Hydraulic Library

Version 4.2 - September 2004

IMAGINE ®

Chapter 1: Tutorial examples

1.1 Introduction

The **AMESim** **Hydraulic** library consists of:

- A collection of commonly used hydraulic components such as pumps, motors, orifices, etc. including special valves.
- Submodels of pipes and hoses.
- Sources of pressure and flow rate.
- Sensors of pressure and flow rate.
- A collection of fluid properties.

Hydraulic systems in isolation are completely useless! It is necessary to do something with the fluid and also to control the process. This means that the library must be compatible with other **AMESim** libraries. The following libraries are frequently used with the **Hydraulic** library:

Mechanical library

Used in fluid power application when hydraulic power is translated into mechanical power.

Signal, Control and Observer library

Used to control the hydraulic system.

Hydraulic component design library

Used to build specialist components from very basic hydraulic and mechanical elements.

Hydraulic resistance library

This is a collection of submodels of bends, tee-junctions, elbows etc. It is used typically in low pressure applications such as cooling and lubrication systems.

Note:

- It is possible to use more than one fluid in the **Hydraulic** library. This is important because you can model combined cooling and lubrication systems of a library.
- The hydraulic library assumes a uniform temperature throughout the system. If thermal effects are considered to be important, the **Thermal Hydraulic** and **Thermal Hydraulic Component Design** libraries should be used.
- There are models of cavitation and air release in the hydraulic library. Note also there is a special **two-phase flow** library. A typical application for this is air conditioning systems.

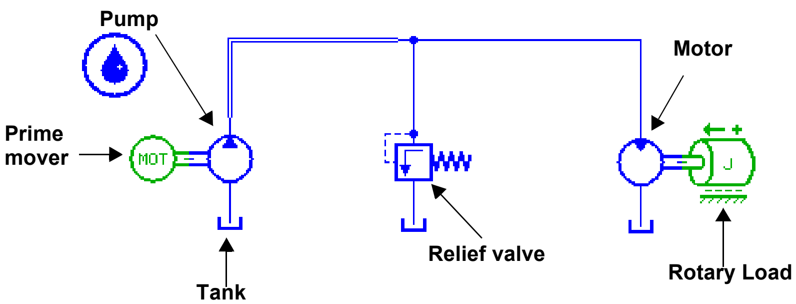
Chapter 1 of the manual consists of a collection of tutorial examples. We strongly recommend that you do these tutorial examples. They assume you have a basic level of experience using **AMESim**. As an absolute minimum you should have done the examples in Chapter 3 of the **AMESim** manual and the first example of Chapter 5 which describes how to do a batch run.

1.2 Example 1: A simple hydraulic system

Objectives

- Construct a very simple hydraulic system
- Introduce the simplest pipe/hose submodels
- Interpret the results with a special reference to air release and cavitation

Figure 1.1: A very simple hydraulic system

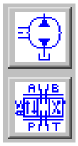


In this exercise you will construct the system shown in Figure 1.1. This is perhaps the simplest possible meaningful hydraulic system. It is built partly from components from the *Hydraulic* category (which are normally blue) and partly from the *Mechanical* category.

The hydraulic part is built up from standard symbols used for hydraulic systems.

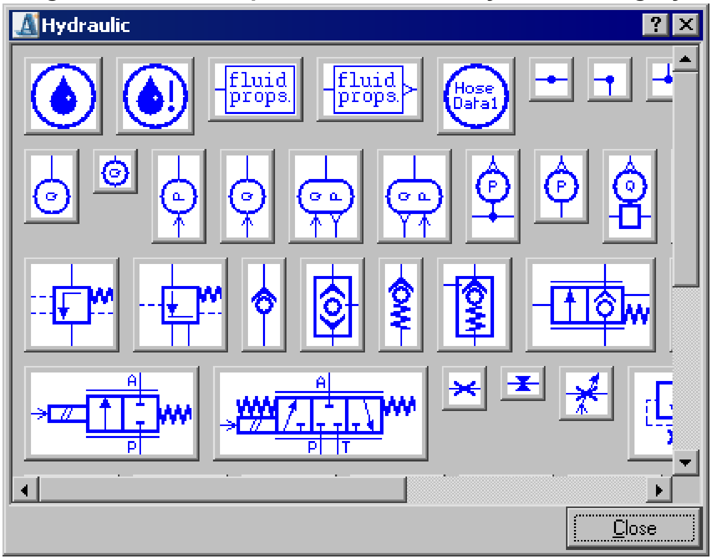
The prime mover supplies power to the pump, which draws hydraulic fluid from a tank. This fluid is supplied under pressure to a hydraulic motor, which drives a rotary load. A relief valve opens when the pressure reaches a certain value. The output from the motor and the relief valve returns to the tank. The diagram shows three tanks but it is quite likely that a single tank is employed.

There are two categories in the *Hydraulic* library. These have blue as the standard color. If you do not have this categories displayed, check the path list in the *Options* menu.



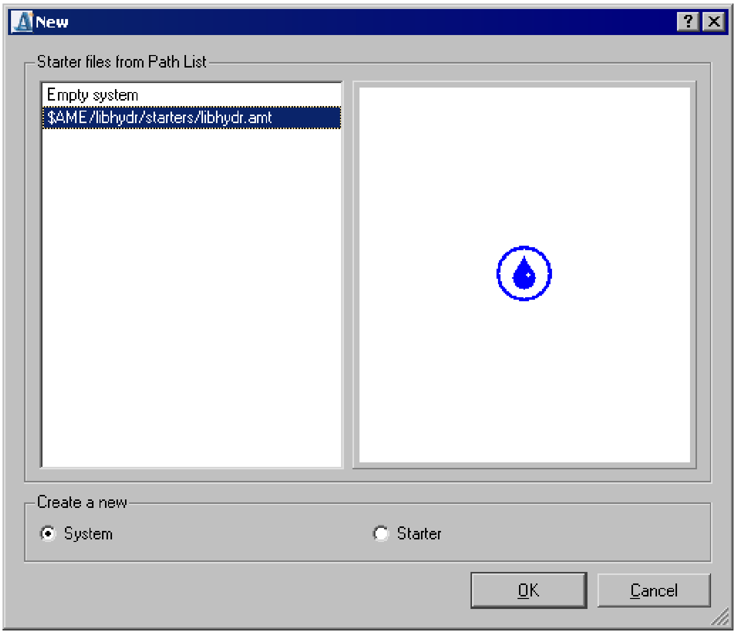
The first category contains general hydraulic components. The seconds contains special valves.The hydraulic components used in the model you will build can all be found in the first of these *Hydraulic* categories. If you click on this category icon, you will have the dialog box shown in Figure 1.2. First look at the components available in this library. Display the title of some components by moving the pointer over the icons.

Figure 1.2: The components in the first hydraulic category.



Step 1: Use File ▶ New... to produce the following dialog box.

Figure 1.3: The hydraulic starter system.



Select the hydraulic starter circuit *libhydr.amt* and then click on *OK*. A new system with a fluid properties icon in the top left corner of the sketch will be created.

You could also have clicked on the *New* icon in the tool bar but if you do this you will have to add the fluid properties icon yourself.

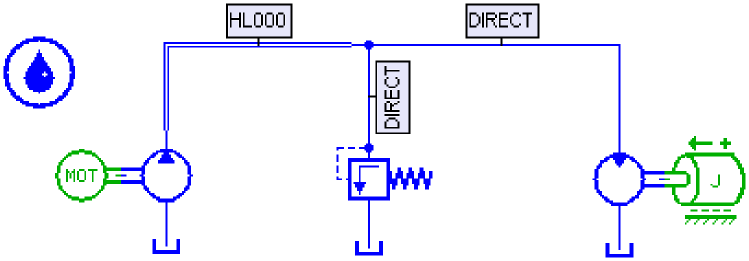
Step 2: Construct the rest of the system and assign submodels

1. Construct the system with the components as shown in Figure 1.1.
2. Save it as *hydraulic1*.
3. Go to *Submodel* mode.

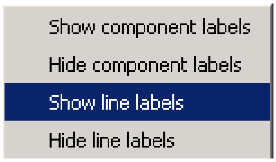
   Notice that the drop, the prime mover, the node and the pipes are not of normal appearance because they do not have submodels associated with them. The easiest way to proceed is:

4. Click on the *Premier submodel* button which is situated in the horizontal menu bar.

Figure 1.4: The line submodels.



5. Press the mouse right-button.
6. Select “Show line labels” in the label menu.



You get something like Figure 1.4. It is possible that your system may have *HL000* associated with one of the other line runs. These minor variations are dependent on the order in which you constructed the lines. They will not influence the simulation results.

An important feature to notice is that a line run has a special submodel (*HL000*) which is not a direct connection. To emphasize this point the line run has a special appearance.

Remember the submodel *DIRECT* does nothing at all. It is as if the ports at the end of the line were connected directly together.

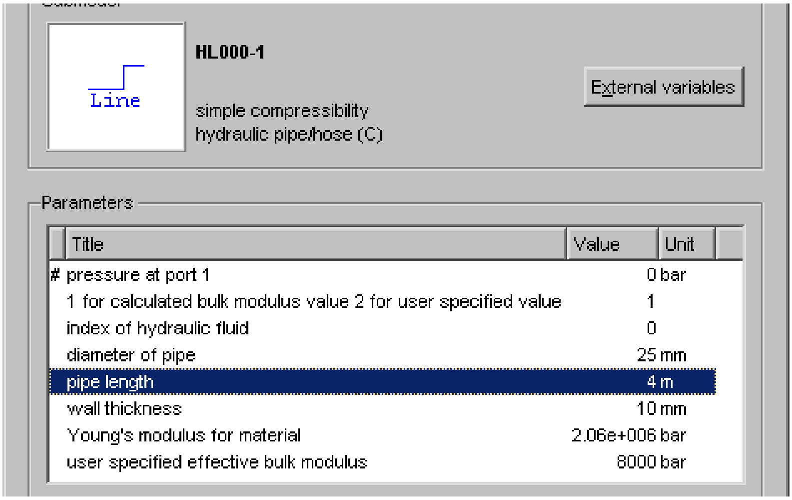
In contrast, *HL000* computes the net flow into the pipe and uses this to determine the time derivative of pressure. If the net flow into the pipe is positive, pressure increases with time. If it is negative, it decreases with time. The pressure created by *HL000* is conveyed to the relief valve inlet. The motor inlet is conveyed by the node and submodel *DIRECT*.

Step 3: Set parameters

1. Change to *Parameter* mode.
2. Set the following parameters and leave the others at their default values:

Submodel	Title	Value
HL000	pipe length [m]	4
RL00	coefficient of viscous friction [Nm/(rev/min)]	0.02

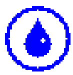
Figure 1.5: Setting the line submodel HL000 parameters.



3. To display the parameters of a line submodel click the left mouse button with the pointer on or near the appropriate line run.

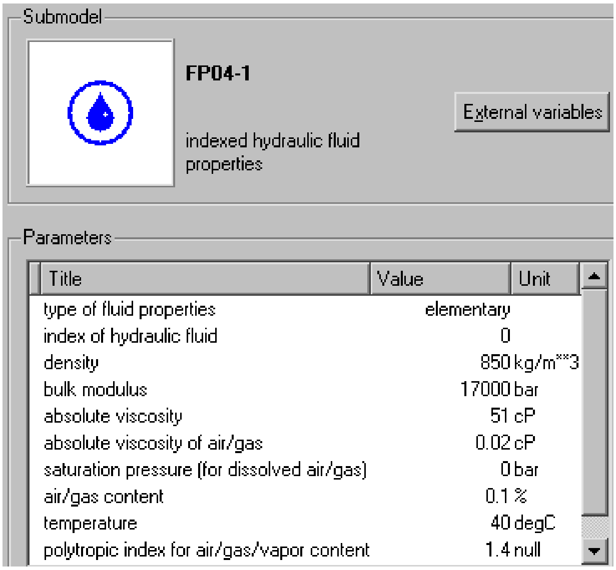
   Part of the dialog box for *HL000* is shown in Figure 1.5. The compressibility of the oil and the expansion of the pipe or hose with pressure are taken into account together with the pipe volume. *HL000* normally requires the bulk modulus of the hydraulic fluid and pipe wall thickness together with the Young's modulus of the wall material. From these values an effective bulk modulus of the combined fluid and pipe walls can be calculated. The effective bulk modulus of a hose is normally very much less than that of a rigid steel pipe.

4. Click on the fluid icon *FP04* in the sketch.

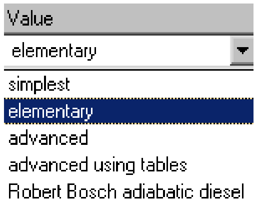
   

   A new dialog box as shown in Figure 1.6 is displayed. This shows you the properties of the hydraulic fluid. Currently they are at their default values and the absolute viscosity, bulk modulus, air/gas content and temperature are given in **common units**.

Figure 1.6: Parameter for fluid properties submodel FP04.



Note that the first item in the list is an enumeration integer parameter. A collection of properties of varying complexity are available but for this exercise *elementary* is satisfactory.



5. Click on OK.

Step 4: Run a simulation

1. Go to *Run* mode and do a simulation run.

   The default values in the *Run Parameters* dialog box are suitable for this example.

2. Click on the *Start run* button.

3. Click on the pump component to produce the dialog box shown in Figure 1.7.

   Some variables such as a pressure have no direction associated with them. A (gauge) pressure of -0.1 bar indicates that the pressure is below atmospheric. In contrast other variables, such as flow rate, do have a direction associated with them. A flow rate of -6 L/min indicates that the flow is in the opposite direction to some agreed standard direction.

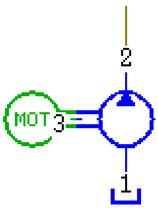
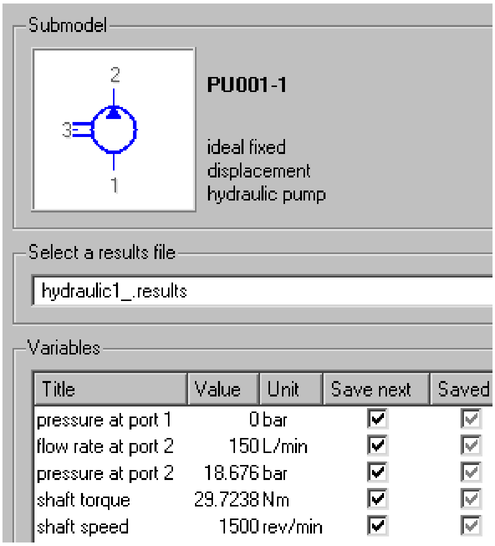
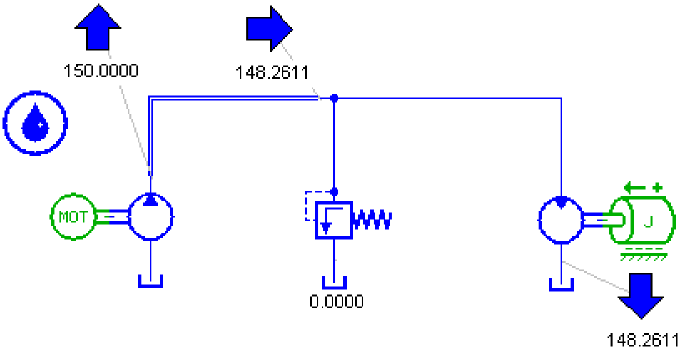


Figure 1.7: The Variable List for PU001.



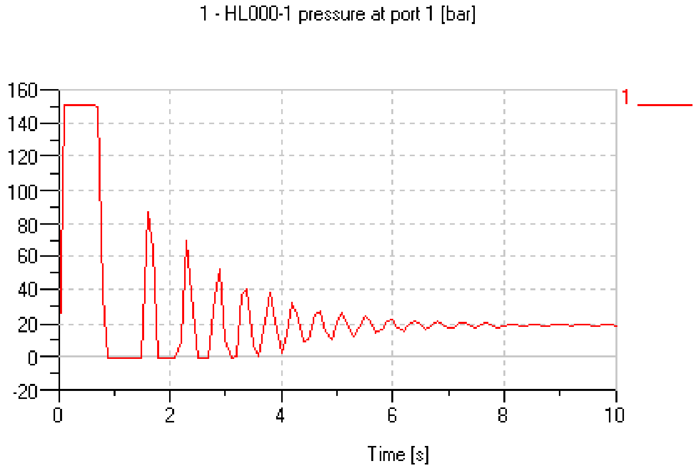
Note that you can use the **Replay facility** to give you a global picture of the results. Figure 1.8 also shows the flow rates in L/min at a time of 10 seconds.

Figure 1.8: Flow rates displayed in replay.



4. To plot a variable associated with a line submodel, click on or near the corresponding line run.
5. Plot *pressure at port 1* for *HL000*.

Figure 1.9: The pressure in the hydraulic pipe.



Notice how the pressure goes up to just over the relief valve setting (150 bar). During this time the load speeds up rapidly and actually 'over-speeds'. At this point the motor is demanding more hydraulic flow than the pump can supply. The result is that the pressure must drop and the relief valve closes. The pressure continues to drop and falls below zero bar gauge. However, pressure is not like voltage or force. We cannot have a pressure of -100 bar. **The absolute zero of pressure is about -1.013 bar gauge.** It is time to introduce two terms.

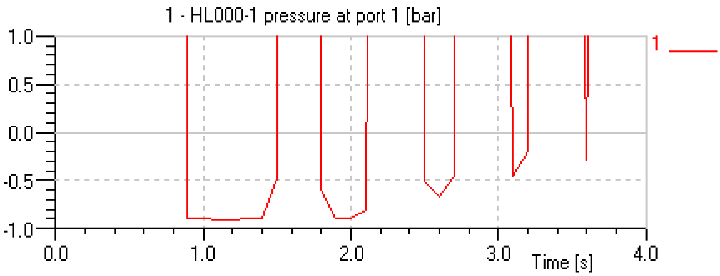
Cavitation and air release

When pressure falls to very low levels, two things can happen:

- Air previously dissolved in the fluid begins to form air bubbles.
- The pressure reaches the saturated vapor pressure of the liquid and bubbles of vapor appear.

