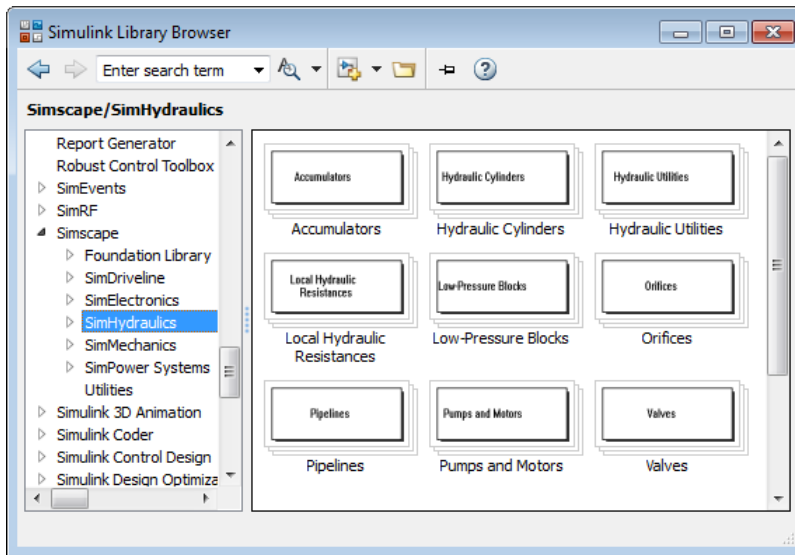
- Foundation library — Contains basic hydraulic, mechanical, and physical signal blocks
- SimHydraulics library — Contains advanced hydraulic diagram blocks, such as valves, cylinders, pipelines, pumps, and accumulators
- Utilities library — Contains essential environment blocks for creating Physical Networks models

You can combine all these blocks in your SimHydraulics diagrams to model hydraulic systems. You can also use the basic Simulink blocks in your diagrams, such as sources or scopes. See “Connecting Simscape Diagrams to Simulink Sources and Scopes” for more information on how to do this.

## Using the Simulink Library Browser to Access the Block Libraries

You can access the blocks through the Simulink Library Browser. To display the Library Browser, click the **Simulink Library** button in the toolbar of the MATLAB desktop. Alternatively, you can type `simulink` in the MATLAB Command Window. Then expand the **Simscape** entry in the contents tree.



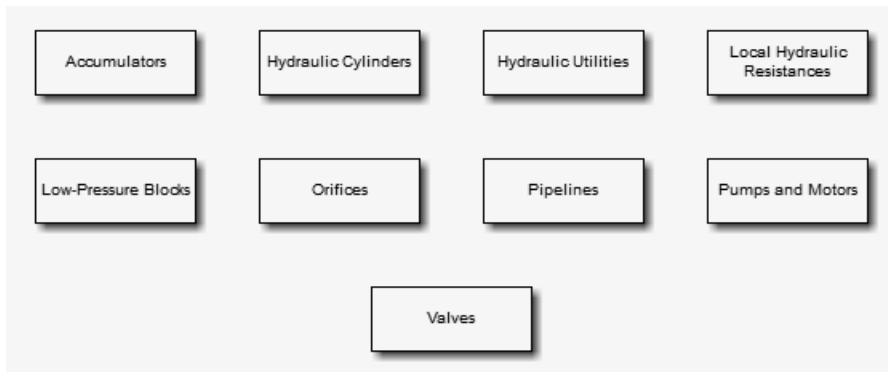


## Using the Command Prompt to Access the Block Libraries

Another way to access the block libraries is to open them individually by using the command prompt:

- To open just the SimHydraulics library, type `sh_lib` in the MATLAB Command Window.
- To open the Simscape library (to access the utility blocks, as well as hydraulic sources, sensors, and other Foundation library blocks), type `simscape` in the MATLAB Command Window.
- To open the main Simulink library (to access generic Simulink blocks), type `simulink` in the MATLAB Command Window.

The SimHydraulics library consists of nine top-level libraries, as shown in the following illustration. Some of these libraries contain second-level sublibraries. You can expand each library by double-clicking its icon.



## Modeling Hydraulic Power and Control Systems

When you model and analyze a hydraulic power or control system using SimHydraulics software, your workflow might include the following tasks:

- 1** Create a Simulink model that includes hydraulic or hydro-mechanical components.

In the majority of applications, it is most natural to model the physical system using Simscape and SimHydraulics blocks, and then develop the controller or signal processing algorithm in Simulink.

For more information about modeling the physical system, see “Essential Hydraulic Modeling Techniques” on page 2-6. For specific modeling techniques, see category-specific topics like “Modeling Power Units” on page 2-27 or “Modeling Low-Pressure Fluid Transportation Systems” on page 2-44.

- 2** Define component data by specifying hydraulic or mechanical properties as defined on a datasheet.

For more information about parameterizing blocks, see individual block reference pages. For information on valve parameterization, see “Modeling Directional Valves” on page 2-31.

- 3** Configure the solver options.

For more information about the settings that most affect the solution of a physical system, see “Setting Up Solvers for Physical Models” in the Simscape documentation.

- 4** Run the simulation.

For more information on how to perform time-domain simulation of a hydraulic system, see “Simulating Hydraulic Models” on page 1-4.



## Essential Hydraulic Modeling Techniques

<b>In this section...</b>
“Overview of Modeling Rules” on page 2-6
“Working with Fluids” on page 2-7

### Overview of Modeling Rules

SimHydraulics models are essentially Simscape block diagrams. When building a SimHydraulics model, you use a combination of SimHydraulics blocks with the blocks from the Simscape Foundation and Utilities libraries. Each SimHydraulics diagram must have at least one Solver Configuration block from the Simscape Utilities library. You can use basic hydraulic, electrical, and one-dimensional translational and rotational mechanical elements from the Simscape Foundation library and directly connect them to SimHydraulics blocks. You can also use basic Simulink blocks, such as sources and scopes, but you need to connect them through the Simulink-PS Converter and PS-Simulink Converter blocks from the Simscape Utilities library.

The rules that you must follow when building a hydraulic model are described in “Basic Principles of Modeling Physical Networks” in the Simscape documentation. This section briefly reviews these rules.

- SimHydraulics blocks, in general, feature Conserving ports  and Physical Signal inports and outports .
- There are three types of Physical Conserving ports used in SimHydraulics blocks: hydraulic, mechanical translational, and mechanical rotational. Each type has specific Through and Across variables associated with it.
- You can connect Conserving ports only to other Conserving ports of the same type.
- The Physical connection lines that connect Conserving ports together are nondirectional lines that carry physical variables (Across and Through variables, as described above) rather than signals. You cannot connect Physical lines to Simulink ports or to Physical Signal ports.
- Two directly connected Conserving ports must have the same values for all their Across variables (such as pressure or angular velocity).
- You can branch Physical connection lines. When you do so, components directly connected with one another continue to share the same Across variables. Any

Through variable (such as flow rate or torque) transferred along the Physical connection line is divided among the multiple components connected by the branches. How the Through variable is divided is determined by the system dynamics.

For each Through variable, the sum of all its values flowing into a branch point equals the sum of all its values flowing out.

- You can connect Physical Signal ports to other Physical Signal ports with regular connection lines, similar to Simulink signal connections. These connection lines carry physical signals between SimHydraulics blocks.
- You can connect Physical Signal ports to Simulink ports through special converter blocks. Use the Simulink-PS Converter block to connect Simulink outputs to Physical Signal inports. Use the PS-Simulink Converter block to connect Physical Signal outputs to Simulink inports.
- Unlike Simulink signals, which are essentially unitless, Physical Signals can have units associated with them. SimHydraulics block dialogs let you specify the units along with the parameter values, where appropriate. Use the converter blocks to associate units with an input signal and to specify the desired output signal units.

For examples of applying these rules when creating an actual hydraulic model, see “Creating and Simulating a Simple Hydraulic Model” on page 2-9.

MathWorks recommends that you build, simulate, and test your model incrementally. Start with an idealized, simplified model of your system, simulate it, verify that it works the way you expected. Then incrementally make your model more realistic, factoring in effects such as friction loss, motor shaft compliance, hard stops, and the other things that describe real-world phenomena. Simulate and test your model at every incremental step. Use subsystems to capture the model hierarchy, and simulate and test your subsystems separately before testing the whole model configuration. This approach helps you keep your models well organized and makes it easier to troubleshoot them.

## Working with Fluids

A change in the working fluid of your SimHydraulics model affects the global parameters of the system. Global parameters, determined by the type of working fluid, are used in equations for most hydraulic blocks. For example, valves, orifices, and pipelines use fluid density and fluid kinematic viscosity; chambers and cylinders use fluid bulk modulus; and so on. When you change the type of fluid, the appropriate changes to the global parameter values are propagated to all the blocks in the hydraulic circuit.

Each topologically distinct hydraulic circuit in a diagram requires exactly one hydraulic fluid to be associated with it. You can specify the fluid by connecting a Hydraulic Fluid block or Custom Hydraulic Fluid block to the circuit.

- The Custom Hydraulic Fluid block, available in the Simscape Foundation library, lets you directly specify the fluid properties, such as fluid density, kinematic viscosity, bulk modulus, and the amount of entrapped air, in the block dialog.
- The Hydraulic Fluid block lets you select a type of fluid from a predefined list and specify the amount of entrapped air and fluid temperature. SimHydraulics software determines the fluid properties associated with this type of fluid and these conditions, and displays them in the block dialog.

In both cases, SimHydraulics software then applies the fluid properties as global parameters to all the blocks in the hydraulic circuit.

---

**Note** If no Hydraulic Fluid block or Custom Hydraulic Fluid block is attached to a circuit, the hydraulic blocks in this circuit use the default fluid, which is equivalent to fluid defined by a Custom Hydraulic Fluid block with the default parameter values. See the [Custom Hydraulic Fluid block reference page](#) for more information.

---

## Creating and Simulating a Simple Hydraulic Model

### In this section...

“Building a SimHydraulics Diagram” on page 2-9

“Modifying Initial Settings” on page 2-16

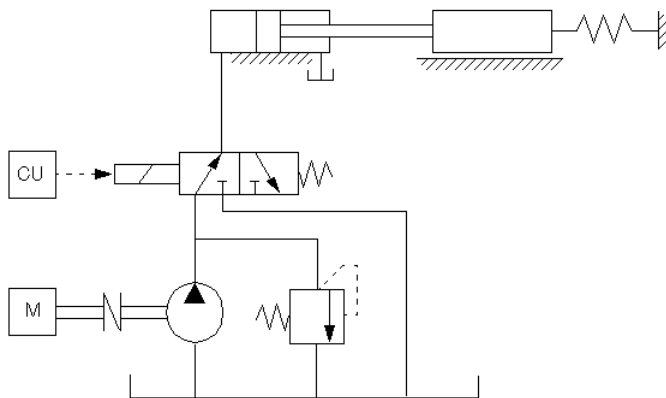
“Running the Simulation” on page 2-20

“Adjusting the Parameters” on page 2-22

### Building a SimHydraulics Diagram

In this example, you are going to model a simple hydraulic system and observe its behavior under various conditions. This tutorial illustrates the essential steps to building a hydraulic model, described in the previous section, and makes you familiar with using the basic SimHydraulics blocks.