These phenomena are known as air release and cavitation respectively. They can cause serious damage. Using the *Zoom* facility, the graph gives a better view of the lower pressure values:

Figure 1.10: Low pressure in the hydraulic pipe.



All **AMESim** submodels have hydraulic pressure in *bar* gauge. The low pressure shown in Figure 1.10 : Low pressure in the hydraulic pipe. is caused by the load speed exceeding its steady state or equilibrium value and it is a highly undesirable behavior as it can result in damage to the real system.

In reality the starting values we have given for the pipe pressure and load speed are not very realistic and the prime mover would start from rest or some valve would be used to regulate the flow to the motor. However, hydrostatic transmission systems like this often do suffer badly from cavitation and air release problems.

Note that all **AMESim** submodels have hydraulic volumetric flow rate in L/min. There are two possible interpretations of this flow rate:

- The flow rate is measured at the local current hydraulic pressure, or
- The flow rate is measured at a reference pressure.

AMESim adopts the second alternative with a reference pressure of 0 *bar* gauge. This means that the volumetric flow rate is always directly proportional to the mass flow rate. In most situations the difference between the two flow rates is negligible. However, there are three situations when there is a significant difference:

1. There is a very large air content; the pressure drops below the saturation pressure for air in the liquid and air bubbles are formed in the liquid.
2. The pressure drops to the level of the saturated vapor pressure of the liquid and cavities of vapor form.
3. Extremely high variations in pressure occur such as in certain types of fuel injection systems.

The first situation is called *air release* and the second *cavitation*. If there is cavitation or significant air release at the inlet to a pump, the flow rate according to the first definition will not be reduced but with the approach adopted by **AMESim** it will be significantly reduced.

The properties of hydraulic fluids vary a great deal. Modeling them is a very specialist process and the model can be extremely simple or highly complex. The run times are greatly influenced by this level of complexity.

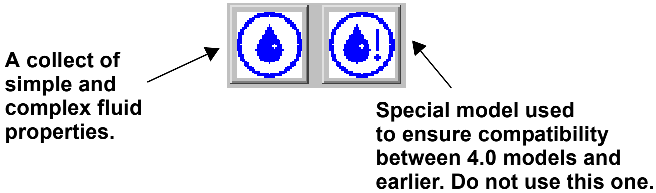
1.3 Example 2: Using more complex hydraulic properties

Objectives:

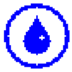
- Use more complex models of fluid properties.
- See how air content changes the performance of the system.

In the *Hydraulic* category two special components can modify the fluid properties:

Figure 1.11: The two fluid properties icons.



In **AMESim** always use this fluid properties icon. It is associated with one submodel: *FP04*. The other icon and its submodel is there only for backward compatibility.

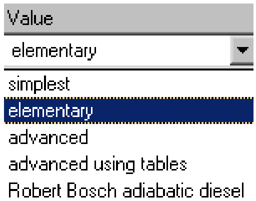


This is an example of a component without ports. We cannot connect this icon to any other.

There are two important thing about *FP04*.

1. It has an integer parameter *index of hydraulic fluid* that is in the range 0 to 100 inclusive. This arrangement means that it is possible to have more than one fluid in an **AMESim** system.

2. The characteristics of the fluid properties are determined by its parameters. One of which is an enumeration integer parameter. There are 5 possibilities:



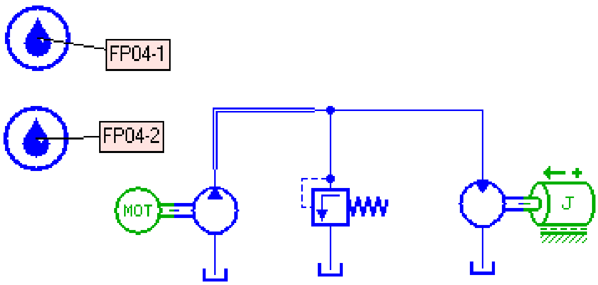
- **elementary** This is the default and features a constant liquid bulk modulus with absolute viscosity. The treatment of fluid properties under air release and cavitation is done.
- **simplest** This has a constant absolute viscosity. The bulk modulus is constant above the gas saturation pressure and is 1/1000 of this value below the gas saturation pressure. This model is very old but is still used by some **AMESim** users.It is likely to give the fastest runs.
- **advanced** This gives you access to some cavitation parameters not accessible in the elementary properties.
- **advanced using tables** This is like the advanced option but you install tables of data to give variation of bulk modulus and absolute viscosity with pressure and temperature.
- **Robert Bosch adiabatic diesel** These properties are provided by Robert Bosch GmbH and comprise a number of common types of diesel fuel.

Using one of the special fluids

Step 1: Use the Advanced fluid properties.

1. Return to the first example of this manual, add another fluid properties icon.
2. Use *Premier submodel* and go to *Parameter* mode. Your sketch should look like this.

Figure 1.12: The sketch with two instances of FP04.



3. Look at parameters of *FP04-2*. Change the enumeration integer parameter to *advanced*. The *Change Parameters* list should now look like this:

Figure 1.13: The advanced fluid properties.

Parameters

Title	Value	Unit
type of fluid properties	advanced	
index of hydraulic fluid	0	
density	850	kg/m**3
bulk modulus	17000	bar
absolute viscosity	51	cP
absolute viscosity of air/gas	0.02	cP
saturation pressure (for dissolved air/gas)	0	bar
air/gas content	0.1	%
temperature	40	degC
polytropic index for air/gas/vapor content	1.4	null
(advanced user) high saturated vapor pressure	-0.5	bar
(advanced user) low saturated vapor pressure	-0.6	bar
(advanced user) absolute viscosity of vapor	0.02	cP
(advanced user) effective molecular mass of vapor	200	null
(advanced user) air/gas density at atmospheric pressure 0 degC	1.2	kg/m**3
name of fluid	unnamed fluid	

Change the *index of hydraulic fluid* in *FP04-2* to 1. This is a number in the range 0 to 100. If you look at the other hydraulic components in the system you will find they have index 0 and hence they will still use the fluid properties of *FP04-1*. We could go into every hydraulic component using this second fluid and set the parameter *index of hydraulic fluid* to 1. This would be extremely tedious with a big system and there is always the possibility of missing one.

Step 2: Set all fluid indices to the same value of 1

The best way to do this is to use the **common parameters** facility.

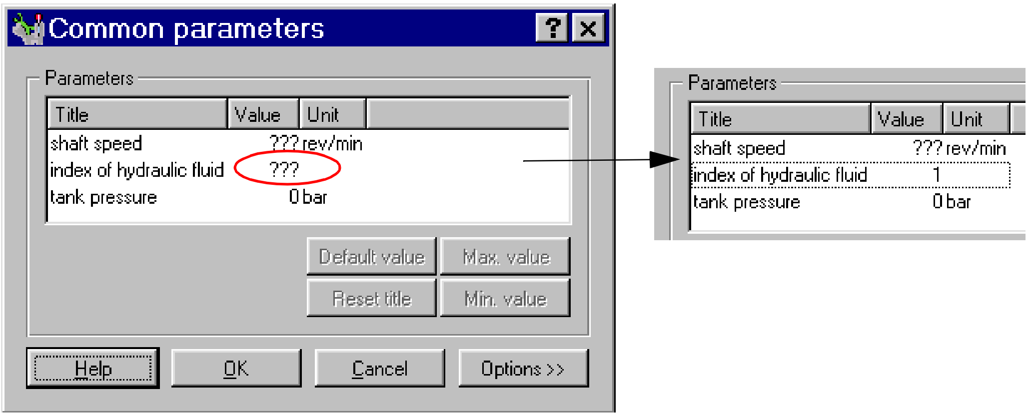
1. Use **Edit ▶Select all**.

   All the system components will be selected, unselect *FP04-1* holding the SHIFT key and clicking on it.

2. Use **Parameters ▶ Common parameters**.

   Figure 1.14 shows the *Common parameters* dialog box. This is a list of common parameters for selected objects. They occur at least twice. Since there are 3 hydraulic tanks and they all have pressures of 0 bar, this value is displayed. There are a number of submodels that have a parameter *index of hydraulic fluid*. In *FP02* the *index of hydraulic fluid* is set to 1 whereas in other submodels its value is 0. The value is displayed as *???*. Similarly the prime mover and rotary load both have a parameter (strictly speaking variable) with title *shaft speed*. Since the two values are different, *???* is displayed.

Figure 1.14: Different values for common parameters



3. Set the parameter *index of hydraulic fluid* to 1. This will change all the parameters in the system except *FP01* (remember we selected *Select all* except *FP01*).

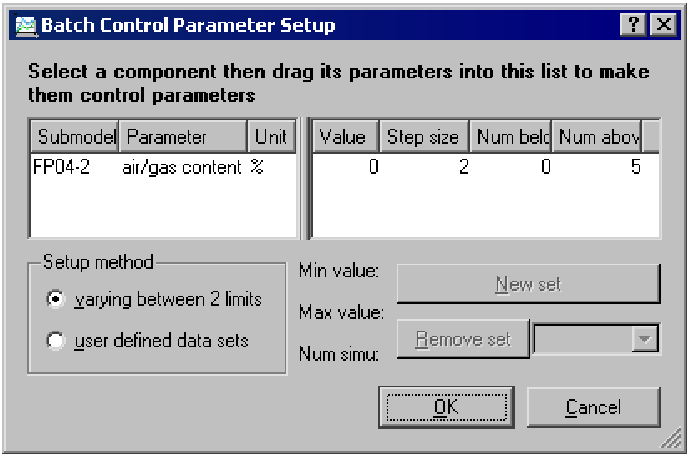
Step 3: Run a simulation and plot some variables

You will probably find the results very much the same as in example 1.

Step 4: Organize a batch run to vary the air content

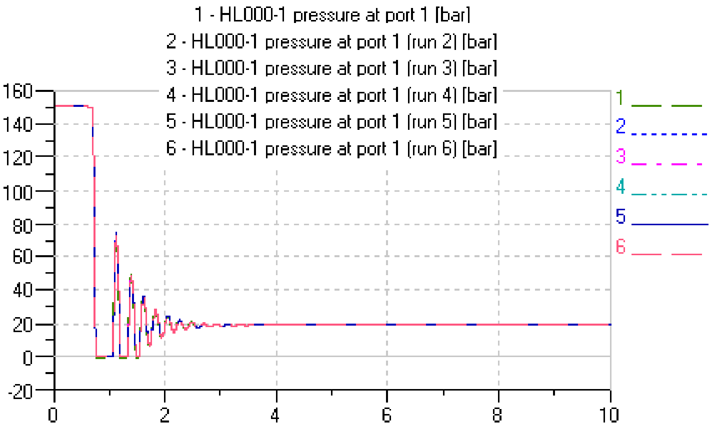
1. In *Parameter* mode use **Parameters ▶ Batch parameters**.
2. Drag and drop the air gas content from *FP04-2* to the *Batch control parameter setup* dialog box.
3. Set up the batch parameters as in Figure 1.15 so that the air content goes from 0% to 10% in steps of 2%.
4. Specify a batch run in the *Run parameters* dialog box and initiate the run.

Figure 1.15: Setting up a batch run varying air content.



5. Plot several graphs of the batch run to compare results with various air contents.

Figure 1.16: Pressure in pipe.

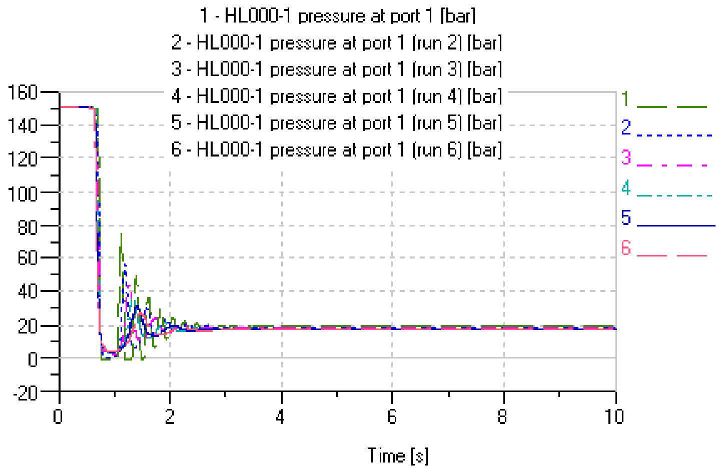


By zooming on the curve in regions where the pressure is below 0 bar you will probably find some, but not a remarkable variation in the results.

6. Change the saturation pressure in *FP04-2* to 400 bar.

7. Repeat the batch run and update your plot.

Figure 1.17: Pressure in pipe with saturation pressure 400 bar.



The variation between the runs is now very pronounced. The dynamic characteristics of the system is completely transformed. A few words of explanation are necessary.

Normally the air content of a hydraulic oil is well below 1%, and 0.1% is typical. It is normally considered good practice to keep the value as low as reasonably possible. However, in a few application, such as lubrication oil in gearboxes, the oil and air are well mixed up and 2.5% is typical and up to about 10% is possible.

A reasonable quantity of air, given time, will completely or partially dissolve in the hydraulic fluid. The lowest pressure at which all the air is dissolved is called the saturation pressure. For very slow systems all the air is dissolved above the saturation pressure and partially dissolved below this pressure. Henry's law gives a reasonable approximation for the fraction of air that is dissolved **in equilibrium**.

Some systems are slow enough to stay very close to this equilibrium position (Figure 1.16). Often classic fluid power systems behave like this. The original saturation pressure is better for the current example.

However, it does take time for the air to dissolve and this time will not be available in fast acting systems. Fuel injection systems are a good example of this. Hence with such systems it may be appropriate to set the saturation pressure artificially high to allow for significant quantities of air to be undissolved at all pressures.

1.4 Example 3: Using more complex line submodels

Objectives:

- Use more complex line submodels.
- Understand the need for a variety of line submodels.
- To understand the importance of setting an **appropriate** line submodel.

The system for this example is the same as for example 2 (Figure 1.12). We will describe the modification of the system to use more complex line submodels and the experiments performed. Finally we give a little of the theory behind the submodels.

Step 1: Change submodels

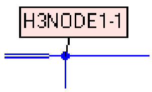
All the submodels in the current system were selected automatically. We will change some of them manually.

1. Go to *Submodel* mode.

   You will now change some line submodels.

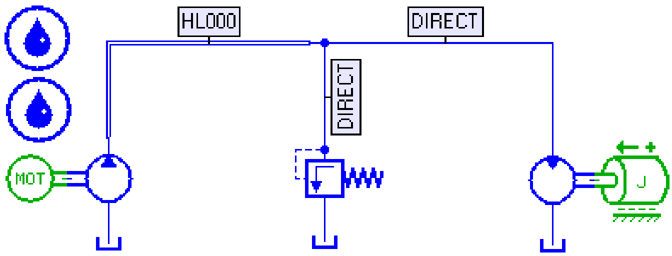
   Before continuing note the following points:

   - The corners in the pipe runs are not physical but diagrammatic.
   - There are three hydraulic pipes and they meet at a point which physically will be a tee-junction.
   - This tee-junction in the sketch is described as a 3 port node and it has the submodel *H3NODE1*. This models the junction as a common pressure with flow rates that give conservation of mass.



   - It is necessary to have a large number of hydraulic pipe submodels.
   - In the present system three submodels are set: *DIRECT*, *DIRECT* and *HL000*.

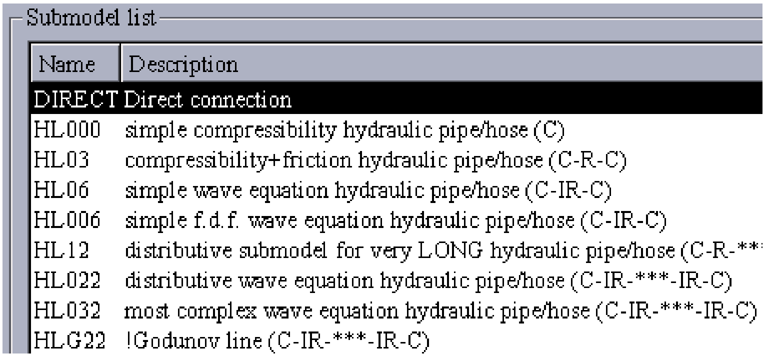
Figure 1.18: The current line submodels.



None of these line submodels take friction into account. We will suppose that the relief valve is close to the node but the pump and the motor are at such distances from the node that the pressure drop along the pipes cannot be ignored. We need to select new pipe submodels that take friction into account for the pipe runs from the pump to the node and from the node to the motor.

2. Click on the line run attached to the pump and select *HL03* in the *Submodel list*.

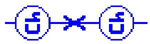
Figure 1.19: The hydraulic line submodels available.



Submodel list

Name	Description
DIRECT	Direct connection
HL000	simple compressibility hydraulic pipe/hose (C)
HL03	compressibility+friction hydraulic pipe/hose (C-R-C)
HL06	simple wave equation hydraulic pipe/hose (C-IR-C)
HL006	simple f.d.f. wave equation hydraulic pipe/hose (C-IR-C)
HL12	distributive submodel for very LONG hydraulic pipe/hose (C-R-**
HL022	distributive wave equation hydraulic pipe/hose (C-IR-***-IR-C)
HL032	most complex wave equation hydraulic pipe/hose (C-IR-***-IR-C)
HLG22	!Godunov line (C-IR-***-IR-C)

Note the brief description of each line submodel. In these descriptions *C* stands for compressibility, *R* for resistance (pipe friction) and *I* for inertia (fluid momentum). *HL000* which we used before takes into account compressibility only. *HL03* takes into account compressibility and friction. It is modeled like two hydraulic compressible volumes with a resistance between them.