The following schematic represents the model you are about to build. It contains a single-acting hydraulic cylinder, which is controlled by an electrically operated 3-way directional valve. The cylinder drives a load consisting of a mass, viscous friction, and preloaded spring.



The power unit consists of a motor, a positive-displacement pump, and a pressure relief valve. Depending on its characteristics, such a power unit can be modeled in a variety of ways, as described in “Modeling Power Units” on page 2-27. In this example, the pump unit is assumed to be powerful enough to maintain constant pressure at the valve inlet. Therefore, we are going to represent it in the diagram by a Hydraulic Pressure Source block.

To create an equivalent SimHydraulics diagram, follow these steps:

- 1 Open the Simulink Library Browser, as described in “SimHydraulics Block Libraries” on page 2-2.
- 2 Create a new model. To do this, from the top menu bar of the Library Browser, select **File > New > Model**. The software creates an empty model in memory and displays it in a new model editor window.

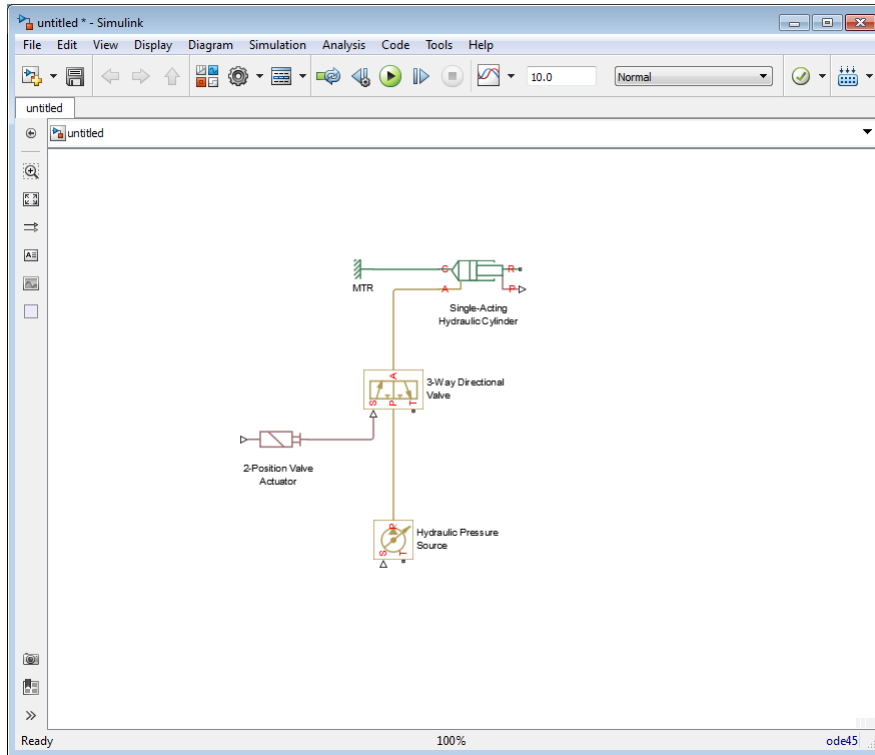
---

**Note** Alternately, you can type `ssc_new` at the MATLAB Command prompt, to create a new model prepopulated with certain required and commonly-used blocks. For more information, see “Creating a New Simscape Model”.

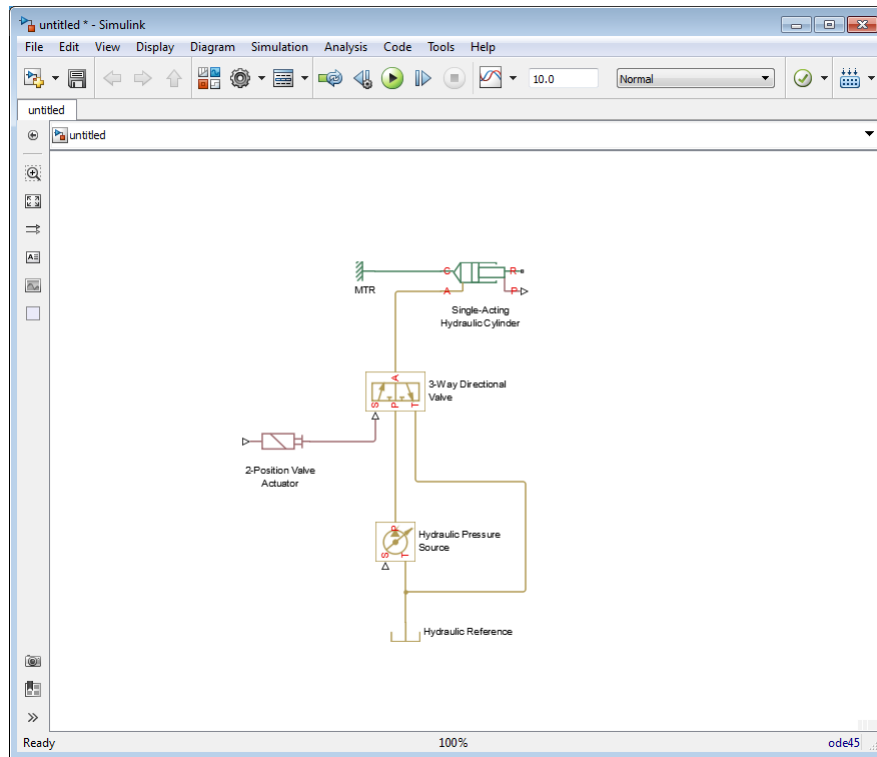
---

- 3 Open the Simscape > Foundation Library > Hydraulic > Hydraulic Sources library and drag the Hydraulic Pressure Source block into the model window.
- 4 Open the Simscape > SimHydraulics > Hydraulic Cylinders library and place the Single-Acting Hydraulic Cylinder block into the model window.
- 5 To model the valve, open the Simscape > SimHydraulics > Valves library. Place the 3-Way Directional Valve block, found in the Directional Valves sublibrary, and the 2-Position Valve Actuator block, found in the Valve Actuators sublibrary, into the model window.
- 6 Connect the blocks as shown in the following illustration.





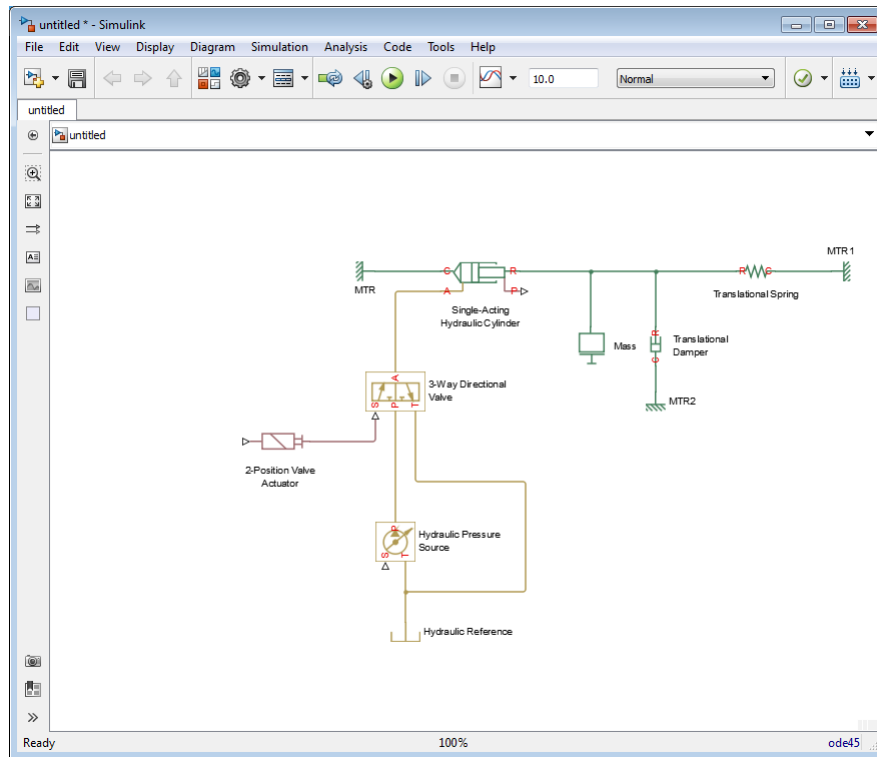
- 7 Ports T of the Hydraulic Pressure Source and 3-Way Directional Valve blocks have to be connected to the tank, at atmospheric pressure. To model this connection, open the Simscape > Foundation Library > Hydraulic > Hydraulic Elements library and add the Hydraulic Reference block to your diagram, as shown below. To do this, connect the only port of the Hydraulic Reference block to port T of the Hydraulic Pressure Source block, then right-click this connection line to create a branching point, and connect this point to port T of the 3-Way Directional Valve block.



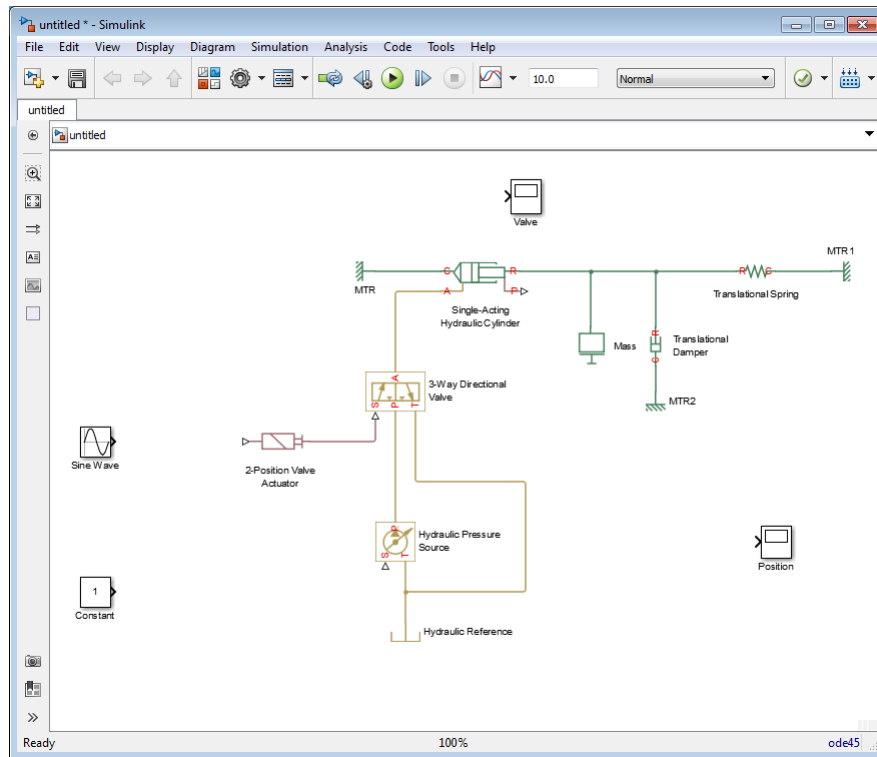
- 8 Model the mechanical load for the cylinder. Open the Simscape > Foundation Library > Mechanical > Translational Elements library and add the Mass, Translational Spring, Translational Damper, and three Mechanical Translational Reference blocks to your diagram.


To indicate that the cylinder case is fixed, connect port C of the Single-Acting Hydraulic Cylinder block to one of the Mechanical Translational Reference blocks. To rotate the Mechanical Translational Reference block, select it and press **Ctrl+R**. You can also shorten the block name to MTR to make the diagram easier to read.

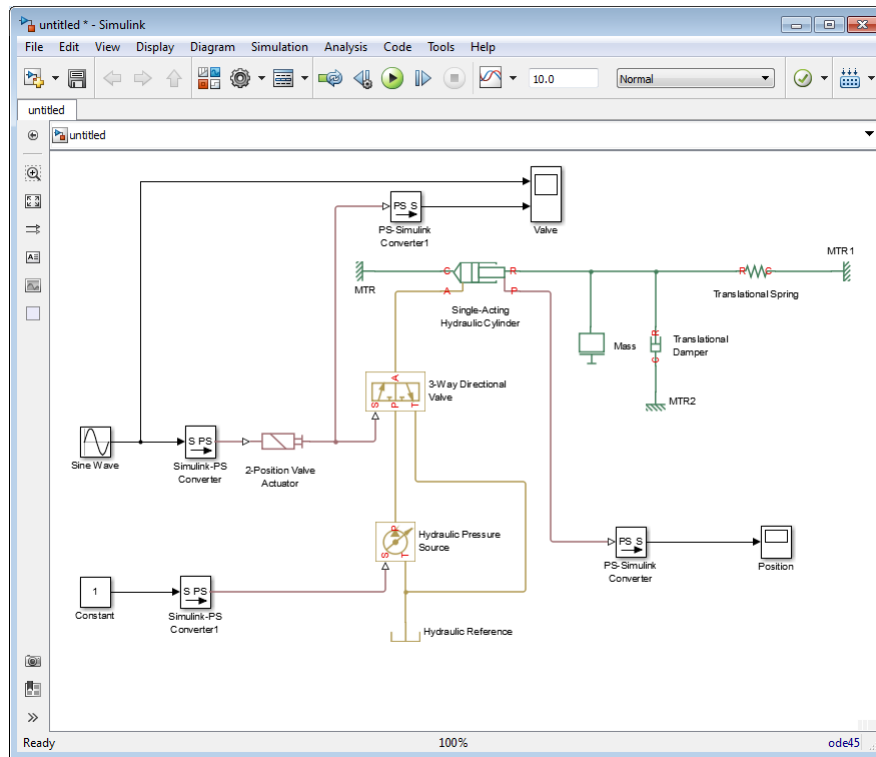
Connect the other blocks to port R of the Single-Acting Hydraulic Cylinder block, as shown below.



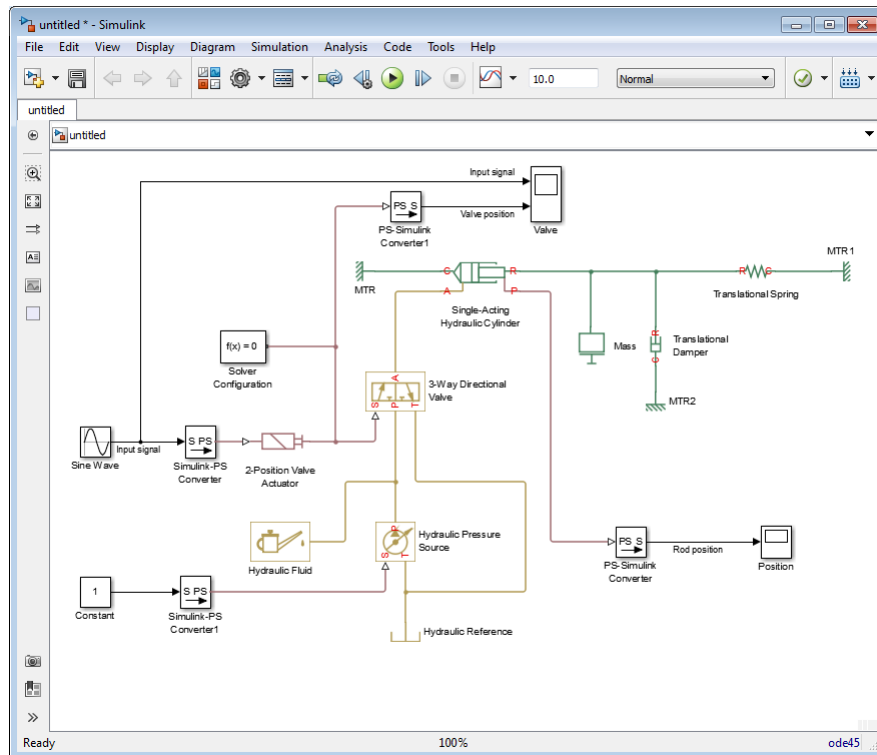
- 9 Now you need to add the sources and scopes. They are found in the regular Simulink libraries. Open the Simulink > Sources library and copy the Constant block and the Sine Wave block into the model. Then open the Simulink > Sinks library and copy two Scope blocks. Rename one of the Scope blocks to **Valve**. It will monitor the valve opening based on the input signal variation. The other Scope block will monitor the position of the cylinder rod; rename it to **Position**.



- 10 Double-click the **Valve** scope to open it. In the scope window, click  to access the scope parameters, change **Number of axes** to 2, then click the **Layout** button, select the vertical layout (two squares on top of each other), and click **OK**. The scope window now displays two sets of axes, and the **Valve** scope in the diagram has two input ports.
- 11 Every time you connect a Simulink source or scope to a SimHydraulics diagram, you have to use an appropriate converter block, to convert Simulink signals into physical signals and vice versa. Open the Simscape > Utilities library and copy two Simulink-PS Converter blocks and two PS-Simulink Converter blocks into the model. Connect the blocks as shown below.



- 12 To specify the fluid properties, add the Hydraulic Fluid block, found in the Simscape > SimHydraulics > Hydraulic Utilities library, to your diagram. You can add this block anywhere on the hydraulic circuit by creating a branching point and connecting it to the only port of the Hydraulic Fluid block.
- 13 Each topologically distinct physical network in a diagram requires exactly one Solver Configuration block, found in the Simscape > Utilities library. Copy this block into your model and connect it to the circuit, similar to the Hydraulic Fluid block. Your diagram now should look like this.



14 Your block diagram is now complete. Save it as `simple_hydro`.

### Modifying Initial Settings

After you have put together a block diagram of your model, as described in the previous section, you need to select a solver and provide the correct values for block parameters. All the blocks have default parameter values that allow them to run “out of the box,” but you may need to change some of them to suit your particular application.

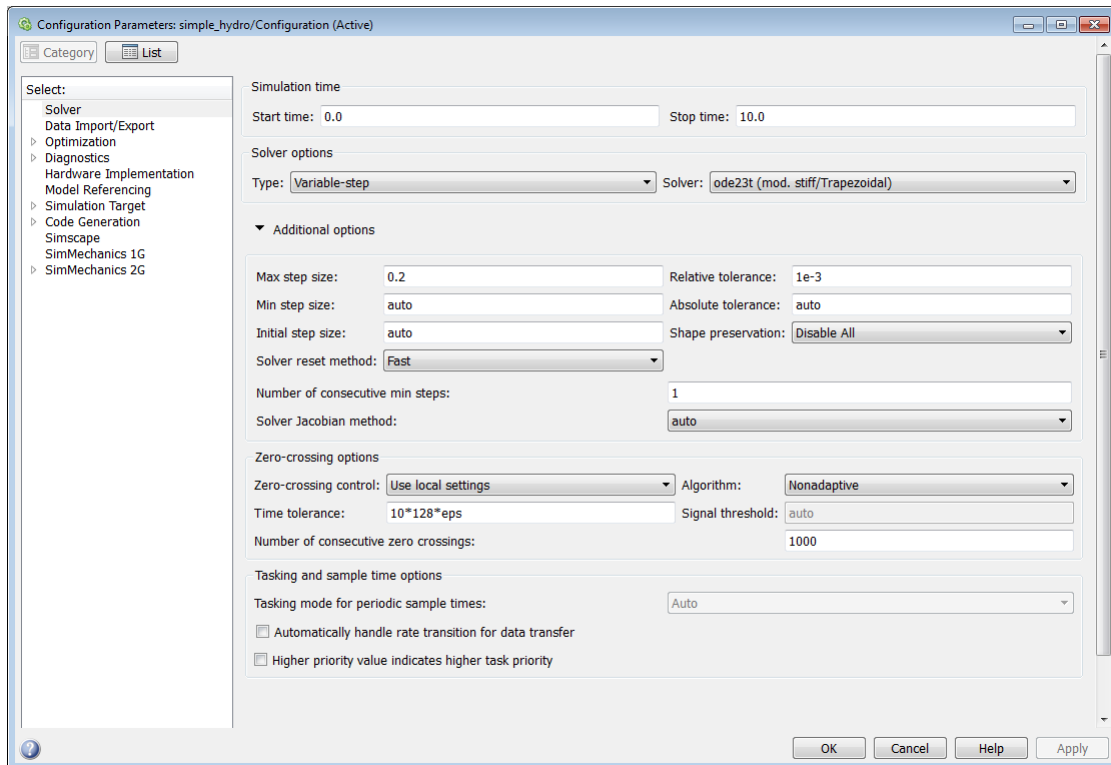
To prepare for simulating the model, follow these steps:

- 1 Select a Simulink solver. On the top menu bar of the model window, select **Simulation > Model Configuration Parameters**. The Configuration Parameters dialog box opens, showing the **Solver** node.

Under **Solver options**, set **Solver** to ode23t (mod.stiff/Trapezoidal).

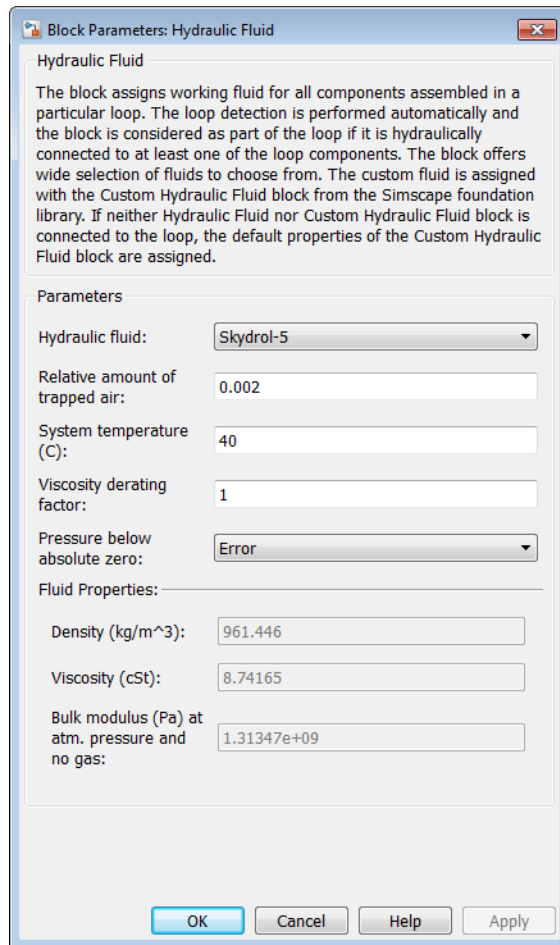
Expand **Additional options** and set **Max step size** to 0.2.