Why did we not choose a more complex submodel that also included inertia? We answer this question later in this exercise.

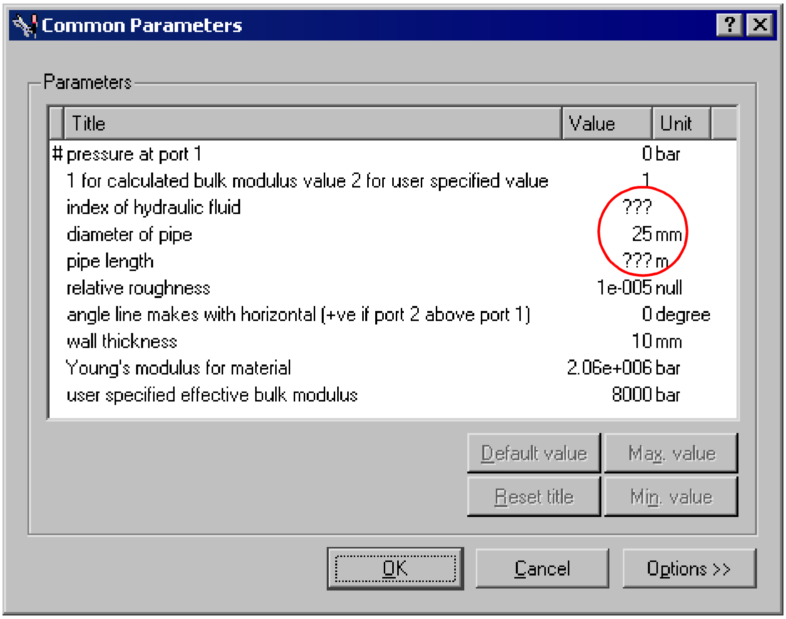
3. For the line from the node to the motor, select the submodel ***HL01***.
4. For the line between the node and the relief valve, the submodel ***DIRECT*** is already selected and this is exactly what we want.

Step 2: Set parameters and run a simulation

1. Go to *Parameter* mode and set parameters for *HL01* and *HL03* so that both **pipe lengths are 5 m and pipe diameters are 10 mm**.

   This can be done one at a time. However, we can do it another way. Press the *Shift* key on click on the *HL03* and *HL01* line runs so that they are selected. Use **Parameters ▶ Common parameters**. Figure 1.20 shows the *Common parameters* dialog box.

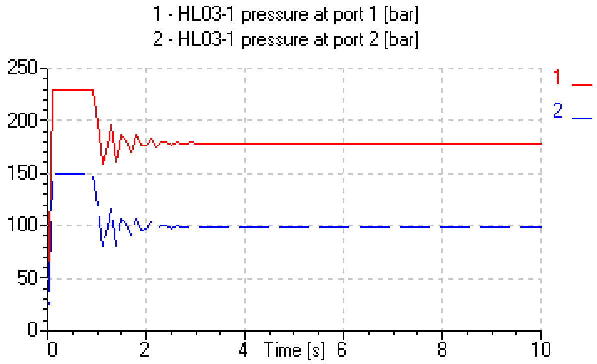
Figure 1.20: The common parameters of the two line submodels.



Note that *???* indicates that different values are set in the line submodels. Set the *index of hydraulic fluid* to 1, *diameter of pipe* to 10 and *pipe length* to 5.

2. In FP04-2 reset the *saturation pressure (for dissolved air/gas)* to 0 bar.
3. Run a simulation with the default run parameters.
4. Plot the two pressures in *HL03*.

Figure 1.21: Pressures at the ends of pipe joining pump to node.

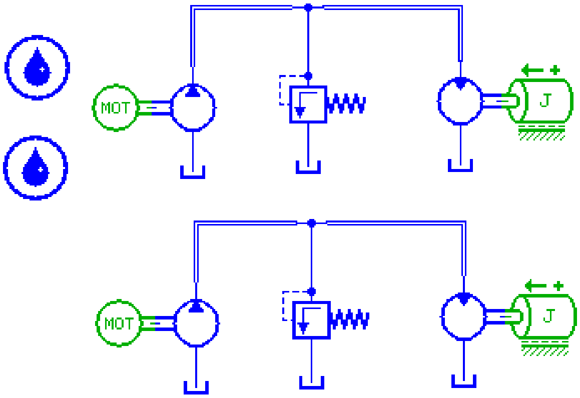


Note that there is a large pressure drop along the line. This could be regarded as a sizing problem but in addition it would be bad practice to site the relief valve so far from the high pressure point.

Step 3: We now investigate other line submodels.

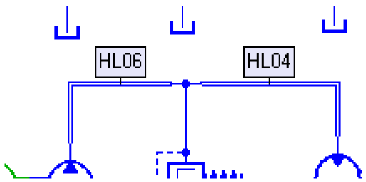
1. Return to *Sketch* mode and *Copy-Paste* part of the system as shown:

Figure 1.22: Part of the system is duplicated.



2. In *Submodel* mode change the lower two line submodels as follows:

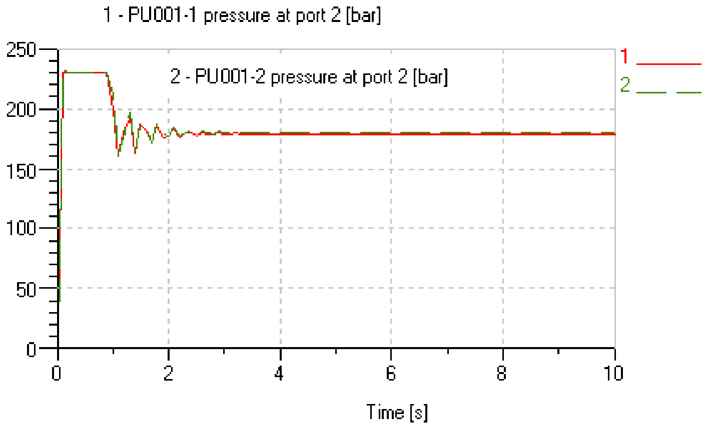
Figure 1.23: New line submodels.



This system will enable you to make direct comparisons between results.

3. Go to *Run* mode and do a simulation. Plot the pressure at the pump outlet (*pressure at port 2*).

Figure 1.24: Pressure at pump outlet.

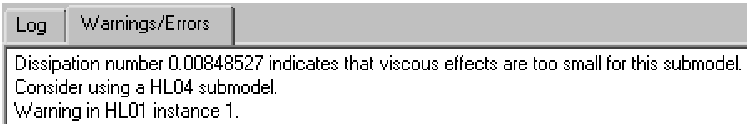


We note that the curves are virtually the same. (Try zooming.) There is absolutely no advantage to using *HL04* and *HL06* instead of *HL01* and *HL03*. If we separated the two systems and ran then independently we would find run times for the more complex submodels were higher.

4. Change the communication interval in the *Run Parameters* dialog box to 0.001s and rerun the simulation.

   If you have a look at the *Warnings/Errors* tab of the *Simulation run* dialog box, you will find that some checks are performed by the line submodels (see Figure 1.25). A similar message is issued for *HL03*.

Figure 1.25: Messages under the Warning tab.

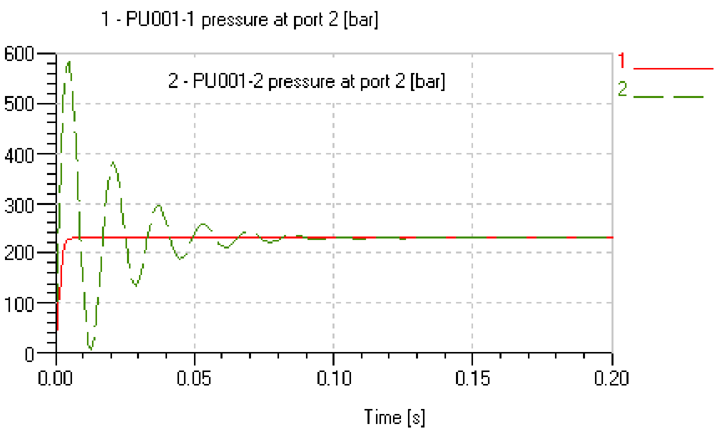


It is suggested that:

- *HL01* should be replaced by *HL04* and
- *HL03* should be replaced by *HL06.*

In other words with this communication interval the lower subsystem is better than the upper. If you replot the pressures at the pump outlets, there are clearly differences. This is what happens if you zoom.

Figure 1.26: Zoomed pressures at pump outlet.



The violent (and unrealistic) start up has created this oscillation in the pressure of about 56 Hz. It is damped out by 0.1 seconds. Why did we get no warning message in the previous run? The answer is that a lot of checks are applied to your submodel choices when the run starts. These take into account the fluid properties, the pipe dimensions and the communication interval. When the communication interval was 0.1 seconds, it would have been impossible to see these oscillations and hence no warnings are issued.

Some very simple arithmetic gives two important points:

1. If you want to see *f* Hz you need a communication interval no bigger than about *1/(10f)* seconds.
2. A communication interval of *x* seconds enables you to see frequencies down to about *1/(10x)* Hz. Thus with 0.1 seconds you can see down to about 1 Hz.

In the current example we are probably not interested below 1 Hz and 0.1 seconds, *HL01* and *HL03* are very suitable. **The following note is vitally important.**

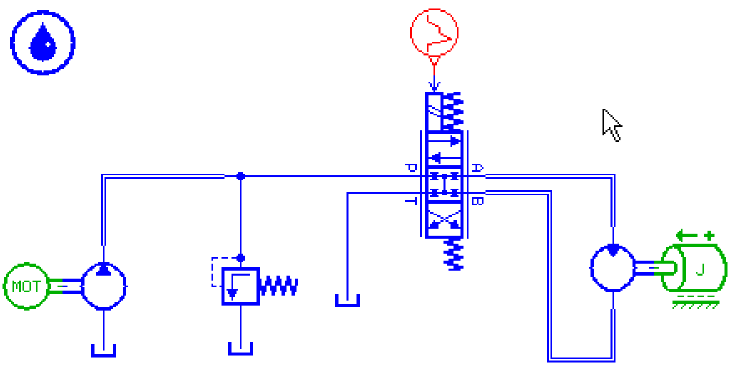
Note: **It is a common mistake in modeling hydraulic systems to always use line submodels of high complexity. The correct procedure is to use the simplest line submodels that will achieve the modeling objective. Be aware of the frequencies you are interested in and the frequencies you can see with the current communication interval.**

If you make a bad choice, you may multiply the CPU time by 10 or 100 and force the integrator to compute high frequency phenomena which are of no interest to you and will be invisible with the communication interval set! The messages under the *Warning* tab are very helpful. Read them.

See also Chapter 4:Selecting submodels for Hydraulic Lines.

1.5 Example 4: Valves with duty cycles

Figure 1.27: Hydraulic system with servovalve.



Objectives

- Introduce valves controlled by duty cycles.
- Use the plot manager to plot flow rates against differential pressure.

Step 1: Build the system and set parameters

1. Build the system shown in Figure 1.27 and save it as *servovalve*.

   Note that you have a directional valve that you will use to change the direction of rotation of the load. You will need to use two new components:

   - A 3-position 4-port direction valve found in the first *Hydraulic* category, and
   - A duty cycle component found in the *Signal, Control and Observers* category.

2. When the new system sketch is complete, use *Premier submodel* to get the simplest combination of submodels.

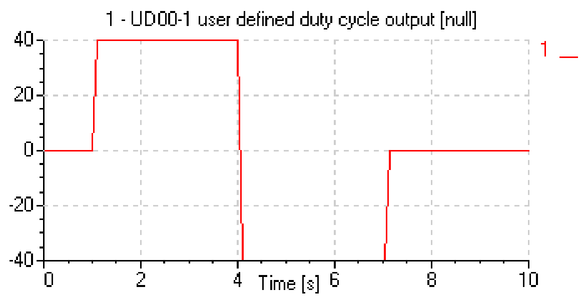
3. Set the parameter values for the duty cycle submodel *UD00* as follows:

Title	Value
duration of stage 1 [s]	1
output at start of stage 2 [null]	40
output at end of stage 2 [null]	40
duration of stage 2 [s]	3
output at start of stage 3 [null]	-40
output at end of stage 3 [null]	-40
duration of stage 3 [s]	3

Note: If you do not change the parameters, the valve will not open. The motor and load will not move at all! For simplicity leave the other submodel components with their default settings.

This gives a signal as follows:

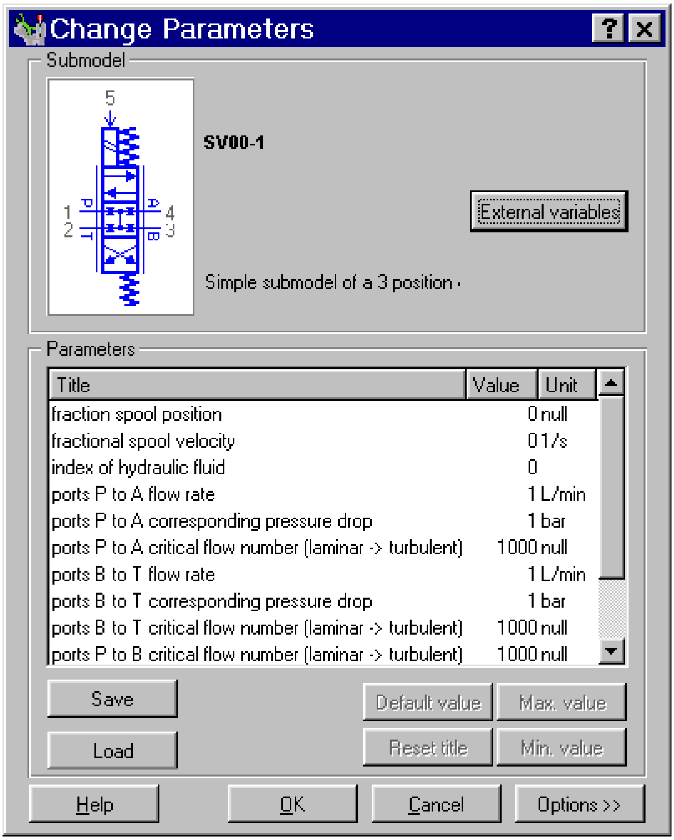
Figure 1.28: The duty cycle controlling the valve.



4. Select the directional valve (Figure 1.27).

   You will not change any parameters but an understanding of the parameters of *SV00* will help you to set those of *UD00*. The spool has some state variables which are the first two items in the list. In *Parameter* mode, their values are the initial values of these state variables. The spool position is a fraction and so is a dimensionless quantity in the range -1 to 1.

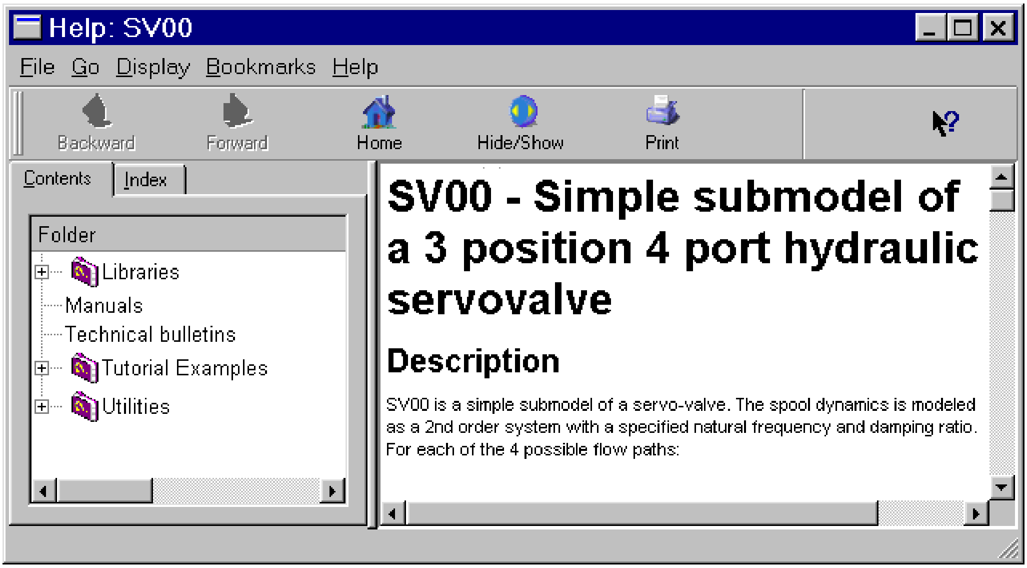
Figure 1.29: Parameters of the servo valve



The next 12 items determine the hydraulic flow characteristics of the valve covering the 4 possible flow paths. When the valve is in one extreme position with fractional spool position +1, P is connected to A and T to B. In the other extreme position the spool position is -1 and the connections are A to T and B to P.

When the spool position is 0, there is no flow. To define the flow characteristics of the valve in the extreme positions a flow rate pressure drop pair is used. The default values of these are 1 L/min and 1 bar. These values can normally be found in a manufacturer's catalogue. The parameter *critical flow number (laminar -> turbulent)* is less important and can be left at its default value. You can find the details for any submodel if you click on the *Help* button. For *SV00*, this produces the dialog box shown in Figure 1.30.

Figure 1.30: Help for submodel SV00.



The rating of the *valve rated current* is set to 40 mA. This means that an input signal of 40 units will produce a fraction spool position of 1. As the spool moves it behaves like a second order system. You can specify the natural frequency and damping ratio.

5. Enable *Discontinuities Printout* in the *Run Parameters* dialog box.
6. Run a simulation with default run parameters.
7. Select the relief valve component and plot the three quantities:
   - *Flow at relief valve outlet [L/min]*
   - *Pressure at relief valve inlet [bar]*
   - *Pressure at relief valve outlet [bar]*

   on the same plot.

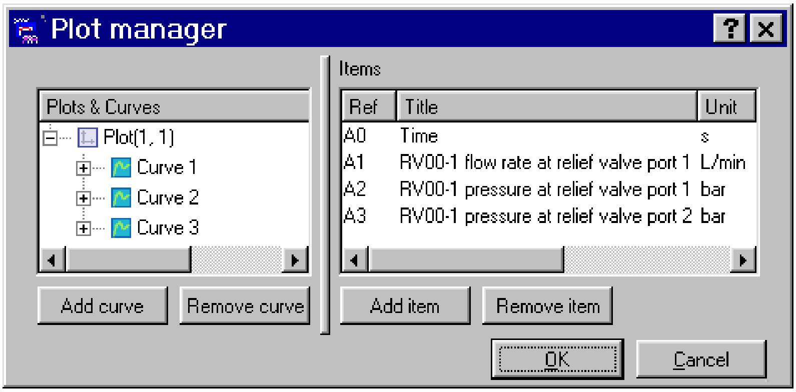
Step 2: Plot the flow rate against the differential pressure for the relief valve.

This is a very common requirement for a 2-port valve and it involves use of the plot manager.

1. Start the *Plot manager* by clicking on the  button in the tool bar on your plot.

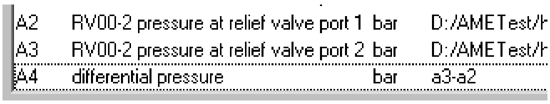
   The Plot manager is displayed as in Figure 1.31.

Figure 1.31: The Plot Manager.

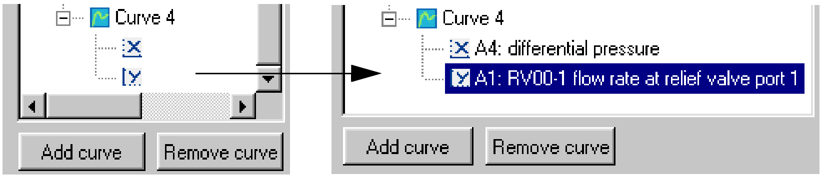


In the right window we have the three quantities we requested and time. We must create a new variable which will be the pressure drop across the valve.

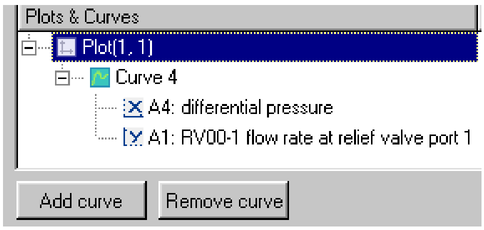
2. Click on *Add item* and construct the new variable.



3. Click on *Add curve* and drag and drop the flow rate onto Y and the pressure drop onto X.

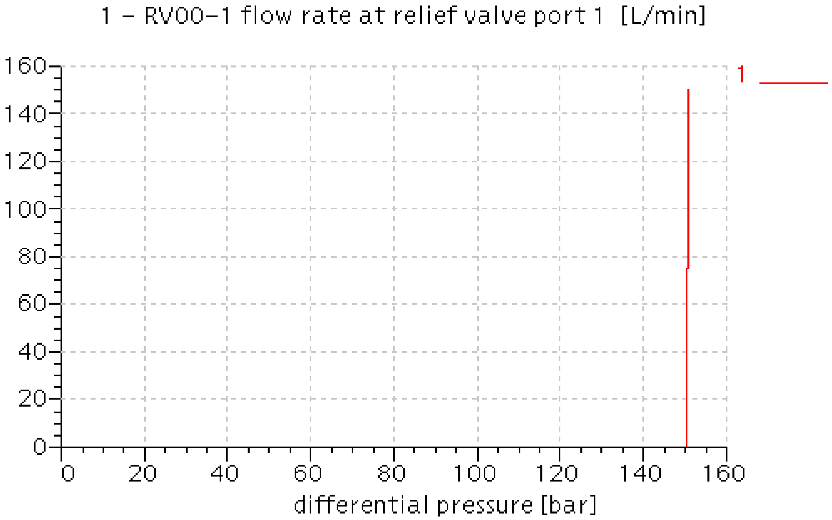


4. *Curve 1* to *3* are no longer required so select each in turn and then click on *Remove curve* so that only *Curve 4* remains.



5. Finally click on *OK* to see the plot.

Figure 1.32: The relief valve flow rate pressure drop characteristics.

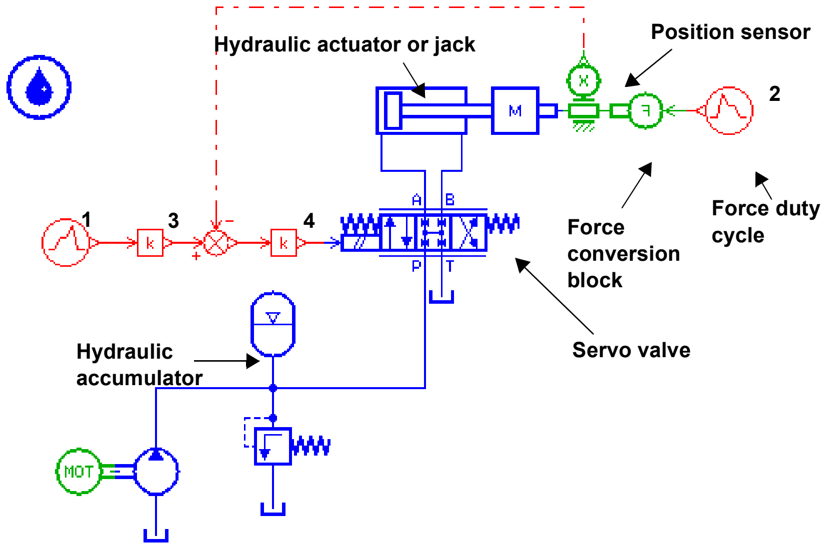


1.6 Example 5: Position control for a hydraulic actuator

Objectives:

- Use a simple proportional control system to achieve a prescribed cycle in a hydraulic system.
- Show the consequences of using an unequal area actuator.
- Show saturation in a servo-valve.
- Study stability and instability in the control system.

Figure 1.33: The position control system.



The system sketch for this exercise is shown in Figure 1.33. The hydraulic actuator (or jack) moves a load and there is control using position feedback. The position sensor is used to convert the actuator displacement to a signal. A position duty cycle is specified by a duty cycle submodel. The duty cycle position is compared with the position indicated by the sensor to produce an error. The error is subjected to a gain and the signal transferred to the servo-valve. A further duty cycle supplies an external force to the actuator via the position transducer.

Step 1: Build the system and set parameters

1. Build the new system and save it as *actuator*.

   The position sensor is found in the category labeled *Mechanical*. A signal port is used to pass the displacement into the feedback loop.

2. Use the *Premier submodel* button to select the simplest possible submodels combinations.

3. Set parameters for the submodels using the suggested values in the following table:

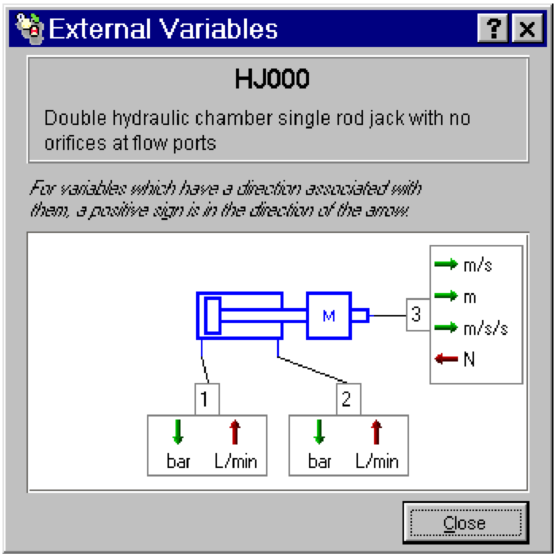
Submodel	**Number on sketch if any**	**Title**	**Value**
HJ000		piston diameter [mm]	30
		diameter of rod [mm]	20
		length of stroke [m]	1
PU001		pump displacement [cc/rev]	35
UD00	1	duration of stage 1 [s]	1
		output at end of stage 2 [null]	0.8
		duration of stage 2 [s]	3
		output at start of stage 3 [null]	0.8
		output at end of stage 3 [null]	0.8
		duration of stage 3 [s]	1
		output at start of stage 4 [null]	0.8
		output at end of stage 4 [null]	0.2
		duration of stage 4 [s]	3
		output at start of stage 5 [null]	0.2
		output at end of stage 5 [null]	0.2
UD00	2	output at the end of stage 1 [null]	1000
		output at end of stage 1 [null]	1000
SV00		valve natural frequency [Hz]	50
		valve damping ratio [null]	1
		valve rated current [mA]	200
DT000		gain for signal output [1/m]	10
GA00	3	value of gain [null]	10
GA00	4	value of gain [null]	250

Note:

- The parameters of *HJ000* give a very 'unequal area actuator' and the plots can demonstrate the consequences of this.
- The external force to the right of the actuator is a constant value of 1000 N.
- The gain in the displacement sensor converts the jack position that is in the range 0 to 1 m to a signal in the range 0 to 10. The gain for the submodel *GA00* attached to the duty cycle submodel, is also 10. By this means, the duty cycle will directly represent the actuator displacement in m.

4. When you set the parameters for *HJ000*, click on the *External variables* button to call up the dialog box shown in Figure 1.34.

Figure 1.34: External variables of HJ000



This indicates that a positive velocity means the rod is moving to the right. The greater the displacement, the further it is to the right. In the current case, a zero displacement and velocity means that the rod and piston are stationary and the piston is at the extreme left end of the jack.

The meaning of the sign of the acceleration and external force should be clear. A positive external force opposes the other variables i.e. makes a negative contribution to the acceleration. Hence it is trying to reduce the velocity and displacement.

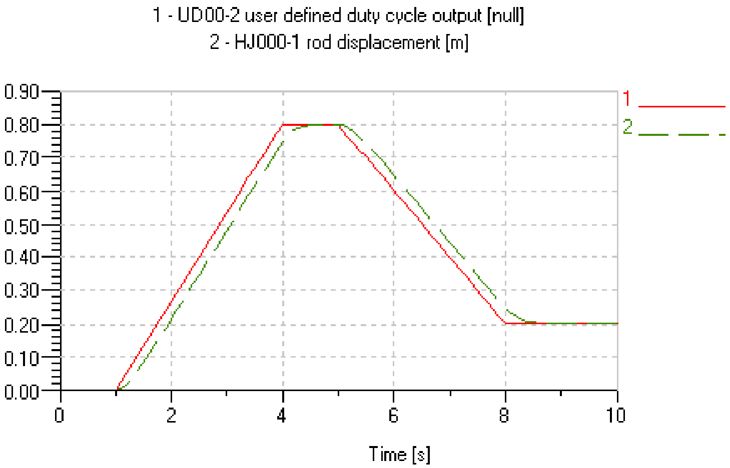
Remove the dialog box by clicking on *Close*.

Step 2: Run simulation and plot results

1. Run a simulation setting a final time of 12 s and a communication interval of 0.05 s.

2. Plot the following graphs:
    - Actuator displacement and duty cycle output on the same graph.
    - Flow rate at the two actuator ports on the same graph.
    - Flow rate at pump outlet and flow rate at relief valve outlet on the same graph.
    - Fractional spool position.

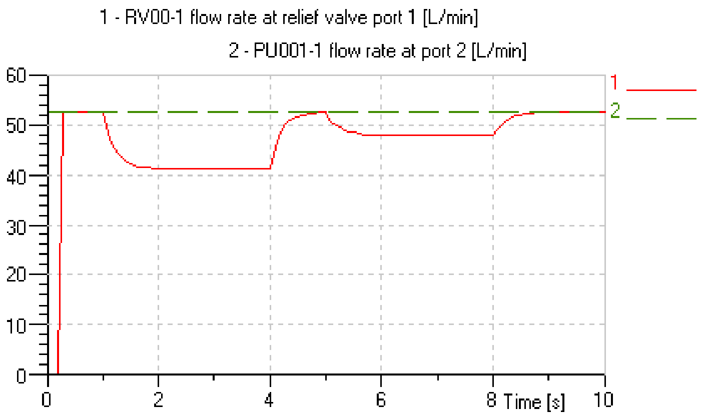
Figure 1.35: The required and the actual displacement.



The first plot (Figure 1.35) gives an idea of how closely the actual performance matches the required duty cycle.

3. Plot the output from the summing junction (strictly speaking a differencing junction) that gives you the position error in *m*.
4. Try changing the gain attached to the servo valve, the servo valve natural frequency and damping ratio.
5. **Include a high gain value that makes the system unstable**.
6. Try introducing a dead band, up to about 10%.

Figure 1.36: Pump and relief valve flow rates.

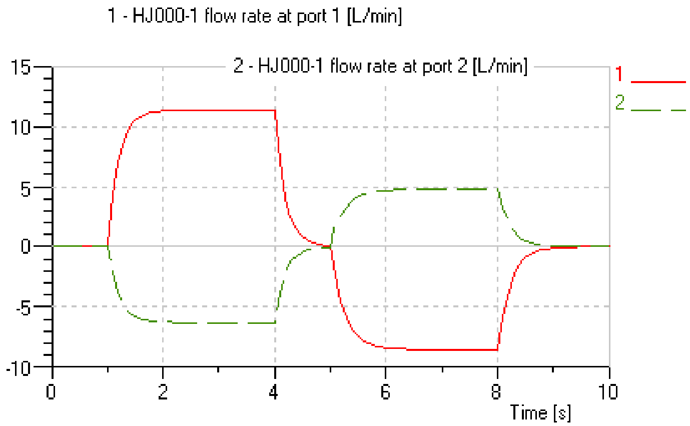


A typical plot for the flow rates from the pump and relief valve outlets is shown in Figure 1.36. If you had chosen the pump inlet flow rate instead of the pump outlet flow rate, negative values would have appeared on the graph. This is easily explained if you click on the *External variables* button of the *Variable List* dialog box. For both ports of the pump a positive flow rate indicates flow out of the pump. It follows that the flow rate at the pump inlet must be negative.

7. Plot the two flow rates in the actuator HJ000.

   For the this submodel, flow rate is an input on both flow ports. This means a positive flow rate indicates flow into the component. Figure 1.37 shows typical results. Note how different the magnitudes of the flow rates are due to the unequal areas.

Figure 1.37: Hydraulic actuator flow rates.



8. Plot the valve spool fractional displacement.

   This gives an idea of how close to saturation the valve is during the duty cycle. If a value of +*1* or -*1* is reached, the valve is saturated.

1.7 Example 6: Simple design exercise for a hydraulic suspension

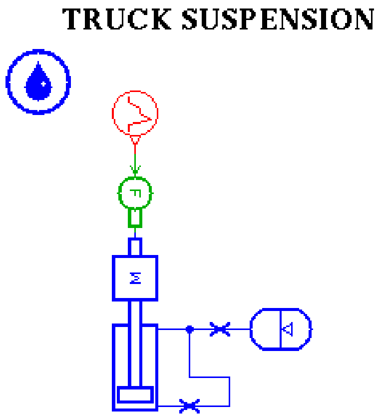
Objectives:

Do a simple initial design study for a hydraulic suspension using:

- Analytical analysis.
- **AMESim** standard runs.
- Batch runs.
- Linear analysis.

The system is shown in Figure 1.38. The hydraulic jack with the two orifices is the damper and the accumulator is the spring. It is proposed to use this suspension on the cab of a truck. The load on each suspension strut is 250kg.

Figure 1.38: A simplified hydraulic suspension.



Step 1: Build the system and run a simulation

1. Build the system using *Premier submodel*.

   Much sizing can be done by simple calculations but simulation can be a great help in rapidly confirming the calculations and adding dynamics to the steady state values. The two ports of the jack are interconnected and in equilibrium. The pressures above and below the jack piston will be the same. Using a force balance in the equilibrium position in terms of the piston area A_{pist} and rod area A_{rod}

$$PA_{pist} \angle P(A_{pist} \angle A_{rod}) = 250\text{x}g$$

   It follows that

$$PA_{rod} = 250g$$

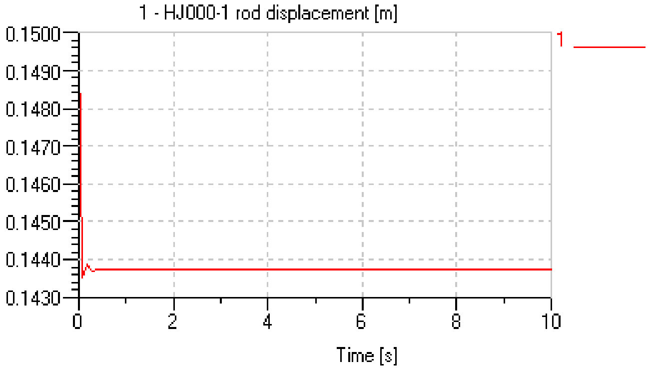
From this if we want an operating pressure of about 70 bar the diameter of the rod must be about 22.3 mm. We will use a rod diameter of 20 mm and a piston diameter of 40 mm.

2. Set the parameters of the following table.

Submodel	Parameter title	Value
HJ000	rod displacement [m]	0.15
	piston diameter [mm]	40
	diameter of rod [mm]	20
	angle rod makes with horizontal [degree]	90
	total mass being moved	250

3. Run a dynamic simulation for 10 s.

Figure 1.39: Pressure and displacement plots.



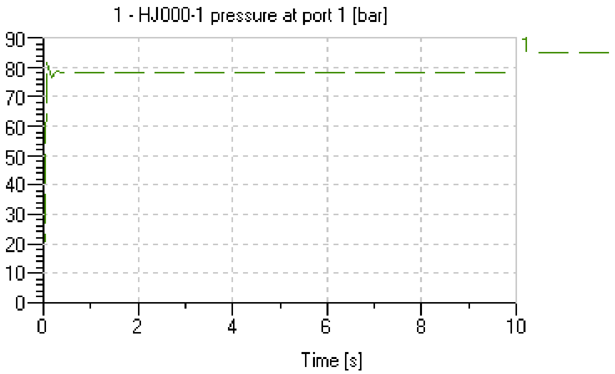


Figure 1.39 shows the system pressure and the displacement.