Also note that **Simulation time** is specified to be between 0 and 10 seconds. You can adjust this setting later, if needed.



Click **OK** to close the Configuration Parameters dialog box.

- 2 Select a fluid. Double-click the Hydraulic Fluid block. In the Block Parameters dialog box, set **Hydraulic fluid** to Skydrol 5 and set the other block parameters as shown below.

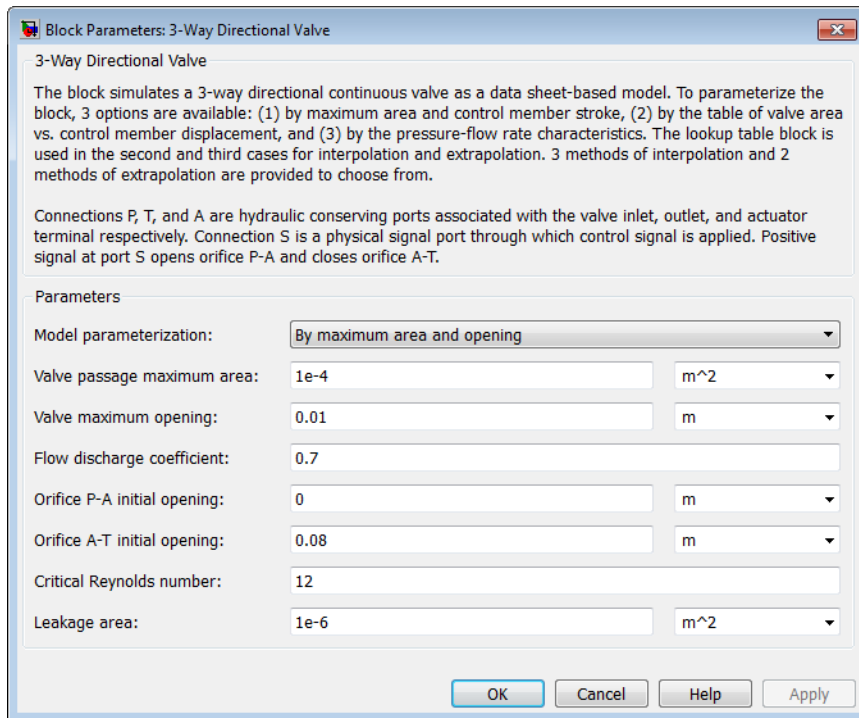


Click **OK** to close the Block Parameters dialog box.

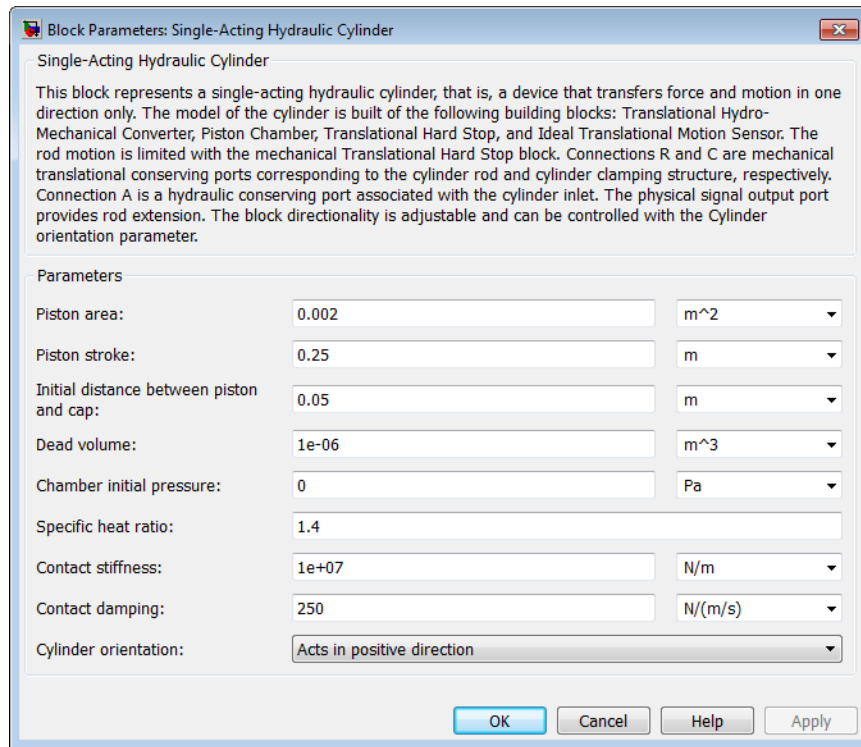
- 3 Specify the units for the pressure input signal. Simulink signals are unitless. When you convert them to physical signals, you can supply units by using the converter blocks. Double-click the Simulink-PS Converter1 block, enter **Pa** in the **Input signal unit** combo box, and click **OK**. When the physical modeling software parses the model, it matches the input signal units with the block input ports and provides error messages if there is a discrepancy. For more information, see “Model Validation”.



- 4 Specify a realistic value for the pressure input signal. Double-click the Constant block, enter  $10e5$  in the **Constant value** text box, and click **OK**.
- 5 Open the 2-Position Valve Actuator block and note that its **Nominal Signal Value** parameter is set to **24**.
- 6 Double-click the Sine Wave block and change its **Amplitude** to a value greater than 50% of the nominal signal value for the 2-Position Valve Actuator block, for example, to **20**.
- 7 Adjust the 3-Way Directional Valve block parameters as shown below.



- 8 Adjust the Single-Acting Hydraulic Cylinder block parameters as shown below.



- 9 Double-click the Mass block and change its **Mass** to 4.5 kg.
- 10 Double-click the Translational Damper block, which models the viscous friction, and change its **Damping coefficient** to 250 N/(m/s).
- 11 Double-click the Translational Spring block. Set its **Spring rate** to 6e3 N/m and **Initial deformation** to 0.02 m.
- 12 Save the model.

### Running the Simulation


After you've put together a block diagram and specified the initial settings for your model, you can run the simulation.

- 1 The input signal for the valve opening is provided by the Sine Wave block. The Valve scope reflects both the input signal and the valve opening as functions of time. The

Position scope outputs the cylinder rod displacement as a function of time. Double-click both scopes to open them.

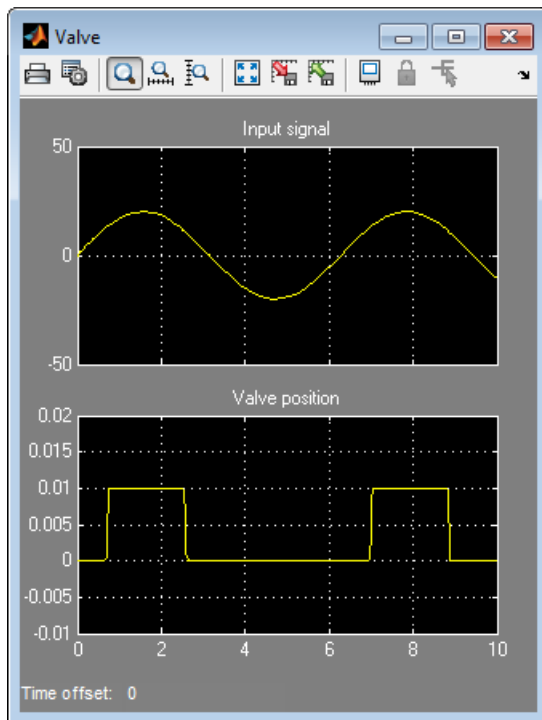
2

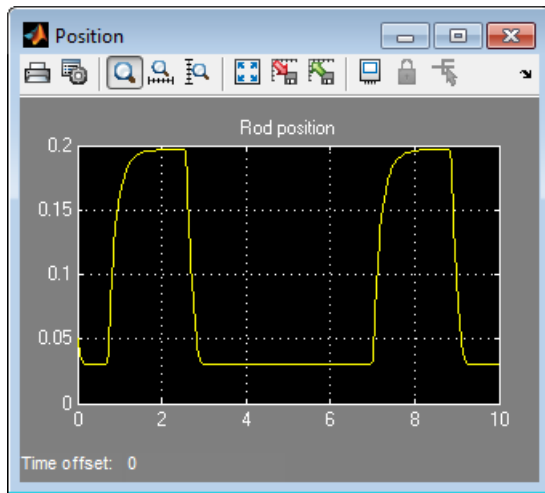


To run the simulation, click  in the model window toolbar. The physical modeling solver evaluates the model, calculates the initial conditions, and runs the simulation. For a detailed description of this process, see “How Simscape Simulation Works”. Completion of this step may take a few seconds. The message in the bottom-left corner of the model window provides the status update.

3

Once the simulation starts running, the Valve and Position scope windows display the simulation results, as shown in the next illustration.





In the beginning, the valve is closed. Then, as the input signal reaches 50% of the actuator's nominal signal, the valve gradually opens to its maximum value and moves the cylinder rod in the positive direction. When the input signal goes below 50% of the nominal signal, the actuator closes the valve. The spring returns the cylinder rod to its initial position.

You can now adjust various inputs and block parameters and see their effect on the valve opening profile and the cylinder rod displacement.

### Adjusting the Parameters

After running the initial simulation, you can experiment with adjusting various inputs and block parameters.

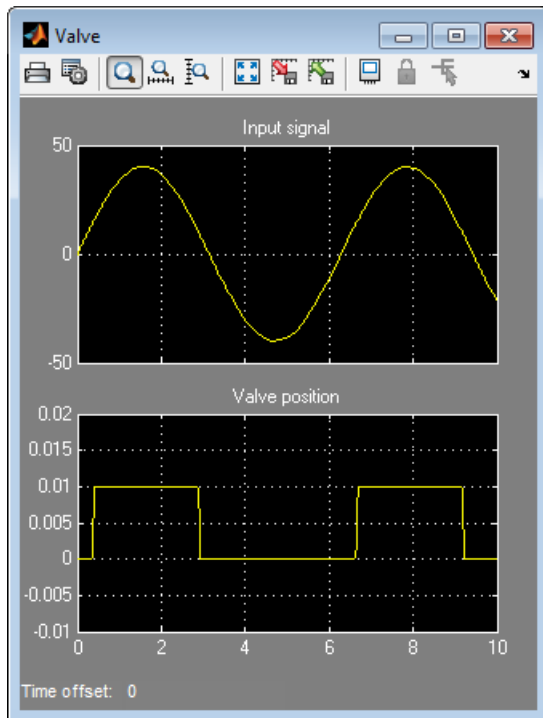
Try the following adjustments:

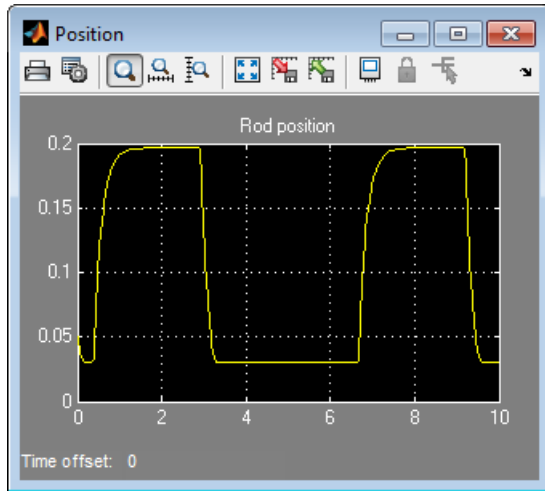
- 1 Change the input signal for valve opening.
- 2 Change the cylinder load parameters.
- 3 Change the rod position output units.

### Changing the Valve Input Signal

This example shows how a change in the input signal affects the opening of the valve, and therefore the cylinder rod displacement.

- 1 Double-click the Sine Wave block, enter 40 in the **Amplitude** text box, and click **OK**.
- 2 Run the simulation. The simulation results are shown in the following illustration. With the increase in the input signal amplitude, it reaches 50% of the actuator's nominal signal sooner, and the valve stays open longer, which in turn affects the cylinder rod position.

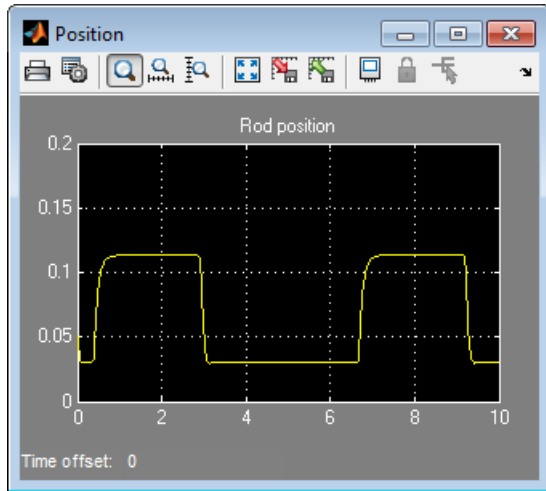




### Changing the Cylinder Load Parameters


In our model, the cylinder drives a load consisting of a mass, viscous friction, and preloaded spring. This example shows how a change in the spring stiffness affects the cylinder rod displacement.

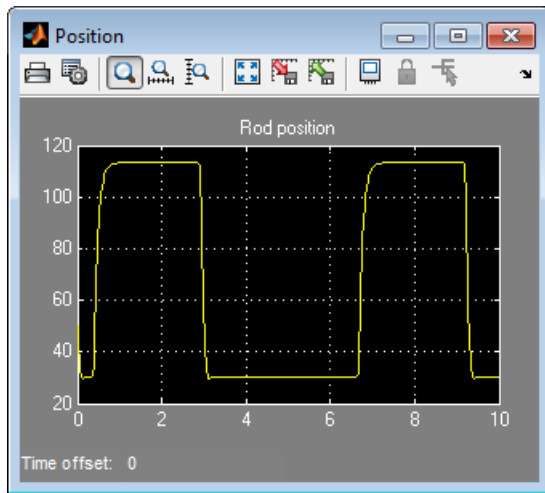
- 1 Double-click the Translational Spring block. Set its **Spring rate** to  $12e3$  N/m.
- 2 Run the simulation. The valve opening profile is not affected, but increase in spring stiffness results in smaller amplitude of cylinder rod displacement, as shown in the following illustration.



### Changing the Rod Position Output Units

In our model, we have used the PS-Simulink Converter block in its default parameter configuration, which does not specify units. Therefore, the `Position` scope outputs the cylinder rod displacement in the units specified for the parameters of the Single-Acting Hydraulic Cylinder block; in this case, in meters. This example shows how to change the output units for the cylinder rod displacement to millimeters.

- 1 Double-click the PS-Simulink Converter block. Type `mm` in the **Output signal unit** text box and click **OK**.
- 2 Run the simulation. In the `Position` scope window, click  to autoscale the scope axes. The cylinder rod displacement is now output in millimeters, as shown in the following illustration.

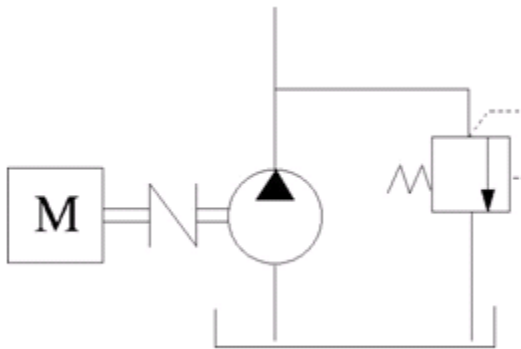




## Modeling Power Units

The power unit is perhaps the most prevalent unit in hydraulic systems. Its main function is to supply the required amount of fluid under specified pressure. There is a wide variety of power unit designs varying by the amount and type of pumps, prime movers, valves, tanks, etc. The set of blocks available in the SimHydraulics libraries allows you to simulate practically any of these configurations. This section considers basic approaches in simulating power units and examples of typical schematics.

A typical power unit of a hydraulic system, as shown in the following illustration, consists of a fixed-displacement or variable-displacement pump, reservoir, pressure-relief valve, and a prime mover that drives the hydraulic pump.



### Typical Hydraulic Power Unit

In developing a model of a power unit, you must reach a compromise between the robustness, speed of simulation, and accuracy, meaning that the model should be as simple as possible to provide acceptable accuracy within the working range of variable parameters.

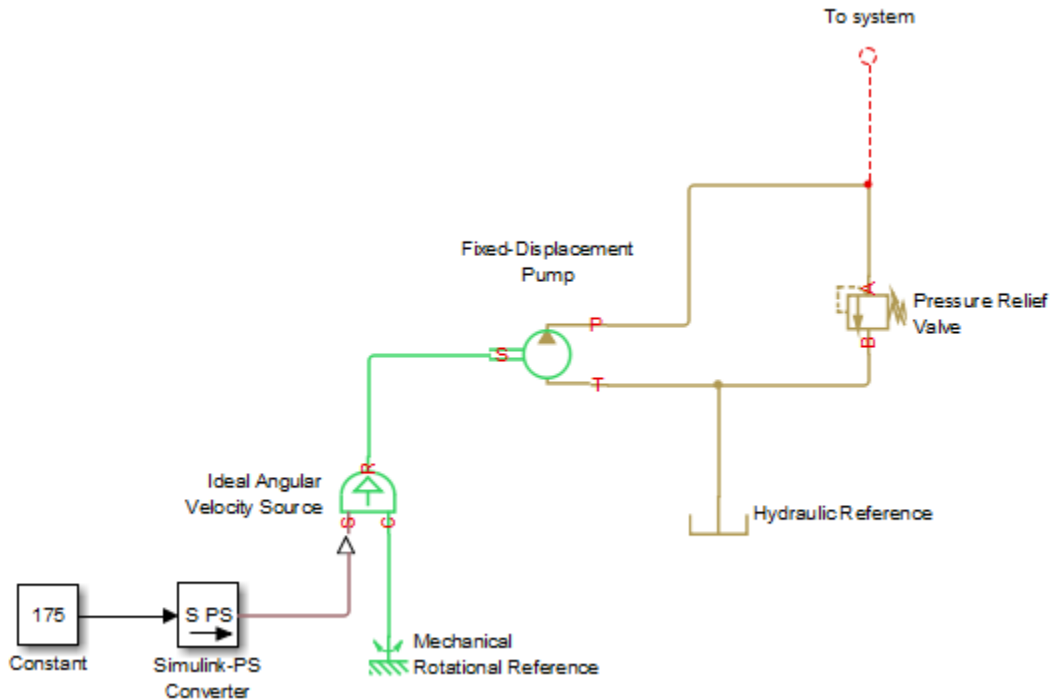
The first option is to simulate a power unit literally, as it is, reproducing all its components. This approach is illustrated in the Power Unit with Fixed-Displacement Pump example. The power unit consists of a fixed-displacement pump, which is driven by a motor through a compliant transmission, a pressure-relief valve, and a variable orifice, which simulates system fluid consumption. The motor model is represented as a source of angular velocity rotating shaft at 188 rad/s at zero torque. The load on the shaft decreases the velocity with a slip coefficient of 1.2 (rad/s)/Nm. The load on the driving

shaft is measured with the torque sensor. The shaft between the motor and the pump is assumed to be compliant and simulated with rotational spring and damper.

The simulation starts with the variable orifice opened, which results in a low system pressure and the maximum flow rate going to the system. The orifice starts closing at 0.5 s, and is closed completely at 3 s. The output pressure builds up until it reaches the pressure setting of the relief valve ( $75e5$  Pa) and is maintained at this level by the valve. At 3 s, the variable orifice starts opening, thus returning system to its initial state.

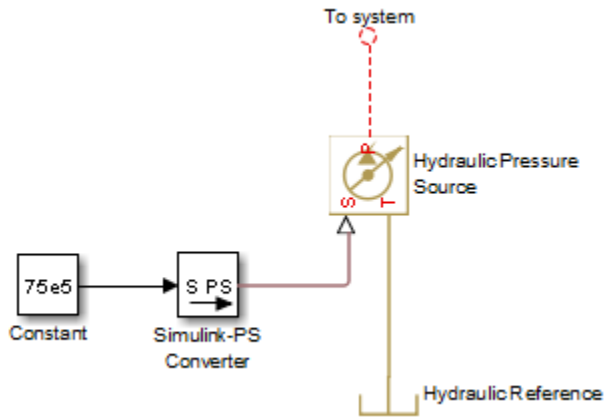
You can implement a considerably more complex model of a prime mover by following the pattern used in the example. For instance, the shaft can be represented with multiple segments and intermediate bearings. The model of a prime mover can be more comprehensive, accounting for its type (DC or AC electric motor, diesel or gasoline engine), characteristics, control type, and so on. In addition, a complex mechanical transmission driven by a diesel or gasoline internal combustion engine modeled using SimDriveline™ software can be combined with the SimHydraulics model of the hydraulic portion of a power unit.

Depending on your particular application, you may be able to simplify the model of a power unit practically without a loss in accuracy. The main factors to be considered in this process are the driving shaft angular velocity variation magnitude and the system pressure variation range. If the prime mover angular velocity remains practically constant during simulated time or varies insignificantly with respect to its steady-state value, the entire driving shaft subsystem can be replaced with the Ideal Angular Velocity Source block, whose output is set to the steady-state value, as it is shown in the following illustration.



### Using the Ideal Angular Velocity Source Block in Modeling Power Units

Furthermore, if pump delivery exceeds the system's fluid requirements at all times, the pump output pressure remains practically constant and close to the pressure setting of the pressure-relief valve. If this assumption is true and acceptable, the entire power unit can be reduced to an ideal Hydraulic Pressure Source block, as shown in the next illustration.



### Using the Hydraulic Pressure Source Block in Modeling Power Units

The two previous examples demonstrate that the use of ideal sources is a powerful means of reducing the complexity of models. However, you should exercise extreme caution every time you use an ideal source instead of a real pump. The substitution is possible only if there is an assurance that the controlled parameter (angular velocity in the first example, and pressure in the second example) remains constant. If this is not the case, the power unit represented with an ideal source will generate considerably more power than its simulated physical counterpart, thus making the simulation results incorrect.