Problem 1: The starting values are poor.

Problem 2: The accumulator spring with its precharge pressure of 100 bar is taking no part in this simulation. The only spring involved at the moment is the hydraulic fluid.

Solution to problem 1:

1. In *Parameter* mode select **Parameters ▶ Set final values**.

   This will give reasonable starting values for state variables. You will find that the piston has dropped slightly from the mid-position.

2. Reset the following parameters:

Submodel	Parameter title	Value
HJ000	rod displacement [m]	0.15
	rod velocity [m/s]	0

3. Run a simulation again and check that the system is in equilibrium with the rod in mid-position.

Solution to problem 2:

The two parameters we can vary are the precharge pressure and volume of the accumulator. For the accumulator to work as a spring, the precharge pressure must be lower than the equilibrium pressure.

The volume of fluid in the jack varies according to the piston position. This is due to the rod. The difference between the minimum and maximum oil volume is

$$A_{rod} \text{ x } stroke$$

which is 0.1 L. The accumulator volume should be a bit bigger than this but certainly not 10 L.

Step 1: Investigate the spring rate

1. Set the following values

Submodel	Parameter title	Value
HA001	gas precharge pressure [bar]	10
	accumulator volume [L]	0.5

1. Do a run and verify that these values do not disturb the equilibrium.

   The values should have changed the spring rate but not the equilibrium position. We need now to investigate the spring rate.

2. Set the following values

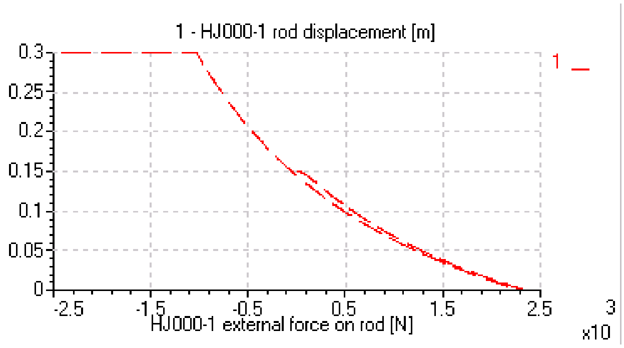
Submodel	Parameter title	Value
UDOO	output at start of stage 1 [null]	0
	output at end of stage 1 [null]	2500
	duration of stage 1 [s]	40
	output at start of stage 2 [null]	2500
	output at end of stage 2 [null]	-2500
	duration of stage 2 [s]	80

3. Do a run for 120 s.
4. Plot graphs of:
    - rod displacement (*HJ000*)
    - pressure at port 1 (*HJ000*)

    against

    - external force on rod (*HJ000*)

Figure 1.40: Displacement against force.



The force value of 2500 N pushes down on the suspension with a value corresponding to the weight of the car. The force of -2500 effectively takes the complete weight off the suspension. The slow evolution of the force duty cycle ensures that the system is very close to equilibrium at all times.

The plot of displacement against force (Figure 1.40) shows the non-linear nature of the spring. It also shows that the suspension does not ‘bottom out’ but it does ‘top out’.

Figure 1.41: Force against pressure.

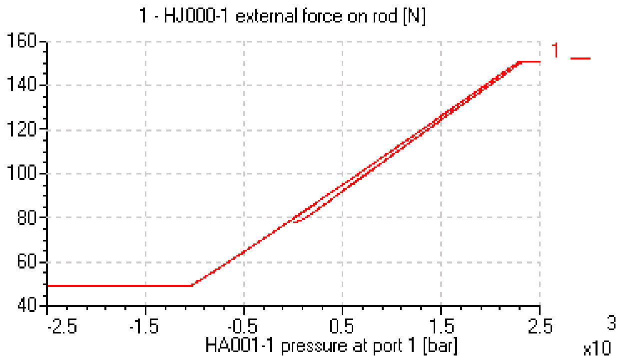


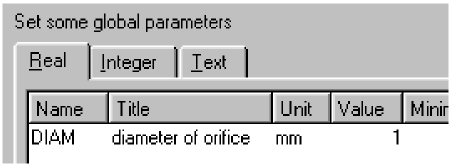
Figure 1.41 shows that maximum pressure is 160 bar and the minimum is about 40 bar which occurs when the suspension tops out.

We could continue by doing further analytical calculations. Alternatively we could do batch runs varying the accumulation pre-charge pressure and accumulator volume and the interested reader could try this.

However, we will end the exercise by considering the damping of the suspension which is mainly provided by the two orifices. For simplicity we will assume they are of the same characteristics.

Step 2: Setup a batch run varying the diameters of the orifices with the vehicle subject to a step change in force

1. Select **Parameters ▶ Global parameters**.
2. Set up the global parameter shown.



3. Set the following parameters for BOTH orifices:

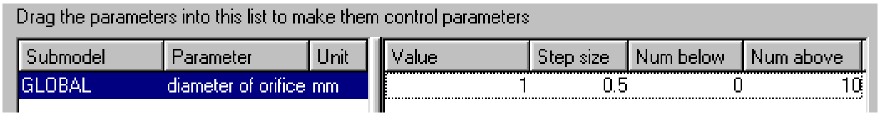
Submodel	Parameter title	Value
OR000	1 for pressure drop/flow rate pair 2 for orifice diameter	2
	equivalent orifice diameter [mm]	DIAM

4. Set up a duty cycle to give a step increase in force:

Submodel	Parameter title	Value
UD00	output at start of stage 1 [null]	0
	output at end of stage 1 [null]	0
	duration of stage 1 [s]	1
	output at start of stage 2 [null]	500
	output at end of stage 2 [null]	500
	duration of stage 2 [s]	9

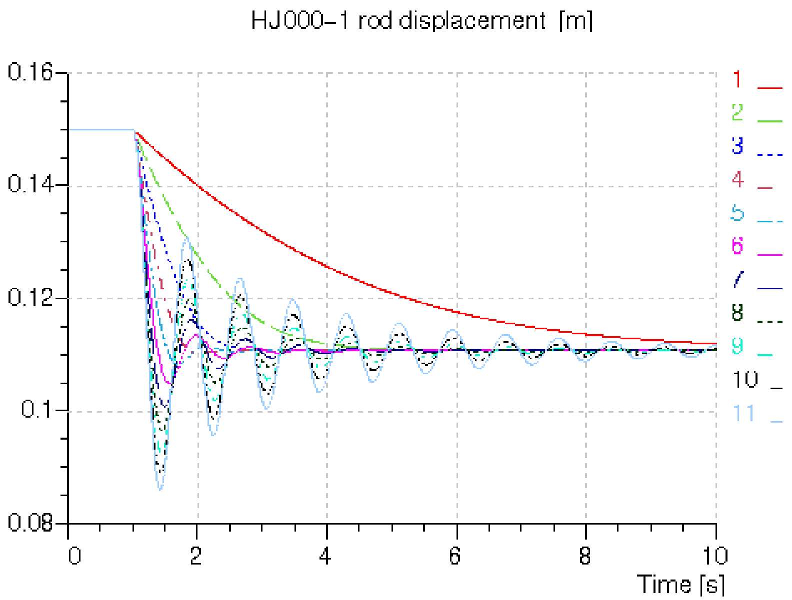
5. Select **Parameters ▶ Batch parameters**.
6. Drag and drop the global parameter into the *Batch parameters* dialog box and set the following values for a batch run:

Figure 1.42: Batch parameters



7. Perform a batch run for 10 s and plot the displacement of the piston.

Figure 1.43: Batch run results for rod displacement



The batch run will use orifice diameters of 1 to 6 mm in steps of 0.5mm. Zooming in on the plot it becomes clear that 3 mm gives a reasonable degree of damping.

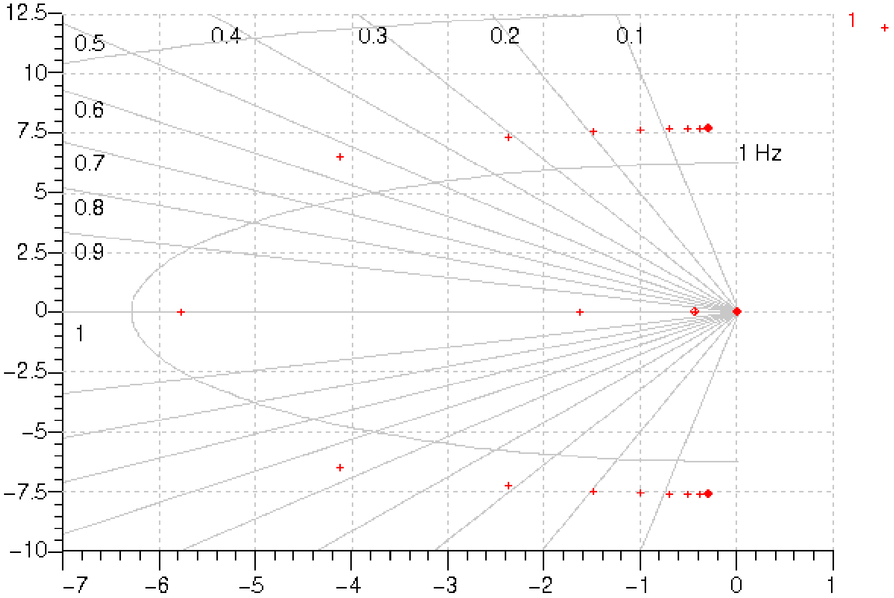
8. Remove the step from *UD000* so that there is a constant force of 0 N.
9. Insert a linearization time at 10 s.
10. Repeat the batch run and look at the damping ratio for the oscillatory frequency.

Looking at the eigenvalues selecting the *.jac0.1* to *.jac0.11* files we see that below 2.5 mm the system is very highly damped. However, the results for the 1 mm diameter give an oscillatory frequency of about 25 Hz which is curious but could be investigated with tools such as modal shapes. For diameters of 2.5 mm and greater there is an oscillatory frequency of about 1.23 Hz and the damping ratio is as follows:

Diameter of orifice [mm]	**Damping ratio**
2.5	0.533
3	0.308
3.5	0.194
4	0.130
4.5	0.091
5	0.067
5.5	0.050
6	0.039

We can see the evolution of these eigenvalues in a root locus plot.

Figure 1.44: Root locus plot.



A more refined search between 2.0 and 3.0 mm would be a good idea but 2.5 mm seems reasonable.

Chapter 2: Theory of fluid properties

We will concentrate mainly on three fluid properties in this chapter:

- The density which leads to mass and hence to hydraulic inertia effects.
- The viscosity which leads to the hydraulic friction effects.
- The compressibility and thus the bulk modulus which leads to the hydraulic system stiffness. Notice that the compressibility effect can be modified by air release, cavitation phenomena and by expansion of a pipe, hose or chamber containing the hydraulic fluid.

2.1 Density and compressibility coefficient

The **density** is the mass of a substance per unit volume:

$$\rho = \frac{M}{V}$$

Density has dimensions of [M/L^3] and is expressed in kilograms per cubic meter [kg/m^3]. As mentioned previously the density is a function of the pressure and the temperature:

$$\rho = \rho(P, T, nature\ of\ fluid)$$

This function can be approximated by the first three terms of a Taylor series: $\rho(P + \Delta P, T + \Delta T) = \rho + \left(\frac{\partial\rho}{\partial P}\right)_T \Delta P + \left(\frac{\partial\rho}{\partial T}\right)_P \Delta T$

This can also be expressed as:

$$\rho = \rho\left(1 + \frac{\Delta P}{B} - \alpha\Delta T\right)$$

with

$$B = \rho\left(\frac{\partial P}{\partial\rho}\right)_T$$

and

$$\alpha = -\frac{1}{\rho}\left(\frac{\partial\rho}{\partial T}\right)_P$$

This equation is the linearized state equation for a liquid. Using the definition of the density, the two coefficients α and B can also be expressed as:

$$B = -V\left(\frac{\partial P}{\partial V}\right)_T \text{ and } \alpha = \frac{1}{V}\left(\frac{\partial V}{\partial T}\right)_P$$

B is known as the **isothermal bulk modulus** or for simplicity the bulk modulus and α is known as the **cubical expansion coefficient**. Since fluid density varies with the applied pressure, this implies that a given mass of fluid submitted to a pressure change changes its volume. This phenomenon leads to the definition of the **compressibility coefficient** β:

$$\beta = -\frac{1}{V}\left(\frac{\partial V}{\partial P}\right)_T$$

where β is expressed in units Pa^{-1} (or m^2/N). Considering the relation $V\rho = M$ for a closed hydraulic circuit the mass is constant, and hence:

$$d(V\rho) = 0 \text{ and } Vd\rho + \rho dV = 0$$

it follows that

$$\frac{d\rho}{\rho} = -\frac{dV}{V}$$

Using the definition of the compressibility coefficient β we obtain:

$$\frac{1}{\beta} = \frac{\rho}{\frac{\partial\rho}{\partial P}}$$

More usually we use the bulk modulus B also known as the volumetric elasticity modulus:

$$B = \frac{\rho}{\frac{\partial\rho}{\partial P}}$$

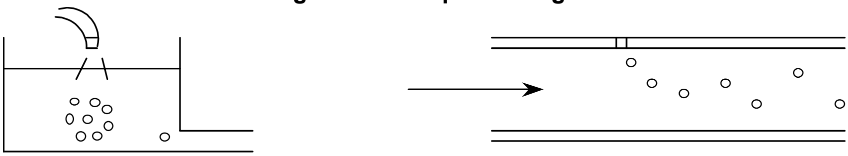
The relation between ρ and B implies mass conservation. This relation must be RIGOROUSLY RESPECTED in the calculations. In the modeling and simulation context of fluid energy systems, disregarding the relation between ρ and B leads to abnormal evolutions of pressure in the closed circuit submitted to compression and expansion cycles. This phenomenon is strongly accentuated if aeration occurs in the circuit (when dissolved air in the fluid reappears in the form of bubbles). We shall approach this point by examining the phenomena of aeration and cavitation.

The air can also have adverse consequences on a fluid compressibility. In liquid air can be present in two forms: entrapped and dissolved.

Entrapped air

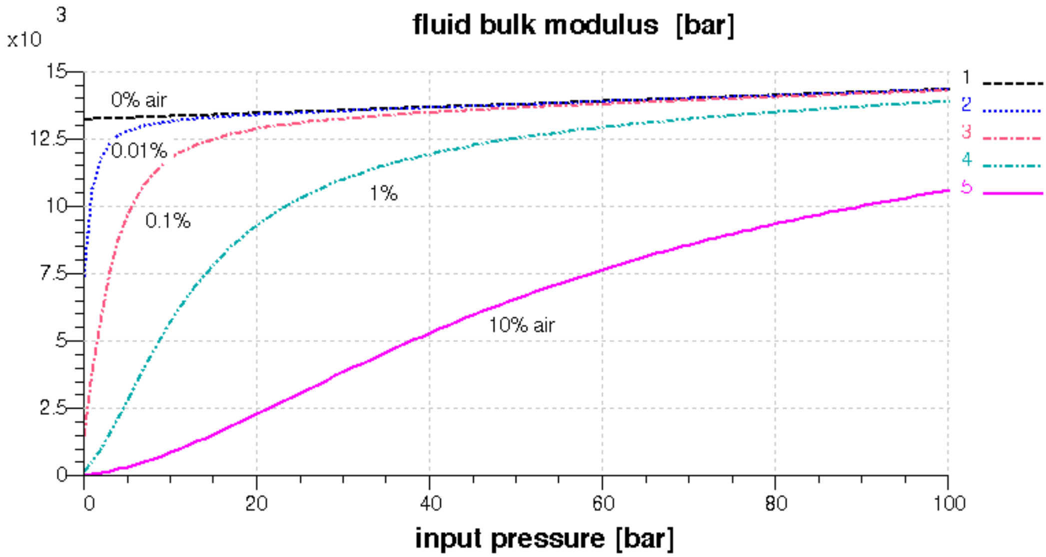
When the return pipe is not submersed in the tank the liquid jet can entrain some air bubbles in the tank. Another phenomenon that affects the quantity of air in liquid is the leakage.

Figure 2.45: Liquid leakage



This air stays in the liquid as cavities and can modify the fluid compressibility. In this context we talk about effective bulk modulus. Figure 2.46 shows the bulk modulus of a diesel fuel at 40 °C with 0, 0.01, 0.1, 1, 10% air. The plot is obtained using the system shown. The model of the diesel fuel properties is based on accurate experimental measurements and are designed for use with injection system which are very fast acting. For this reason air is assumed to be entrained rather than dissolved.

Figure 2.46: Air is entrained



Dissolved air

Air can also be dissolved in a liquid. A certain amount of air molecule can be part of the liquid. In this case the dissolved air does not significantly change the fluid properties.

2.2 Air release and cavitation

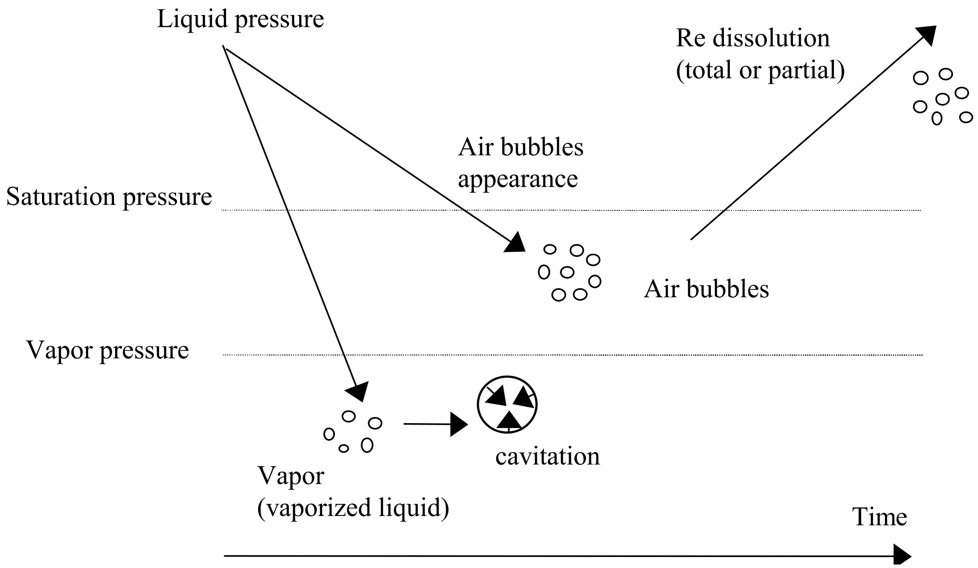
Air can be dissolved or entrained in liquids and it is possible for air to change from one of these two forms to the other depending on the conditions to which the fluid is subjected.