## Modeling Directional Valves

### In this section...

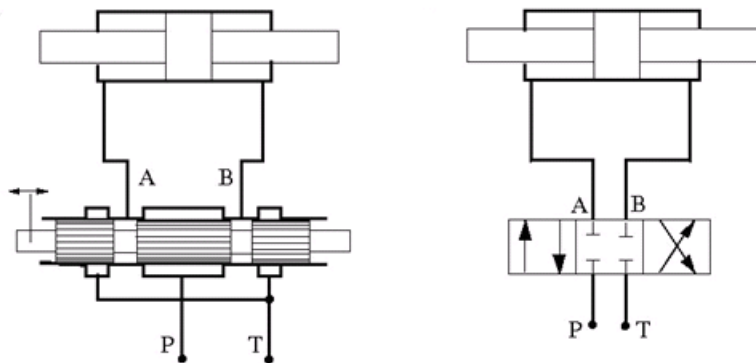
- “Types of Directional Valves” on page 2-31
- “4-Way Directional Valve Configurations” on page 2-32
- “Building Blocks and How to Use Them” on page 2-36
- “Build a Custom Directional Valve” on page 2-40

### Types of Directional Valves

The main function of directional valves in hydraulic systems is to direct and distribute flow between consumers. As far as the valve modeling is concerned, the valves are classified by the following main characteristics:

- Number of external paths (connecting ports) — One-way, two-way, three-way, four-way, multiple-way
- Number of positions a control member of the valve can assume — Two-position, three-position, multiple-position, continuous (can assume any position within working range)
- Control member type — Spool, poppet, sliding flat spool, and so on

As an example, the following illustration shows a portion of a hydraulic system with a 4-way, 3-position directional valve controlling a double-acting cylinder, next to its schematic diagram.



Throughout SimHydraulics libraries, hydraulic ports are identified with the following symbols:

- P — Pressure port
- T — Return (tank) port
- A, B — Actuator ports
- X, Y — Pilot or control ports

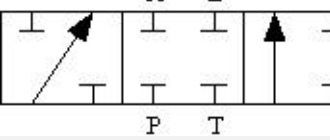
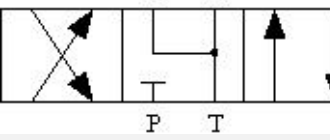
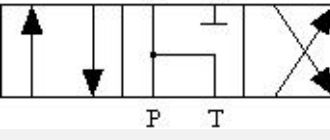
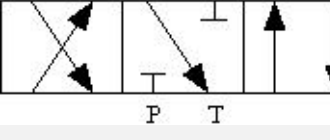
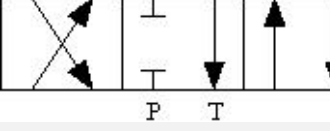
### 4-Way Directional Valve Configurations

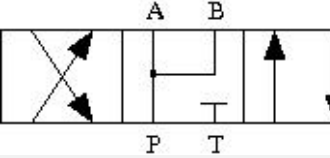
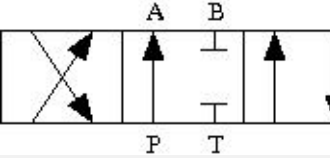
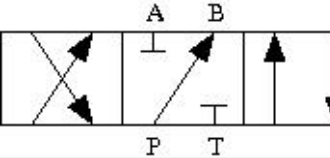
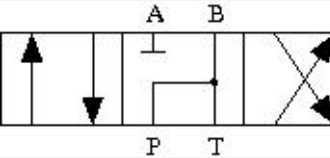
4-way directional valves are available in multiple configurations, depending on the port connections in three distinctive valve positions: leftmost, neutral, and rightmost. Each configuration is characterized by the number of variable orifices, the way the orifices are connected, and initial openings of the orifices. Ten 4-way directional valve blocks in SimHydraulics libraries represent twenty most typical valve configurations. Configurations that differ only by the values of initial openings are covered by the same model.

The basic 4-Way Directional Valve block lets you model eleven most popular configurations by changing the initial openings of the orifices, as shown in the following table.

#### Basic 4-Way Directional Valve Configurations

No	Configuration	Initial Openings
1		<p>All four orifices are overlapped in neutral position:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &lt; 0</li> <li>• Orifice P-B initial opening &lt; 0</li> <li>• Orifice A-T initial opening &lt; 0</li> <li>• Orifice B-T initial opening &lt; 0</li> </ul>
2		<p>All four orifices are open (underlapped) in neutral position:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &gt; 0</li> <li>• Orifice P-B initial opening &gt; 0</li> <li>• Orifice A-T initial opening &gt; 0</li> <li>• Orifice B-T initial opening &gt; 0</li> </ul>

No	Configuration	Initial Openings
3		<p>Orifices P-A and P-B are overlapped. Orifices A-T and B-T are overlapped for more than valve stroke:</p> <ul style="list-style-type: none"> <li>• <b>Orifice P-A initial opening</b> &lt; 0</li> <li>• <b>Orifice P-B initial opening</b> &lt; 0</li> <li>• <b>Orifice A-T initial opening</b> &lt; - <i>valve_stroke</i></li> <li>• <b>Orifice B-T initial opening</b> &lt; - <i>valve_stroke</i></li> </ul>
4		<p>Orifices P-A and P-B are overlapped, while orifices A-T and B-T are open:</p> <ul style="list-style-type: none"> <li>• <b>Orifice P-A initial opening</b> &lt; 0</li> <li>• <b>Orifice P-B initial opening</b> &lt; 0</li> <li>• <b>Orifice A-T initial opening</b> &gt; 0</li> <li>• <b>Orifice B-T initial opening</b> &gt; 0</li> </ul>
5		<p>Orifices P-A and A-T are open in neutral position, while orifices P-B and B-T are overlapped:</p> <ul style="list-style-type: none"> <li>• <b>Orifice P-A initial opening</b> &gt; 0</li> <li>• <b>Orifice P-B initial opening</b> &lt; 0</li> <li>• <b>Orifice A-T initial opening</b> &gt; 0</li> <li>• <b>Orifice B-T initial opening</b> &lt; 0</li> </ul>
6		<p>Orifice A-T is initially open, while all three remaining orifices are overlapped:</p> <ul style="list-style-type: none"> <li>• <b>Orifice P-A initial opening</b> &lt; 0</li> <li>• <b>Orifice P-B initial opening</b> &lt; 0</li> <li>• <b>Orifice A-T initial opening</b> &gt; 0</li> <li>• <b>Orifice B-T initial opening</b> &lt; 0</li> </ul>
7		<p>Orifice B-T is initially open, while all three remaining orifices are overlapped:</p> <ul style="list-style-type: none"> <li>• <b>Orifice P-A initial opening</b> &lt; 0</li> <li>• <b>Orifice P-B initial opening</b> &lt; 0</li> </ul>

No	Configuration	Initial Openings
		<ul style="list-style-type: none"> <li>• Orifice A-T initial opening <math>&lt; 0</math></li> <li>• Orifice B-T initial opening <math>&gt; 0</math></li> </ul>
8		<p>Orifices P-A and P-B are open, while orifices A-T and B-T are overlapped:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening <math>&gt; 0</math></li> <li>• Orifice P-B initial opening <math>&gt; 0</math></li> <li>• Orifice A-T initial opening <math>&lt; 0</math></li> <li>• Orifice B-T initial opening <math>&lt; 0</math></li> </ul>
9		<p>Orifice P-A is initially open, while all three remaining orifices are overlapped:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening <math>&gt; 0</math></li> <li>• Orifice P-B initial opening <math>&lt; 0</math></li> <li>• Orifice A-T initial opening <math>&lt; 0</math></li> <li>• Orifice B-T initial opening <math>&lt; 0</math></li> </ul>
10		<p>Orifice P-B is initially open, while all three remaining orifices are overlapped:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening <math>&lt; 0</math></li> <li>• Orifice P-B initial opening <math>&gt; 0</math></li> <li>• Orifice A-T initial opening <math>&lt; 0</math></li> <li>• Orifice B-T initial opening <math>&lt; 0</math></li> </ul>
11		<p>Orifices P-B and B-T are open, while orifices P-A and A-T are overlapped:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening <math>&lt; 0</math></li> <li>• Orifice P-B initial opening <math>&gt; 0</math></li> <li>• Orifice A-T initial opening <math>&lt; 0</math></li> <li>• Orifice B-T initial opening <math>&gt; 0</math></li> </ul>

The other nine configurations are covered by the remaining 4-way directional valve blocks (A through K), as shown in the next table.



**Other 4-Way Directional Valve Blocks**

Block Name	Configuration	Description
4-Way Directional Valve A		<p>Contains two additional normally-open, sequentially-located orifices. Valve displacement to the left or to the right closes the path to tank.</p>
4-Way Directional Valve B		<p>Ports P and A are permanently connected through fixed orifice. Orifices P-B and B-T are initially open (underlapped).</p>
4-Way Directional Valve C		<p>Ports P and B are permanently connected through fixed orifice. Orifices P-A, A-T and B-T are initially open (underlapped).</p>
4-Way Directional Valve D		<p>Two orifices are installed in the P-A link. Port A never connects to port T.</p>
4-Way Directional Valve E		<p>Two orifices are installed in the P-B link. Port B never connects to port T.</p>
4-Way Directional Valve F		<p>Two parallel orifices in the P-A arm and two series orifices in the A-T arm.</p>

Block Name	Configuration	Description
4-Way Directional Valve G		Two parallel orifices in the P-B arm and two series orifices in the B-T arm.
4-Way Directional Valve H		Two parallel orifices in the P-B arm and two series orifices in the P-T arm.
4-Way Directional Valve K		Two parallel orifices in the P-A arm and two series orifices in the P-T arm.

## Building Blocks and How to Use Them

The Directional Valves library offers several prebuilt directional valve models. As indicated in their descriptions, all of them are symmetrical, continuous valves. In other words, the control member in 2-way, 3-way, 4-way, and 6-way valves can assume any position, controlled by the physical signal port S. The valves are symmetrical in that all the orifices the valve is built of are of the same type and size. The only possible difference between orifices is the orifice initial opening.

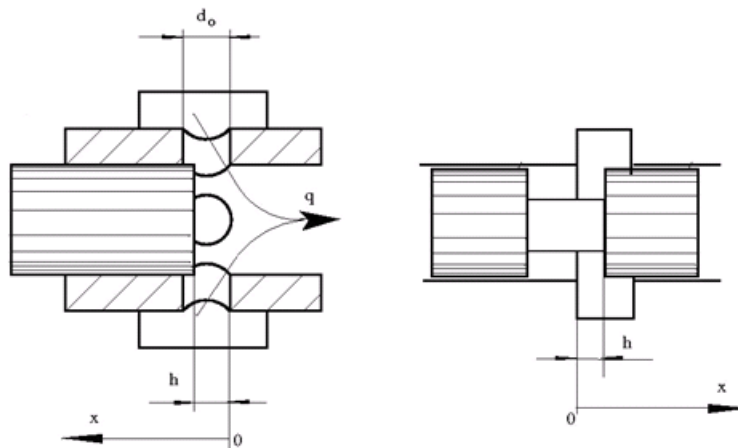
These configurations cover a substantial portion of real valves, but the directional valves family is so diverse as to make it practically impossible to have a library model for every member. Instead, SimHydraulics libraries offer a set of building blocks that is comprehensive enough to build a model for any real configuration. This section describes the rules of building a custom model of a directional valve.