Suppose the fluid is in equilibrium with a certain percentage of dissolved gas (usually air: nitrogen and oxygen). Lowering the pressure above a critical value called the **saturation pressure** induces aeration. This is the process where the dissolved gas forms air bubbles in the liquid until all the dissolved gases or air are free. The exact point where all the dissolved gas has come out of solution is difficult to pin-point because it depends on the chemical composition and behavior of the gas. This is a non-symmetrical dynamic process: the growing process does not have the same dynamics as when air bubbles disappear. In consequence the total amount of bubbles created when the pressure drops may or may not be redissolved in the liquid when it rises again.

If the pressure is dropped further and above another critical value called the **vapor pressure**, the fluid itself starts to vaporize. It corresponds to a liquid phase change. At some point only fluid vapor and gas exist. In liquid systems the term cavitation usually refers to the formation and collapse of cavities in the liquid even if cavities contain air or liquid vapor.

To summarize with a sketch what we have introduced see above:

Figure 2.47: Air release and cavitation

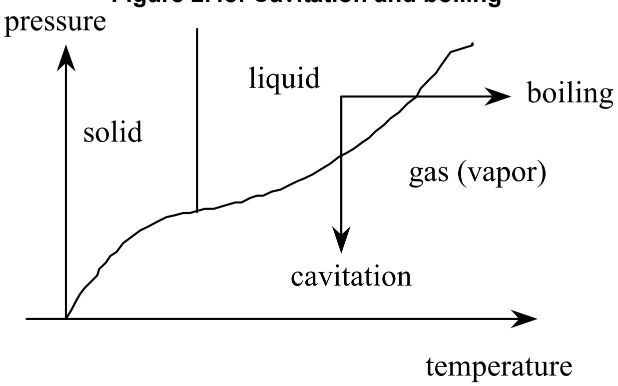


The development of a cavity is now recognized as being associated with a nucleation center such as microscopic gas particles, wear or wall asperities. When the liquid is subjected to a tensile stress, cavities do not form as a result of liquid rupture but are caused by the rapid growth of these nuclei.

To understand this, think of beer (or champagne if you prefer) in a bottle, when it is closed you see no air bubbles and the liquid does not look fizzy. The pressure in the bottle is above the saturation pressure of the gas in the liquid. When you open the bottle suddenly bubbles appear and so the dissolved gas (molecules of gas held in the liquid) starts to appear as gas. In fact the liquid is gas saturated and the atmospheric pressure is less than the saturation pressure of the liquid. This phenomenon is clearly not cavitation but air release (aeration). Considering nuclei effects, bubbles form only at particular places in your glass: around the glass (due to small asperities) and round any particles present in the liquid. Theoretically, if your liquid was perfectly pure and the wall of the system perfectly regular, air release or cavitation would occur with great difficulty!

The key point about cavitation is that it is a phase change: the liquid changes to vapor. A comparison can be made between cavitation and boiling. If we look at the phase diagram below:

Figure 2.48: Cavitation and boiling



Boiling is a phase change at constant pressure and variable temperature and cavitation is a phase change at constant temperature and variable pressure.

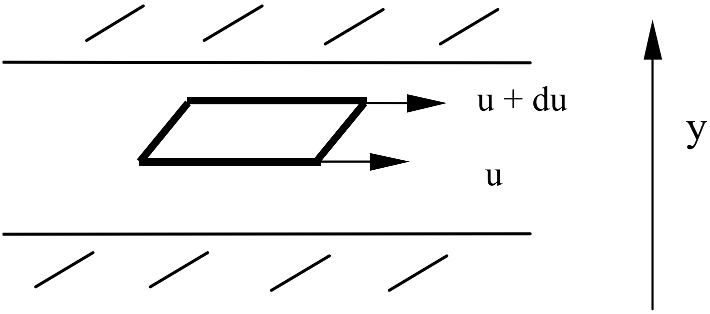
In any system air release starts first and if the pressure decreases further, cavitation may occur. This means that, sometimes, people talk about cavitation when the real phenomenon is air release. Both phenomena can lead to destruction of the material or component.

In both cases it is entrained gas that causes the troubles. When cavities encounter high pressure in the downstream circuit, these bubbles or cavities can be unstable and can collapse implosively. The pressure developed at collapse can be large enough to cause severe mechanical damage in the containing vessel. It is well-known that hydraulic pumps and pipework can be badly damaged by cavitaton and air release.

In all classical hydraulic systems air release and cavitation must be avoided to prevent material destruction but sometimes it is required like for injection systems to prepare the spray formation.

2.3 Viscosity

Figure 2.49: Viscosity



Viscosity is a measure of the resistance of the fluid to flow. This characteristic has both positive and negative effects on fluid power systems. A low viscosity leads to oil leaks in the dead zone formed between the mechanical parts in movement, and a high viscosity will lead to loss of pressure in hydraulic ducts.

Viscosity is a characteristic of liquids and gases and is manifested in motion through internal damping. Viscosity results from an exchange of momentum by molecular diffusion between two layers of fluid with different velocities. In this sense, the viscosity is a fluid property and not a flow property.

Figure 2.49 shows the relation between shearing constraint and difference of flow velocity between two layers

The definition of viscosity was first given by Newton. Between two layers of distance *dy*, the exerted force between these two layers is given by:

$$F = \mu A \frac{dU(y)}{dy}$$

where *U(y)* is the velocity depending on the radial position *y* and *dU/dy* the velocity gradient. This proportionality expresses the notion of Newtonian fluid and allows the introduction of μ defined as the dynamic viscosity or the absolute viscosity. The dimension of μ is $[ML^{-1}T^{-1}]$ and the SI unit is *kg/m/*s or *Pa* s. The older unit is the Poise, *P*, which is 0.1 *kg/m/s.* However, this is very small and hence the milli Poise, *mP*, is the common unit which is 10^{-4} kg/m/s.

The dynamic viscosity is the constant of proportionality between a stress and the intensity of shearing between two neighboring layers:

$$\tau = \mu \frac{dU(y)}{dy} \text{ and hence } \mu = \frac{\tau}{\frac{dU(y)}{dy}} = \frac{\text{shear stress}}{\text{shear rate}}$$

However the absolute viscosity is not very often used in fundamental equations. For example the dynamics of the elementary volume between the two layers is expressed as:

$$A\frac{d\tau}{dy}dy = \rho A dy\frac{dU(y)}{dt}$$

and thus using the shear stress calculation:

$$\frac{dU(y)}{dt} = \frac{1}{\rho}\frac{d\tau}{dy} = \frac{\mu}{\rho}\frac{d^2U(y)}{dy^2}$$

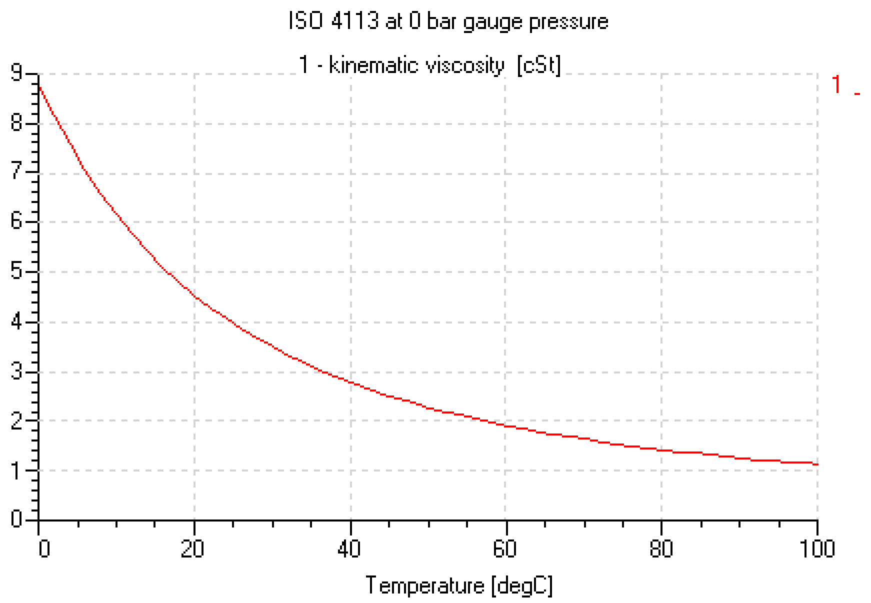
In other formulas (e.g. Navier Stokes) the ratio between the absolute viscosity and the density occurs so often that a new parameter called the kinematic viscosity ν is introduced

$$\nu = \frac{\mu}{\rho}$$

of dimension [L^2 T^{-1}] and so the SI unit is the m^2/s. The older unit of kinematic viscosity is the Stoke, *S*t, which is 10^{-4} m^2/s. However, even this is a very small unit and hence the centistoke *cSt* is the common unit with 1 *cSt* = 10^{-6} m^2/s. This parameter is easily measured with viscometers.

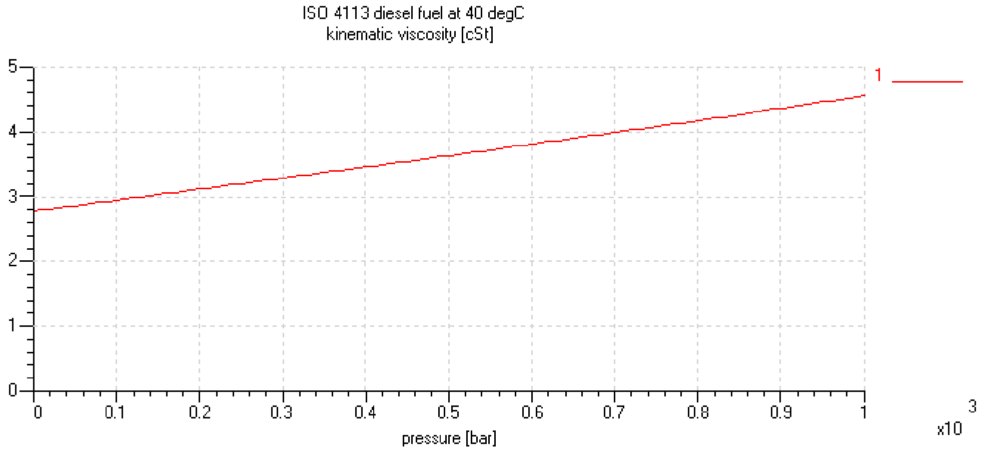
Note that the viscosity varies significantly with the fluid temperature.

Figure 2.50: Viscosity against temperature



Normally in absence of air release and cavitation the variation with pressure is not great unless the pressure is very extreme.

Figure 2.51: Variation with pressure



Viscosity influence on the flow

Another important aspect of the viscosity is its influence on the flow conditions of the fluid. We can distinguish two types of flow conditions:

- **Laminar flow** for which the flow lines are parallel and shearing forces create a pressure drop.
- **Turbulent flow** for which the fluid particles have a disordered, random movement leading to a loss of pressure.

These two conditions can be distinguished using the Reynolds number which is defined as follows:

$$Re = \frac{U\rho d}{\mu} = \frac{Ud}{\nu} = \frac{\text{inertia effects}}{\text{viscous effects}}$$

with

U: average fluid velocity

d: diameter of the duct (hydraulic diameter for others geometries)

ρ: density

μ: dynamic viscosity

ν: kinematic viscosity

The transition between laminar to turbulent flow occurs at the critical Reynolds number. This is not well defined, there exists always a transition region. In a hydraulic line, the critical Reynolds number is generally between 1500 to 2000. For uneven geometries (thin-walled orifices), the critical Reynolds number can be lower than 100.

For non-circular cross sections, the hydraulic diameter can be used to determine the Reynolds number. Hydraulic diameter is defined as follows:

$$d_h = \frac{4\text{xcross sectional area}}{\text{wet perimeter}}$$

We now give two examples:

- Circular orifice of diameter:

$$d_h = \frac{4\pi\left(\frac{d}{2}\right)^2}{\pi d} = d$$

- Rectangular orifice, length L and width l:

$$d_h = \frac{4Ll}{2(L+l)} = \frac{2Ll}{L+l}$$

Hence $d_h \approx 2l$ if $L \geq l$.

Flow through orifices

Orifices (also called restrictions) can be fixed or variable and occur in huge numbers in fluid systems. Not surprisingly in Engineering courses a mathematical description is presented. This is usually based on Bernoulli's equation and leads to the form

$$Q = C_q A \sqrt{\frac{2(P_{up} \angle P_{down})}{\rho}}$$

where C_q is the flow coefficient. This is variously described as

typically 0.7 or

varying with orifice geometry and Reynolds number.

The second alternative is obviously more correct. If we do take a constant value, we are forced to have the gradient of Q against $\Delta P = P_{up} \angle P_{down}$ infinity at the origin! This cannot be and if you try to implement it is a numerical disaster!

Clearly the flow is laminar for sufficiently small pressure drops which means that C_q is certainly not constant. One solution is to perform detailed experiments and compute C_q against Reynold's number. In the context of the orifice (not necessarily circular) the Reynold's number is $Re = \frac{Ud_h}{\nu}$ where U is a mean velocity and d_h the hydraulic diameter. If we take $U=Q/A$, we end up with the form $C_q = f(Q)$ and ultimately with

$$Q = F(Q)$$

It is possible to work with an implicit relationship like this but we would prefer an explicit formula.

This is provided by introducing another dimensionless number known as the flow number and denoted by λ [reference 1]. This is defined as

$$\lambda = \frac{d_h}{\nu}\sqrt{\frac{2(P_{up} \angle P_{down})}{\rho}}$$

From a modeling point of view λ contains quantities we know. Using λ we have

$$Q = \frac{C_q A \nu \lambda}{d_h}$$

and provided we have $C_q \equiv C_q(\lambda)$, we have an explicit relationship which is easy to evaluate. There are no more problems to obtain measurements for $C_q \equiv C_q(\lambda)$ than for $C_q = C_q(Re)$ and so the flow number form has many advantages.

Note:

- Both P_{up} and P_{down} are needed. ΔP is not enough because a pressure drop of 1 bar to 0 bar is not the same as 1001 bar to 1000 bar.
- It is not clear which pressure should be used to calculate ρ and ν. Possibilities are P_{up}, P_{down}, $(P_{up} + P_{down})/2$ or the pressure at the vena contracta. **AMESim** uses $(P_{up} + P_{down})/2$.
- Tables of $C_q \equiv C_q(\lambda)$ can also be compiled using CFD (computational fluid dynamics) software.
- For high values of λ, C_q is approximately constant.
- The lowest value of λ at which C_q is approximately constant is called the critical flow number λ_{crit}.
- The critical flow number for a thin or sharp edge orifice is about 100 and for a long orifice is about 3000.
- For a long edge orifice the constant C_q value is also the maximum value.
- For a sharp edge orifice the maximum C_q value can be slightly greater than the constant value and occurs at a λ value slightly below λ_{crit}.

For general use the **AMESim** submodels OR000 and OR002 require λ_{crit} and the limiting value of C_q. The value of λ is computed from

$$\lambda = \frac{d_h}{\nu}\sqrt{\frac{2(P_{up} \angle P_{down})}{\rho}}$$

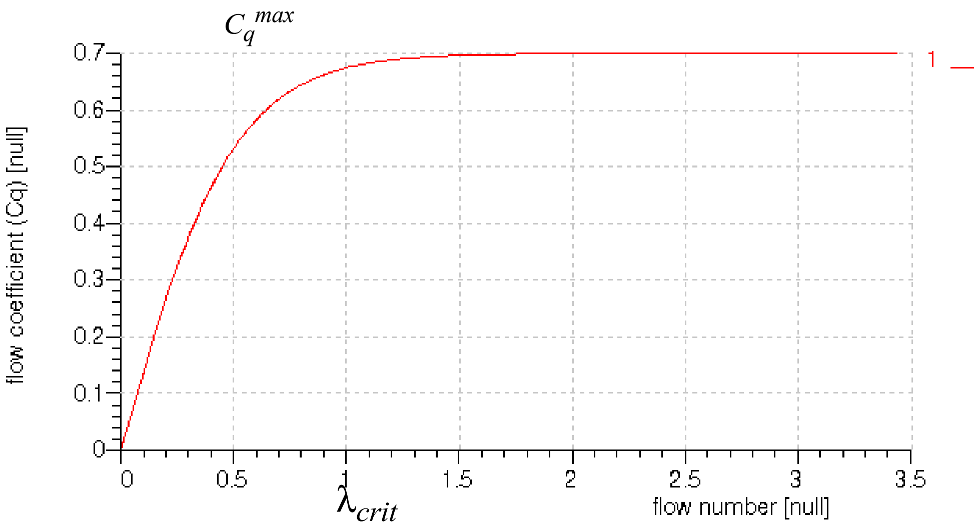
and the flow coefficient is calculated as

$$C_q = C_q^{max}\tanh\left(\frac{2\lambda}{\lambda_{crit}}\right)$$

When $\lambda = \lambda_{crit}$, C_q is about 96% of C_q^{max}.

Figure 2.52 shows a graph of Cq against λ.

Figure 2.52: Graph of Cq against l



Frictional drag

Submodels belonging to this category are used to model resistance to flow in straight tubes and conduit. The pressure losses along a straight tube of constant cross-sectional area are calculated from the Darcy-Weisbach equation:

$$\Delta p = \lambda \cdot \frac{l}{D} \cdot \frac{\rho Q^2}{2A_{min}^2}$$

where:

λ friction coefficient of the segment of relative length *l/Dh=1;*

D_h hydraulic or equivalent diameter;

l length of flow segment.

For this type of submodel the total friction factor ζ is given by:

$$\zeta = \lambda \cdot \frac{l}{D_h}$$

In straight tubes, the resistance to the motion of a liquid or a gas under conditions of **laminar flow** is due to the force of internal friction. This happens when one layer of the liquid (or gas) has a relative motion compared to the others. These viscosity forces are proportional to the first power of the flow velocity. We then have:

$$\lambda \equiv \lambda(Re)$$

As the Reynolds number increases, the inertia forces, which are proportional to the velocity squared, begin to dominate. As **flow becomes turbulent**, there is a significant increase in the resistance to motion. Part of this increase is due to the roughness of the wall surface. Therefore, we have:

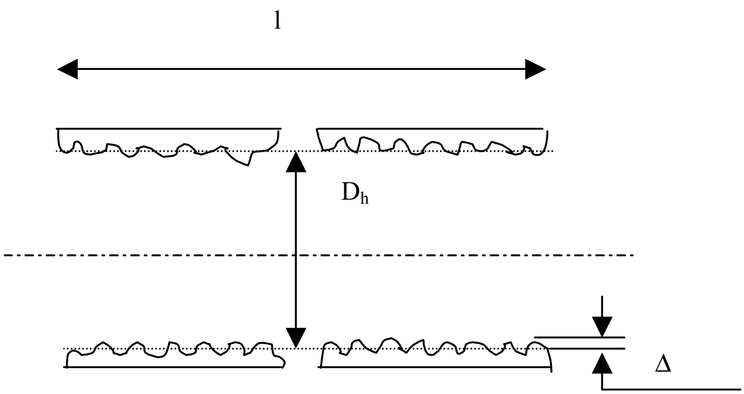
$$\lambda \equiv \lambda(Re, rr)$$

where:

rr is the relative roughness.

The relative roughness is calculated as the ratio of the average height of asperities to the tube diameter. See details in Figure 2.53 :

Figure 2.53: Relative roughness



The relative roughness of a pipe is given by:

$$rr = \frac{\Delta}{D_h}$$

where:

Δ is the equivalent uniform roughness of the pipe;

D_h is the hydraulic diameter of the pipe.

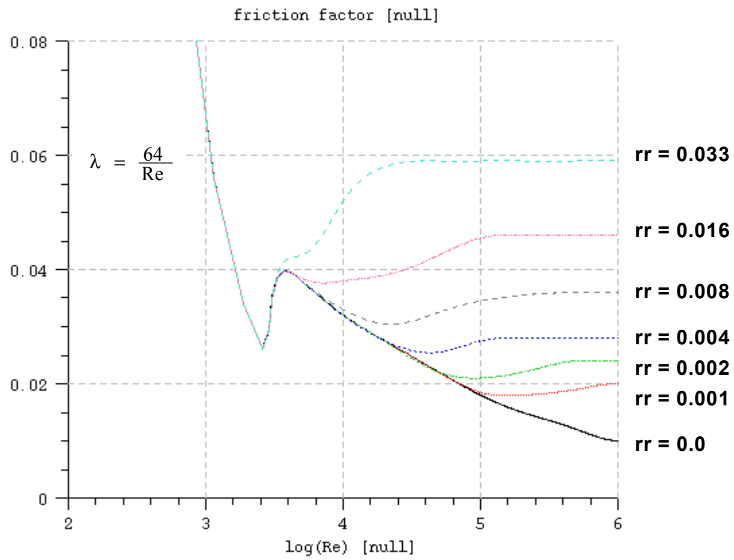
A sampling of absolute pipe roughness Δ for new clean pipes is proposed by Binder [2]:

Pipe material	Absolute roughness Δ
Drawn brass	1.5 µm
Drawn copper	1.5 µm
Commercial steel	45 µm
Wrought iron	45 µm

Pipe material	Absolute roughness Δ
Asphalted cast iron	120 μm
Galvanized iron	150 μm
Cast iron	260 μm
Wood stave	0.2 to 0.9 mm
Concrete	0.3 to 3 mm
Riveted steel	0.9 to 9 mm

The dependence of the friction coefficient λ on the Reynolds number and the relative roughness as shown in Figure 2.54 is often known as the harp of Nikuradse.

Figure 2.54: Evolution of the frictional drag factor with the Reynolds number and with relative roughness



All lines with friction in the hydraulic category use such a frictionnal drag factor.

References

[1] McCloy D, Discharge Characteristics of Servo Valve Orifices, 1968 Fluid International Conference, pp 43-50.

[2] R.C. Binder, "Fluid Mechanics". 3rd Edition, 3rd Printing. Prentice-Hall, Inc., Englewood Cliffs, NJ. 1956.

Chapter 3: AMESim Fluid Properties

3.1 Introduction

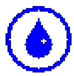
AMESim allows you to use systems with several fluids in a single sketch. For each fluid you use, you need to add a fluid property icon to your sketch.

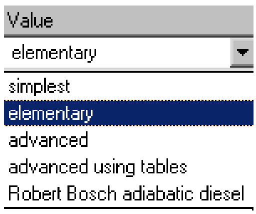
You can use each icon to install an *index of hydraulic fluid* in the range *0* to *100*. These icons give you access to a number of submodels which will now be described.

We do not describe here the *FPDROP* submodel since it is considered as obsolete and it is available only for compatibility with old systems (4.0 and earlier).

FP04

This submodel which is associated with the icon shown has an enumeration parameter which gives you access to a collection of fluid properties of varying levels of complexity.





We now describe the parameters associated with each enumeration option.

elementary

This has the following parameters:

- *index of hydraulic fluid*
- *density*
- *bulk modulus*
- *absolute viscosity*
- *saturation pressure (for dissolved air/gas)*
- *air/gas content*
- *temperature*
- *polytropic index for air/gas/vapor content*
- *absolute viscosity of air/gas*
- *name of fluid*

This option makes the following assumptions:

1. The bulk modulus of the liquid with zero air/gas content is constant. This means the corresponding density varies exponentially with pressure.
2. The viscosity of the liquid with zero air/gas content is constant.

There is an air release and cavitation model included. Note that name of fluid is a text string (e.g. “cooling water”) that identifies the fluid.

advanced

This is similar to **elementary** but there are addition parameters.

When the pressure reaches the saturation pressure of the fluid, some air/gas is released. If the pressure continues to decrease, the high saturated vapor pressure of the fluid can be reached and some vapor appears (cavitation: the liquid starts to boil). Remember the fluids used in real engineering systems are not chemically pure substances. For this reason cavitation is assumed to occur over a range of pressures and the low saturation vapor pressure is the pressure at which it is assumed that all liquid has become vapor. All these changes of state strongly modify the fluid characteristics.

With the **elementary** option this behavior is taken into account with some reasonable fixed cavitation parameters. However, with **advanced** you are allowed to set these values yourself. They are:

- *high saturated vapor pressure*
- *low saturated vapor pressure*
- *absolute viscosity of vapor*
- *effective molecular mass of vapor*

With the **elementary** option, advanced user parameters do not appear, but have the following constant values:

- high saturated vapor pressure: 0.9 bar
- low saturated vapor pressure: 0.95 bar
- absolute viscosity of vapor: 0.02 cP
- effective molecular mass of vapor: 200

advanced using tables

This option has been created to use values for the fluid density, bulk modulus and viscosity depending on the current pressure and temperature. The parameters used for this submodel are the following ones:

- *index of hydraulic fluid*
- *saturation pressure (for dissolved air/gas)*

- *air/gas content*
- *temperature*
- *polytropic index for air/gas/vapor content*
- *absolute viscosity of air/gas*
- *(advanced users) high saturated vapor pressure (cavitation)*
- *(advanced users) low saturated vapor pressure (cavitation)*
- *(advanced users) absolute viscosity of vapor*
- *(advanced users) effective molecular mass of vapor*
- *(advanced users) air/gas density at atmospheric pressure 0 degC*
- *name of fluid*
- *name of file specifying fluid properties*

Note that density, bulk modulus and viscosity do not appear in the parameters. They are calculated from values in tables processed by specific functions. These functions apply interpolation processing to calculate the fluid characteristics from tables. These tables are given in a text file specified by the name of file specifying fluid properties parameter of the submodel.

Three samples of such files are supplied in the **AMESim** installation CD:

tblprop1.txt

tblprop2.txt

tblprop3.txt

You should be able to copy these files from the directory:

$AME/misc (for Unix) or

%AME%\misc (for Windows)

Each file describes a particular **mode** of definition of the fluid properties. For the **density** and **bulk modulus**, **three modes** are available:

- In **mode 1**, density and bulk modulus are defined from a reference density, a reference pressure and a set of tables of bulk modulus values against pressure. Each table is written for a given temperature (see *tblprop1.txt*).
- In **mode 2**, density and bulk modulus are defined from a set of tables of density values against pressure. Each table is written for a given temperature (see *tblprop2.txt*).
- In **mode 3**, density and bulk modulus are defined from a reference density, a reference pressure and a set of tables of speed of sound values against pressure. Each table is written for a given temperature (see *tblprop3.txt*).

The **viscosity** of the fluid is also given in these files after the definition of the density and the bulk modulus. **Two modes** are available for the **viscosity**:

- In **mode 1** the absolute viscosity is defined from tables of absolute viscosities in cP. Each table is written for a given temperature (see tblprop1.txt).
- In **mode 2** the absolute viscosity is defined from tables of kinematic viscosities in cSt. Each table is written for a given temperature (see tblprop2.txt).

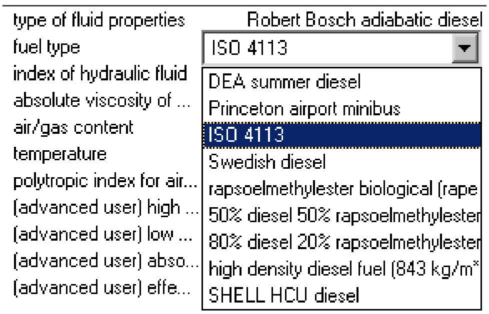
The best plan if you want to use this facility is to copy these files to a suitable local area and examine them in an editor. Lines beginning with a '#' are comments and these comments give further information on how the data is arranged. Then you can select the file that uses the modes you find suitable, and modify it in order to use your own data.

Robert Bosch adiabatic diesel

This submodel is for Diesel fluid properties and is kindly supplied by Robert Bosch GmbH. Its parameters are:

- *fuel type*
- *index of hydraulic fluid*
- *(advanced users) high saturated vapor pressure*
- *(advanced users) low saturated vapor pressure*
- *(advanced users) effective molecular mass*
- *absolute viscosity of air/gas*
- *(advanced users) absolute viscosity of vapor*
- *air/gas content*
- *temperature*

The *fuel type* is an enumeration integer parameter which gives access to 9 diesel fuels.



It is assumed that these fluids are used in fast acting injection systems and there is no time for the air content to dissolve or undissolve. The user sets a fixed temperature and the local temperature is computed using an approximate relationship for an adiabatic change.

simplest

This option gives the simplest hydraulic fluid properties. Its parameters are:

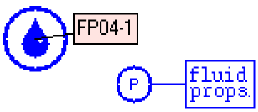
- *index of hydraulic fluid*
- *density*
- *bulk modulus*
- *absolute viscosity*
- *saturation pressure*
- *air/gas content*
- *temperature*
- *polytropic index for air/gas content*
- *name of fluid*

This submodel can be useful in difficult cases. The integrator has an easier task during cavitation and air release and so it may be possible to get a solution when other methods are unsuccessful.

3.2 Tutorial example

Copy *$AME/misc/tblprop1.txt* or *%AME%\misc\tblprop1.txt* in a suitable directory. Start **AMESim** an build the system shown in Figure 3.55 in this same directory

Figure 3.55: A simple system for plotting fluid properties.



FP04 is the only submodel available for this icon. Set the *index of hydraulic fluid* of the hydraulic submodels to 1. Change the parameters of the pressure input to get a ramp from 0 to 100 bar in 10 seconds. Change the parameter name of *file specifying fluid properties* so that it specified your own file *tblprop1.txt*.

Start a simulation and plot the density, the bulk modulus and the viscosity of the *FPROP* submodel against the pressure:

Now edit the values of your file *tblprop1.txt* and rerun the simulation. Note how the properties change.

Chapter 4: Selecting submodels for Hydraulic Lines

This problem can create a lot of worry for some users and hence in this chapter we try to give some pragmatic rules to help you select an appropriate submodel. In the formulae below it is assumes that SI units are used throughout.

4.1 Introduction

The submodels for lines are arranged with the simplest submodel at the top and the most complex at the bottom. Why are there so many line submodels?

The main problem is the complexity of flow in hydraulic pipes and hoses. The following features might be important:

- Variation of fluid compressibility and expansion of pipe/hose walls with pressure.
- Inertia of fluid.
- Variation of bulk modulus with pressure.
- Variation of viscosity with pressure.
- Laminar, turbulent and transition flow.
- Frequency dependent friction.
- Air release and cavitation.

Lumped parameter and distributed parameter line submodels

Normally it is perfectly satisfactory to use a **lumped parameter** submodel in which properties such as pressure are represented by a single representative value. In other words, within the pipe we assume there is negligible variation of pressure with position. However, if the pipe is extremely long or if wave dynamics are significant a **distributed parameter** submodel should be used. For this type of submodel, there are quantities such as pressure calculated at a number of positions. Often these are stored as an array of values.

Capacitance, resistance and inertia in line submodels

Liquids are compressible - not as much as gases but it is essential to recognize in modeling that they are compressible. When subjected to high pressure the density of a liquid increases. In addition the pipe or hose containing the liquid expands with pressure. The net result of this is a **capacitance** (spring) effect.

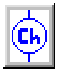
To cause a hydraulic fluid to travel along a horizontal pipe we must provide a pressure gradient to drive the fluid. This is a **resistance** effect.

The moving fluid has mass and hence it has **inertia**.

Zero-dimensional line submodels

The simplest line submodels are *DIRECT* and *HL000* and these can be described as zero-dimensional.

The *DIRECT* line submodel assumes that the two ports are very close together and the fluid and pipe in between contributes nothing.



HL000 considers the capacitance only. The length of the line is too small for significant resistance. The fluid velocity and the mass are too small to give significant inertia. The hydraulic chamber submodel *HC00* is essentially the same as *HL000*.

One-dimensional line submodels

If we take into account resistance or inertia, we have a one-dimensional (or more accurately one spatial dimension) submodel. It is possible to have 2- and 3-dimensional submodels but this is the field of computation fluid dynamics. For modeling systems one dimension is normally enough.

Until **AMESim** 4.2 all hydraulic line submodels in **AMESim** employ an implicit fomulation which was very stable. This ensures that when the system is close to equilibrium very large integration steps may be taken.

There are other classes of methods which have much more limited stability and have a strict limitation on step size due to the CFL (Courant Friedrichs Lewy) condition. **AMESim** 4.2 contains an experimental version of one of these methods. These are *HLG20*, *HLG21* and *HLG22* and implement the Godunov method. They should be used as alternatives to *HL020*, *HL021* and *HL022*. In other words for very low viscosity situations.

You can try these methods but they tend to give slow simulation runs and are less robust that the regular line submodels. They will be replaced soon by an implementation of a different method which, although still restricted by the CFL condition, is faster and more robust than Godunov.

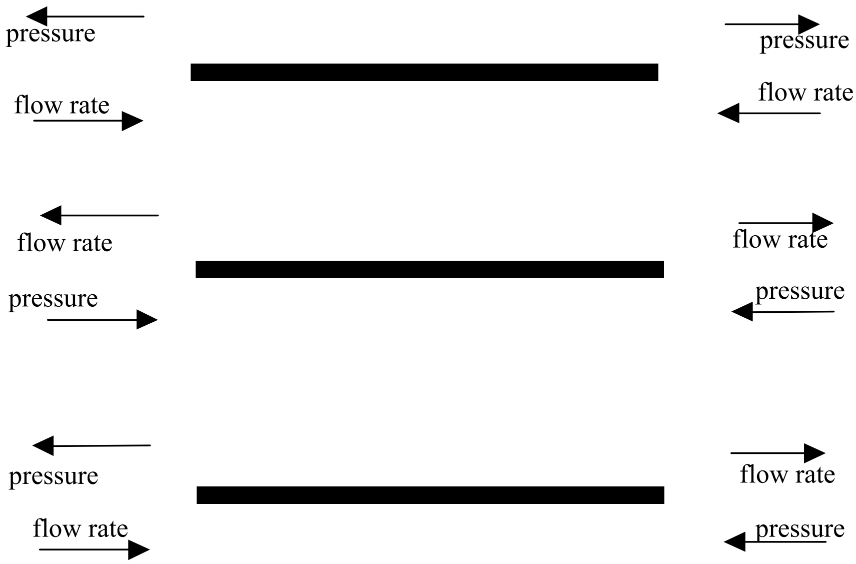
4.2 Line submodels occur in threes

AMESim line submodels normally occur in groups of three. The reason for this is the input and output characteristics of external variables of a submodel. If we connect a pipe to a component, the component submodel normally does one of two following things at the connection port:

- It calculates the flow rate (output) from the pressure (input) or
- It calculates the pressure (output) from the flow rate (input)

In each case the pipe submodel must provide the correct variable for the component submodel. Figure 4.1 shows the three standard possibilities. The arrows indicate the direction of the flow of information. Thus in the left port of the first case the flow rate is calculated by the attached component submodel whereas the pressure is calculated by the line submodel. The important feature is that most line submodels are produced in groups of three to cover the three possibilities.