All directional valve models are built of variable orifices. In SimHydraulics libraries, the following variable orifice models are available:

- Annular Orifice
- Orifice with Variable Area Round Holes

- Orifice with Variable Area Slot
- Variable Orifice
- Ball Valve
- Ball Valve with Conical Seat
- Needle Valve
- Poppet Valve

To simplify the way variable orifices are combined in a model, their instantaneous opening is computed in the same way for all types of orifices. The orifice opening is always computed in the direction the spool, or any other control member, opens the orifice. In other words, positive value of the opening corresponds to open orifice, while negative value denotes overlapped, or closed, orifice. The origin always corresponds to zero-lap position, when the edge of the control member coincides with the edge of the orifice. In the illustration below, origins are marked with 0 for the orifice with variable area round holes (schematic on the left) and for the orifice with variable slot (schematic on the right). The  $x$  arrow denotes the direction in which orifice opening is measured in both cases.



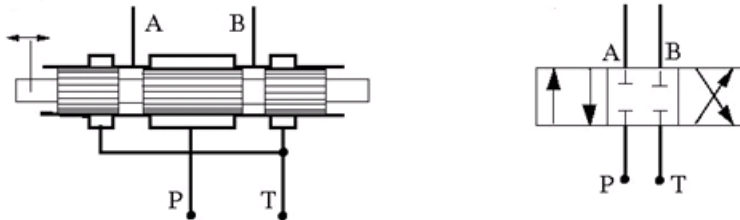
The instantaneous value of the orifice opening is determined as

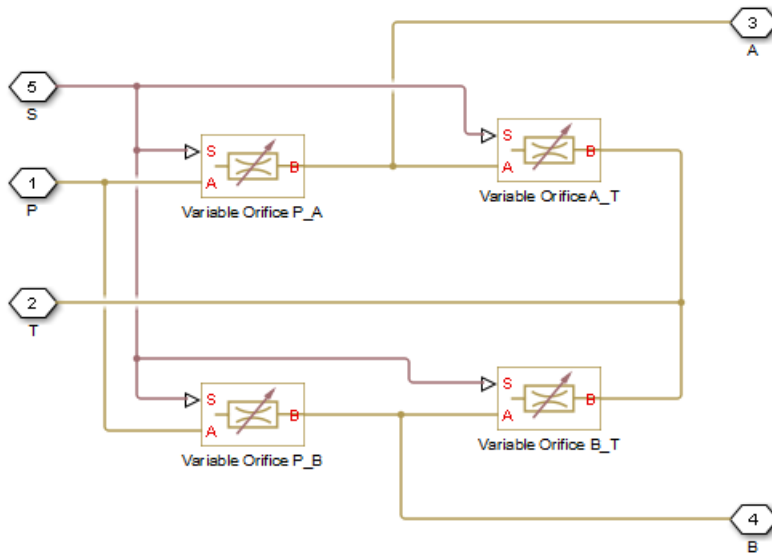
$$h = x_0 + x_{sp} \cdot OR$$

where

$h$	Instantaneous orifice opening.
$x_0$	Initial opening. The initial opening value is positive for initially open (underlapped) orifices and negative for overlapped orifices.
$x_{sp}$	Spool (or other control member) displacement from initial position, which controls the orifice.
$or$	Orifice orientation indicator. The variable assumes +1 value if a spool displacement in the globally assigned positive direction opens the orifice, and -1 if positive motion decreases the opening.

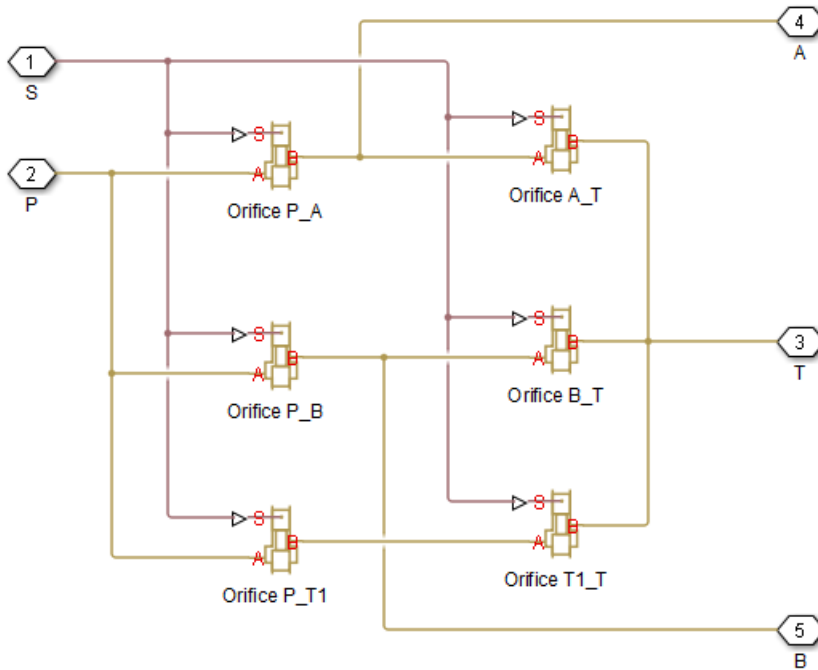
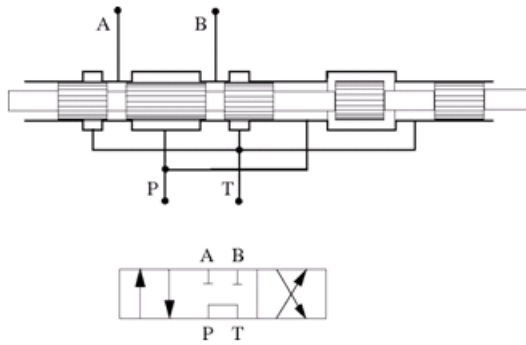
The number of variable orifices and the way they are connected are determined by the valve design. Usually, the model of a valve mimics the physical layout of a real valve. The illustration below shows an example of a 4-way valve, its symbol, and an equivalent circuit of its SimHydraulics model.





The 4-way valve in its simplest form is built of four variable orifices. In the equivalent circuit, they are named P\_A, P\_B, A\_T, and B\_T. The Variable Orifice block, which is the most generic model of a variable orifice in the SimHydraulics libraries, is used in this particular example. You can use any other variable orifice blocks if the real valve design employs a configuration backed by a stock model, such as an orifice with round holes or rectangular slots, poppet, ball, or needle. In general, all orifices in the model can be simulated with different blocks or with the same block, but with different way of parameterization. For instance, two orifices can be represented by their pressure-flow characteristics, while two others can be simulated with the table-specified area variation option (for details, see the **Variable Orifice** block reference page).

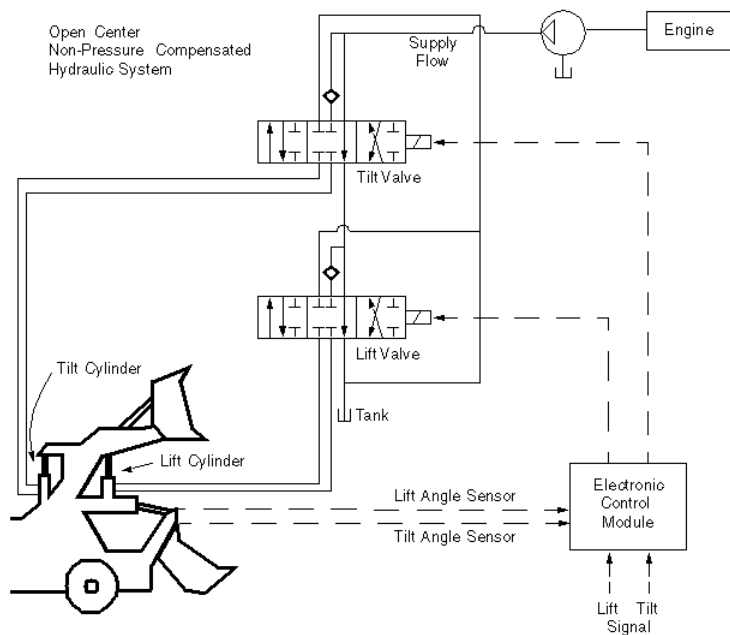
The next example shows another configuration of a 4-way directional valve. This valve unloads the pump in neutral position and requires six variable orifice blocks. The Orifice with Variable Area Round Holes blocks have been used as a variable orifice in this model. Port T1 corresponds to an intermediate point between ports P and T.



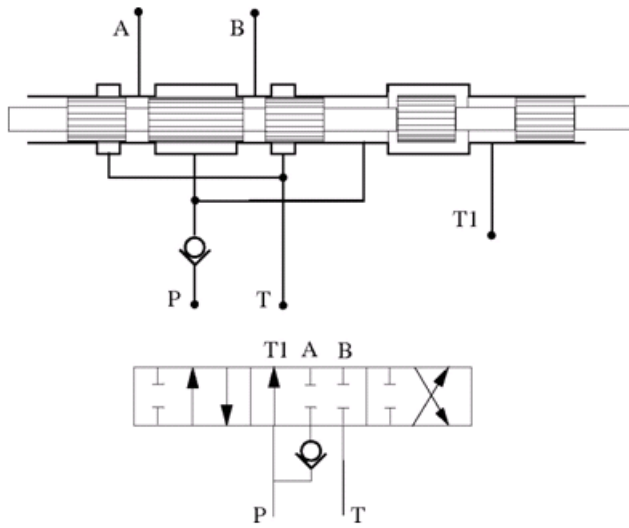
## Build a Custom Directional Valve

Finally, let us consider a more complex directional valve example. The figure below shows basic elements of a front loader hydraulic system. Both the lift and the tilt

cylinders are controlled by custom 3-position, 5-way valves, developed for this particular application. The valves are designed in such a way that the pump delivery is diverted to tank (unloaded) if both cylinders are commanded to be in neutral position. The pump is disconnected from the tank if either of the two control valves is shifted from neutral position.

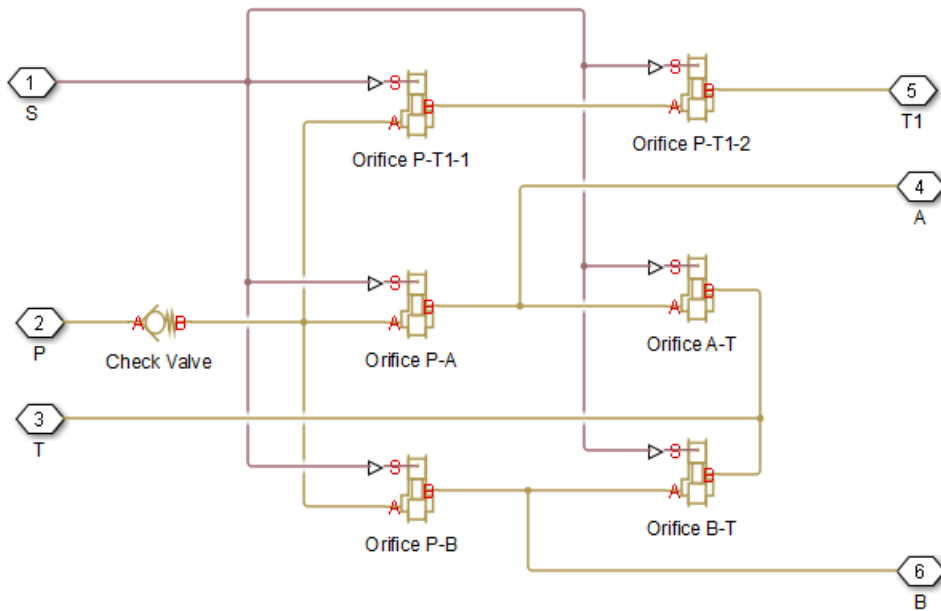


To develop a model, the physical version of the valve must be created first. The following illustration shows one of the possible configurations of the valve.



The SimHydraulics model, shown below, is an exact copy of the physical valve configuration.





All the orifices in the model are closed (overlapped) in valve neutral position, except orifices P\_T1\_1 and P\_T1\_2. These two orifices should be set open to an extent that allows pump delivery to be discharged at low pressure.

## Modeling Low-Pressure Fluid Transportation Systems

### In this section...

“How Fluid Transportation Systems Differ from Power and Control Systems” on page 2-44

“Available Blocks and How to Use Them ” on page 2-46

“Low-Pressure Fluid Transportation System” on page 2-48

### How Fluid Transportation Systems Differ from Power and Control Systems

In hydraulics, the steady uniform flow in a component with one entrance and one exit is characterized by the following energy equation

$$-\frac{\dot{W}_s}{\dot{m}g} = \frac{V_2^2 - V_1^2}{2g} + \frac{p_2 - p_1}{\rho g} + z_2 - z_1 + h_L$$

where

$\dot{W}_s$	Work rate performed by fluid
$\dot{m}$	Mass flow rate
$V_2$	Fluid velocity at the exit
$V_1$	Fluid velocity at the entrance
$p_1, p_2$	Static pressure at the entrance and the exit, respectively
$g$	Gravity acceleration
$\rho$	Fluid density
$z_1, z_2$	Elevation above a reference plane (datum) at the entrance and the exit, respectively
$h_L$	Hydraulic loss

Subscripts 1 and 2 refer to the entrance and exit, respectively. All the terms in Equation 2-1 have dimensions of height and are named kinematic head, piezometric head, geometric head, and loss head, respectively. For a variety of reasons, analysis of hydraulic power and control systems is performed with respect to pressures, rather than to heads, and Equation 2-1 for a typical passive component is presented in the form

$$\frac{\rho}{2} V_1^2 + p_1 + \rho g z_1 = \frac{\rho}{2} V_2^2 + p_2 + \rho g z_2 + p_L$$

where

$V_1, p_1, z_1$	Velocity, static pressure, and elevation at the entrance, respectively
$V_2, p_2, z_2$	Velocity, static pressure, and elevation at the exit, respectively
$p_L$	Pressure loss

Term  $\frac{\rho}{2} V^2$  is frequently referred to as kinematic, or dynamic, pressure, and  $\rho g z$  as piezometric pressure. Dynamic pressure terms are usually neglected because they are very small, and Equation 2-2 takes the form

$$p_1 + \rho g z_1 = p_2 + \rho g z_2 + p_L$$

The size of a typical power and control system is usually small and rarely exceeds 1.5 – 2 m. To add to this, these systems operate at pressures in the range 50 – 300 bar. Therefore,  $\rho g z$  terms are negligibly small compared to static pressures. As a result, SimHydraulics components (with the exception of the ones designed specifically for low-pressure simulation, described in “Available Blocks and How to Use Them ” on page 2-46) have been developed with respect to static pressures, with the following equations

$$p = p_L = f(q)$$

$$q = f(p_1, p_2)$$

where

$p$	Pressure difference between component ports
$q$	Flow rate through the component

Fluid transportation systems usually operate at low pressures (about 2-4 bar), and the difference in component elevation with respect to reference plane can be very large. Therefore, geometrical head becomes an essential part of the energy balance and must be accounted for. In other words, the low-pressure fluid transportation systems must

be simulated with respect to piezometric pressures  $p_{pz} = p + \rho gz$ , rather than static pressures. This requirement is reflected in the component equations

$$p = p_L = f(q, z_1, z_2)$$

$$q = f(p_1, p_2, z_1, z_2)$$

Equations in the form Equation 2-5 must be applied to describe a hydraulic component with significant difference between port elevations. In hydraulic systems, there is only one type of such components: hydraulic pipes. The models of pipes intended to be used in low pressure systems must account for difference in elevation of their ports. The dimensions of the rest of the components are too small to contribute noticeably to energy balance, and their models can be built with the constant elevation assumption, like all the other SimHydraulics blocks. To sum it up:

- You can build models of low-pressure systems with difference in elevations of their components using regular SimHydraulics blocks, with the exception of the pipes. Use low-pressure pipes, described in “Available Blocks and How to Use Them ” on page 2-46 .
- When modeling low-pressure systems, you must use low-pressure pipe blocks to connect all nodes with difference in elevation, because these are the only blocks that provide information about the vertical locations of the system parts. Nodes connected with any other blocks, such as valves, orifices, actuators, and so on, will be treated as if they have the same elevation.

### Available Blocks and How to Use Them

When modeling low-pressure hydraulic systems, use the pipe blocks from the Low-Pressure Blocks library instead of the regular pipe blocks. These blocks account for the port elevation above reference plane and differ in the extent of idealization, just like their high-pressure counterparts:

- Resistive Pipe LP — Models hydraulic pipe with circular and noncircular cross sections and accounts for friction loss only, similar to the Resistive Tube block, available in the Simscape Foundation library.
- Resistive Pipe LP with Variable Elevation — Models hydraulic pipe with circular and noncircular cross sections and accounts for friction losses and variable port elevations. Use this block for low-pressure system simulation in which the pipe ends change their positions with respect to the reference plane.

- Hydraulic Pipe LP — Models hydraulic pipe with circular and noncircular cross sections and accounts for friction loss along the pipe length and for fluid compressibility, similar to the Hydraulic Pipeline block in the Pipelines library.
- Hydraulic Pipe LP with Variable Elevation — Models hydraulic pipe with circular and noncircular cross sections and accounts for friction loss along the pipe length and for fluid compressibility, as well as variable port elevations. Use this block for low-pressure system simulation in which the pipe ends change their positions with respect to the reference plane.
- Segmented Pipe LP — Models circular hydraulic pipe and accounts for friction loss, fluid compressibility, and fluid inertia, similar to the Segmented Pipe block in the Pipelines library.

Use these low-pressure pipe blocks to connect all hydraulic nodes in your model with difference in elevation, because these are the only blocks that provide information about the vertical location of the ports. Nodes connected with any other blocks, such as valves, orifices, actuators, and so on, will be treated as if they have the same elevation.

The additional models of pressurized tanks available for low-pressure system simulation include:

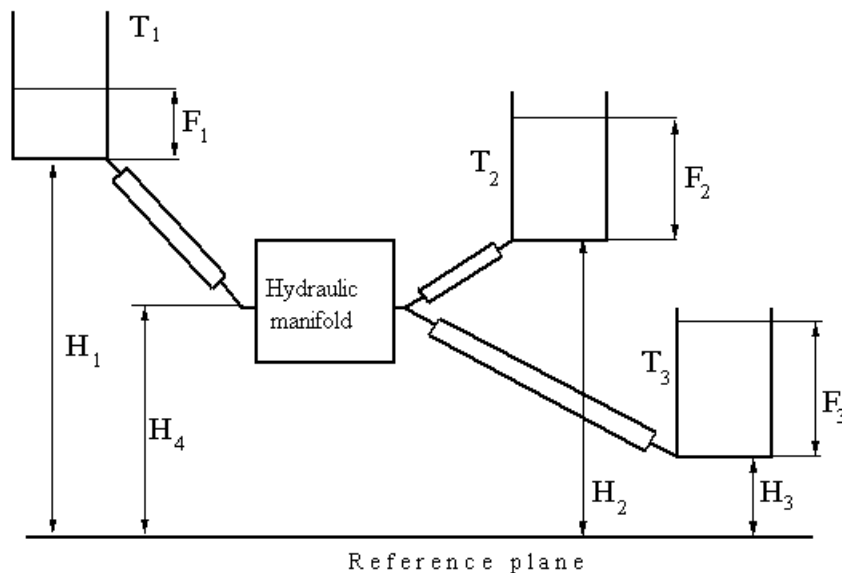
- Constant Head Tank — Represents a pressurized hydraulic reservoir, in which fluid is stored under a specified pressure. The size of the tank is assumed to be large enough to neglect the pressurization and fluid level change due to fluid volume. The block accounts for the fluid level elevation with respect to the tank bottom, as well as for pressure loss in the connecting pipe that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficient. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V.
- Variable Head Tank — Represents a pressurized hydraulic reservoir, in which fluid is stored under a specified pressure. The pressurization remains constant regardless of volume change. The block accounts for the fluid level change caused by the volume variation, as well as for pressure loss in the connecting pipe that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficient. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V.
- Variable Head Two-Arm Tank — Represents a two-arm pressurized tank, in which fluid is stored under a specified pressure. The pressurization remains constant regardless of volume change. The block accounts for the fluid level change caused by the volume variation, as well as for pressure loss in the connecting pipes that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the

pressure loss coefficient at each outlet. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V.

- Variable Head Three-Arm Tank — Represents a three-arm pressurized tank, in which fluid is stored under a specified pressure. The pressurization remains constant regardless of volume change. The block accounts for the fluid level change caused by the volume variation, as well as for pressure loss in the connecting pipes that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficient at each outlet. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V.

### Low-Pressure Fluid Transportation System

The following illustration shows a simple system consisting of three tanks whose bottom surfaces are located at heights  $H_1$ ,  $H_2$ , and  $H_3$ , respectively, from the reference plane. The tanks are connected by pipes to a hydraulic manifold, which may contain any hydraulic elements, such as valves, orifices, pumps, accumulators, other pipes, and so on, but these elements have one feature in common – their elevations are all the same and equal to  $H_4$ .



The models of tanks account for the fluid level heights  $F_1$ ,  $F_2$ , and  $F_3$ , respectively, and represent pressure at their bottoms as

$$p_i = \rho g F_i \quad \text{for } i = 1, 2, 3$$

The components inside the manifold can be simulated with regular SimHydraulics blocks, like you would use for hydraulic power and control systems simulation. The pipes must be simulated with one of the low-pressure pipe models: Resistive Pipe LP, Hydraulic Pipe LP, or Segmented Pipe LP, depending on the required extent of idealization. Use the Constant Head Tank or Variable Head Tank blocks to simulate the tanks. For details of implementation, see the Water Supply System and the Fluid Transportation System with Three Tanks examples.