Figure 4.56: Three different causalities.



4.3 Three important quantities

Aspect ratio

The checking algorithm in **AMESim** issues warning messages when you use a one-dimensional submodel that has an **aspect ratio** *length/diameter* ratio less than

6. This is defined in terms of the length L and diameter D as follows:

$$A_{ratio} = \frac{L}{D}$$

Short fat pipes require different submodels than long thin pipes.

For distributed line submodels the line is divided into a collection of cells and the test is that the cell length/diameter ratio must not be greater than 6.

Dissipation number

Another important measure is the **dissipation number**. This is defined as:

$$N_{diss} = \frac{4L\nu}{cD^2}$$

ν the kinematic viscosity and c the speed of sound.

$$c = \sqrt{\frac{B}{\rho}}$$

When the dissipation number reaches 1, the principal eigenvalues become real and wave effects are not significant. When this is true no models that take into account inertia should be used.

If the dissipation number is significant less than 1, it may be important to consider wave effects. This motivates the following table.

Line submodels	**Takes into account:**	**Dissipation number**	**Use for**
HL01 *HL02* *HL03*	Capacitance + resistance	> 0.8	relatively short pipes with high dissipation number
HL004 *HL005* *HL006*	Capacitance + resistance + inertia+ frequency dependent friction	< 1.2 and > 0.8e-3	relatively short pipes with low dissipation number
HL04 *HL05* *HL06*	Capacitance + resistance + inertia	< 1.2e-3	relatively short pipes with very low dissipation number
HL02I	Resistance + inertia		relatively short pipes with very high fluid velocity

HL10 *HL11* *HL12*	Capacitance + resistance	> 0.8	long pipes with high dissipation number
HL030 *HL031* *HL032*	Capacitance + resistance + inertia+ frequency dependent friction	< 1.2 and > 0.8e-3	moderate lengths with low but not very low dissipation number
HL020 *HL021* *HL022* *HLG20* *HLG21* *HLG22*	Capacitance + resistance + inertia	< 1.2e-3	moderate lengths with very low dissipation number

The result of this test must be qualified by considering the next important number.

Communication interval

The time taken for a wave to travel down the pipe is

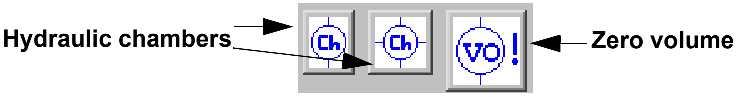
$$T_{wave} = \frac{L}{c}$$

If this time is significantly less than the **communication interval**, you will never see the waves in plots and so it is not useful to use a wave dynamics submodel. This is why changing the communication interval leads to the appearance/disappearance of warning messages.

We will compare T_{wave} with the communication interval T_{com} to determine if we are likely to 'see' waves in results.

The hydraulic volume submodels *HC00* and *HC01*, which are basically the same as *HL000*, are included for completeness. Similarly the zero volume submodel *ZEROHV* is also included.

Figure 4.57: Three other 'lines'.

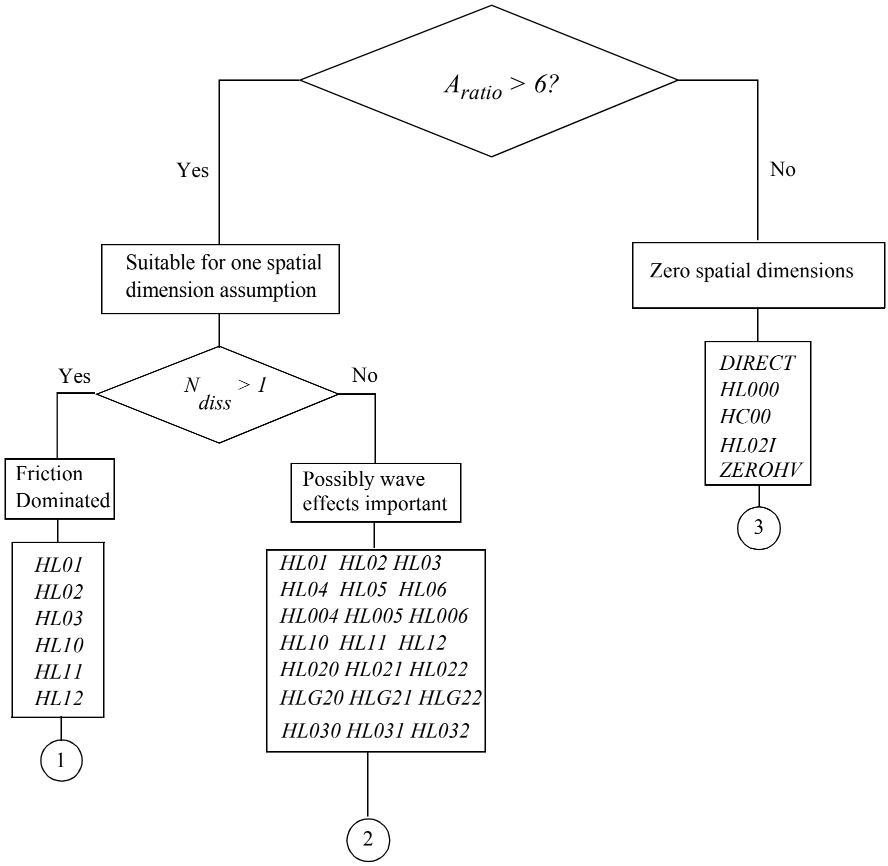


We are about to display charts which help decide which line submodel to select. These must be studied bearing in mind the following notes.

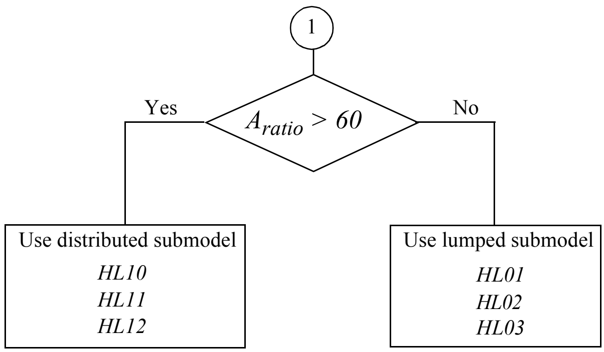
Note:

- Since many lines in **AMESim** are constituted with several segments (1, 5, 10 or 20) it can be noticed that in the below selection process the aspect ratio is compared to 6, 30 or 60 corresponding to segment of length *L*, *L/6*, and *L/10*.
- The decision process employed in the charts that follow is very similar to one employed by **AMESim** when it checks the suitability of your submodels. If the submodel is regarded as unsuitable, a warning message is issued, values of aspect ratio and dissipation number are given for one segment of the choosen line.
- Often the final result from the chart is three submodels such as *HL01*, *HL02*, *HL03*. Since **AMESim** will check causality, only one submodel, the one that is compatible with adjoining submodels, will be offered to you as a choice.
- The charts are intended for general guidance and give a good choice most of the time. However, there are circumstances in which an advanced users may wish to break these rules.

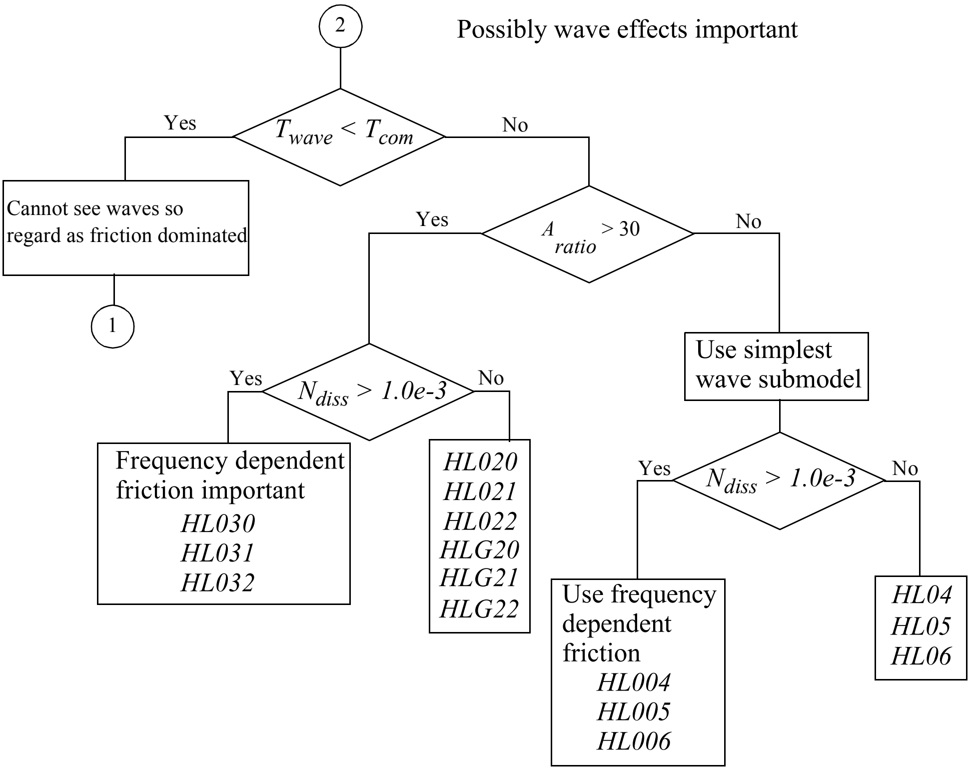
4.4 The selection process



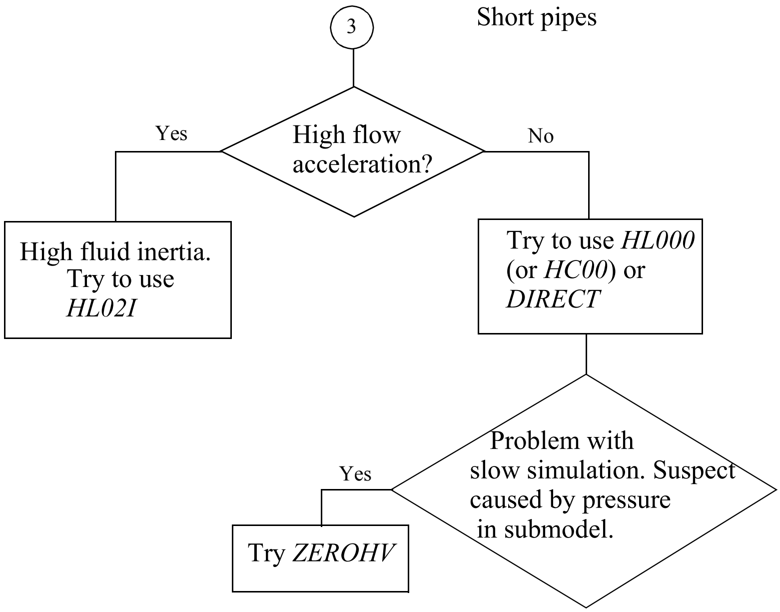
Friction dominated pipes and hoses



Possibly wave effects important



Short pipes



Index

G

H

I

L

P

R

S

T

V

W

Reporting Bugs and using the Hotline Service

AME is a large piece of software containing many hundreds of thousands of lines of code. With software of this size it is inevitable that it contains some bugs. Naturally we hope you do not encounter any of these but if you use **AME** extensively at some stage, sooner or later, you may find a problem.

Bugs may occur in the pre- and post-processing facilities of **AMESim**, **AMERun**, **AMESet**, **AMECustom** or in one of the interfaces with other software. Usually it is quite clear when you have encountered a bug of this type.

Bugs can also occur when running a simulation of a model. Unfortunately it is not possible to say that, for any model, it is always possible to run a simulation. The integrators used in **AME** are robust but no integrator can claim to be perfectly reliable. From the view point of an integrator, models vary enormously in their difficulty. Usually when there is a problem it is because the equations being solved are badly conditioned. This means that the solution is ill-defined. It is possible to write down sets of equations that have no solution. In such circumstances it is not surprising that the integrator is unsuccessful. Other sets of equations have very clearly defined solutions. Between these extremes there is a whole spectrum of problems. Some of these will be the marginal problems for the integrator.

If computers were able to do exact arithmetic with real numbers, these marginal problems would not create any difficulties. Unfortunately computers do real arithmetic to a limited accuracy and hence there will be times when the integrator will be forced to give up. Simulation is a skill which has to be learnt slowly. An experienced person will be aware that certain situations can create difficulties. Thus very small hydraulic volumes and very small masses subject to large forces can cause problems. The *State count* facility can be useful in identifying the cause of a slow simulation. An eigenvalue analysis can also be useful.

The author remembers spending many hours trying to understand why a simulation failed. Eventually he discovered that he had mistyped a parameter. A hydraulic motor size had been entered making the unit about as big as an ocean liner! When this parameter was corrected, the simulation ran fine.

It follows that you must spend some time investigating why a simulation runs slowly or fails completely. However, it is possible that you have discovered a bug in an **AMESim** submodel or utility. If this is the case, we would like to know about it. By reporting problems you can help us make the product better.

On the next page is a form. When you wish to report a bug please photocopy this form and fill the copy. You telephone us, having the filled form in front of you means you have the information we need. Similarly include the information in an email.

To report the bug you have three options:

- reproduce the same information as an email
- telephone the details
- fax the form

Use the fax number, telephone number or email address of your local distributor.

HOTLINE REPORT

Creation date:	*Created by:*
Company:	*Contact:*

Keywords (at least one):

Problem type: ☐ **Bug** ☐ **Improvement** ☐ **Other**

Summary:

Description:

Involved operating system(s):

☐ **All**	☐ **Unix (all)**	☐ **PC (all)**
	☐ **HP**	☐ **Windows 2000**
	☐ **IBM**	☐ **Windows NT**
	☐ **SGI**	☐ **Windows XP**
	☐ **SUN**	☐ **Linux**
	☐ **Other:**	☐ **Other:**

Involved software version(s):

☐ **All**	☐ **AMESim (all)**	☐ **AMERun (all)**	☐ **AMESet (all)**	☐ **AMECustom (all)**
	☐ **AMESim 4.0**	☐ **AMERun 4.0**	☐ **AMESet 4.0**	☐ **AMECustom 4.0**
	☐ **AMESim 4.0.1**	☐ **AMERun 4.0.1**	☐ **AMESet 4.0.1**	☐ **AMECustom 4.0.1**
	☐ **AMESim 4.0.2**	☐ **AMERun 4.0.2**	☐ **AMESet 4.0.2**	☐ **AMECustom 4.0.2**
	☐ **AMESim 4.0.3**	☐ **AMERun 4.0.3**	☐ **AMESet 4.0.3**	☐ **AMECustom 4.0.3**
	☐ **AMESim 4.1**	☐ **AMERun 4.1**	☐ **AMESet 4.1**	☐ **AMECustom 4.1**
	☐ **AMESim 4.1.1**	☐ **AMERun 4.1.1**	☐ **AMESet 4.1.1**	☐ **AMECustom 4.1.1**
	☐ **AMESim 4.1.2**	☐ **AMERun 4.1.2**	☐ **AMESet 4.1.2**	☐ **AMECustom 4.1.2**
	☐ **AMESim 4.1.3**	☐ **AMERun 4.1.3**	☐ **AMESet 4.1.3**	☐ **AMECustom 4.1.3**
	☐ **AMESim 4.2**	☐ **AMERun 4.2**	☐ **AMESet 4.2**	☐ **AMECustom 4.2**

Web Site
http://www.amesim.com

Headquarter & Development Center

IMAGINE S.A. Roanne

Tel: +33 4-77-23-60-30
Fax: +33 4-77-23-60-31
E-mail: hotline@amesim.com

FRANCE - SWITZERLAND - SPAIN - PORTUGAL - BENELUX

IMAGINE S.A. Paris

Tel: +33 1-39-43-08-12
Fax: +33 1-39-43-52-19
E-mail: hotline@amesim.com

ITALY - SWITZERLAND

IMAGINE S.A. Lyon

Tel: +33 4-37-69-72-30
Fax: +33 4-78-54-39-61
E-mail: hotline@amesim.com

UK

IMAGINE U.K.

Tel: +44 (0) 1869 351 994
Fax: +44 (0) 1869 351 302
E-mail: support.uk@amesim.com

GERMANY - AUSTRIA - FINLAND - DENMARK - NETHERLANDS - NORWAY - SWEDEN - SWITZERLAND - EASTERN EUROPE

IMAGINE Software GmbH

Tel: +49 (0) 89 / 548495-35
Fax: +49 (0) 89 / 548495-11
E-Mail: hotline.germany@amesim.com

HUNGARY

Budapest University of Technology & Economics

Tel: (36) 1 463 4072 / 463 2464
Fax: (36) 1 463 3464
E-Mail: vad@simba.ara.bme.hu

NORTH AMERICA

IMAGINE Software, Inc.

Tel: (1) 734-207-5557
Fax: (1) 734-207-0117
E-Mail: support.us@amesim.com

SOUTH AMERICA

KEOHPS Ltd

Tel: (55) 48 239 – 2281
Fax: (55) 48 239 – 2282
E-Mail: info@keohps.com

JAPAN

IMAGINE Japan K.K.

Tel : +81 (0) 3 3351 9691
Fax : +81 (0) 3 3351 9692
E-mail: imagine-japan@amesim.com

CHINA

IMAGINE China

Tel: + 86 21 34 12 34 58
E-mail: qing@amesim.com

United Right Technology

Tel: (86) 10-67082450(52)(53)(54)
Fax: (86) 10-67082449
E-Mail: urt@urtgroup.com

SOUTH KOREA

SHINHO Systems Co., Ltd

Tel: +82 31 608 0434
Fax: +82 31 608 0439
E.Mail: iclee@shinho-systems.co.kr

ISRAEL

Tel : +972 3534 4432
Fax : +972 3535 5514
E-mail: fempp@netvision.net.il