Interlink Systems, Inc.
5116 Bissonnet #341, Bellaire, TX 77401
Telephone and Fax: (713) 663-6361
jamesmcadams@alumni.rice.edu

Control Simulator 8
Technical Reference

A discussion of all input parameters and what they mean.

James P. McAdams, PE
Interlink Systems, Inc.
Texas Registered Engineering Firm F-8069

Rev.	Date	Description	Origin.	Check.	Approv.
09	32 July 2015	Documented the new feature allowing multiple source (input) lines on manifolds and trees. Went over that whole section of the manual making clarifications and documenting some trigger functions that were not documented earlier. The glossary was extended. Many small typeos were fixed. Several new features that had not been documented are documented now.	JPM		
08	21 Mar 2015	Clarified the wording of reservoir handling in the HPU.	JPM		
07	16 Mar 2015	Documented the option for sorting the case name dialog. Added description of valve Repeat Count and Period. Documented the pressure vs. time option for HPU output. Added a technical Notes section for stepping chokes.	JPM		
06	20 Nov 2013	Added the Technical Notes section. Added some details to the Timing Device description.	JPM		
05	24 Apr 2012	Small type-o's.	JPM		
04	8 Oct 2011	Changed valve spring curve description to include uneven point spacing. Added description of a pressure function for the orifice discharge.	JPM		
03	11 Mar 11	Documented new features: clone a valve in a valve module, rearrange valves in a module.	JPM		
02	5 Mar 2011	Expanded to include all components and devices.	JPM		
01	2 Nov 2010	Initial version	JPM		

TABLE OF CONTENTS

1. INTRODUCTION

This technical manual describes all of the input parameters for the various library components and case devices in CS8 including limitations and special considerations. If you have questions not answered by this manual, or find errors in the manual, please contact Interlink Systems so that I can add explanations to cover your questions or correct the errors.

2. MAIN MENU

Across the top of the main program is the main menu. Many of these menu items function just as you would expect in a Windows program. But even so, I will list the actions of the menu items here.

2.1 File

2.1.1 New Project

Create and open a new, blank project file for editing. It will have an empty library and no cases. It will be ready to edit. The project window must be blank (no open project) in order to create a new project.

2.1.2 Open Project

Open an existing project for viewing and editing. It will be opened in the state it was in the last time it was closed. It will be ready to view and/or edit. The project window must be blank (no open project) in order to create a new project.

2.1.3 Close Project

Close the project file and leave the program with a blank project window. If Autosave is enabled, the file will be automatically saved before closing. Otherwise you will be given the opportunity to save the file.

2.1.4 Save Project

Save the latest copy of the project file. If this is a new file that has never been saved, Save will actually do a SaveAs. (see section 2.1.5)

2.1.5 Save Project As

Save the project, but give the operator an opportunity to save it with a different name. This does not necessarily do a Save of the original file.

2.1.6 Exit Program

Close the file and terminate program execution. You will get an opportunity to save the file.

2.2 View

The project Manager and the Project Toolbar can be minimized, in which case, they appear as small icons at the bottom of the program main screen. You can never get rid of them completely while a project is open.

If they are minimized, you can bring them back up to normal size by using View and then selecting the window you want to restore to normal size. You may also use the mouse and click on the small restore icon on the minimized window (see the figure at the right).

2.3 Case

The Case main menu option is only visible if you have a project open.

2.3.1 New

Create a new, blank case and open its window in the project area for editing.

2.3.2 Open

Open an existing case for viewing and editing.

2.3.3 Rename

To rename a case, open the case that you want to rename. Then click anywhere in the case window you want to rename so that its title bar is highlighted. Then select Rename and use the dialog box that pops up to select a new name for the case. Rename creates a clone with the new name and then deletes the original.

2.3.4 Clone

Clone creates a new case that duplicates all of the devices and connections that are in the selected open case. To clone a case, open the case that you want to clone. Then click anywhere in that case window so that its title bar is highlighted. Finally, click Clone and use the dialog box that pops up to select a new name for the clone. The original case is left unchanged.

2.3.5 Delete

Delete deletes the currently selected, open case. To delete a case, open the case that you want to delete. Then click anywhere in that case window so that its title bar is highlighted. Finally, click Delete the case will be deleted. ***Caution: There is no way to reverse a deletion other than entering it all over.***

2.3.6 Delete Via Case List

Some users build projects with zillions of cases. Deleting them one by one is a royal pain. This menu selection brings up a list of all cases in the project. You use the mouse click in combination with the ctrl or Shift keys to highlight one or more cases. You then use the Delete button **to delete them ALL. All at once. For good. No ctrl-Z. Be careful**.

2.3.7 Copy to Clipboard

Copy to Clipboard copies the case window image on the screen to the Windows clip board. You can then paste the image into Word documents, emails, photo editors, etc.

2.4 Options

2.4.1 Autosave

If Autosave is checked, then your project will be saved once per minute, just before a case is run, and before closing a case or the project. You will not be asked for permission to save.

If Autosave is not checked, then a Save will only occur when you do it manually from the File menu item, or when you try to close a case or the project.

2.4.2 Show Device Names

In a case, each device has a name. You can display the names of the devices above the device icon, or you can turn off the display of the names using this selection.

2.4.3 Sort Case Names

When you click the Case main menu item and select New, Open, or Delete by Case List, you get a dialog box showing all of the cases in the project. If Sort Case Names is checked, the cases will be in alphabetical order. If not checked, the list will be in the order that the cases were created. Alphabetical is selected unless you use “Options / Sort the case name list” to change it.

2.5 Help

2.5.1 CS8 Technical Manual

This uses Adobe Reader to open the file you are reading now. This selection looks for a file named "CS8 Technical Manual.pdf" in the same folder where the executable CS8.exe is located.

2.5.2 CS8 Quick Start

This uses Adobe Reader to open the quick start manual, which contains tips on setting up a simple project. This selection looks for a file named "CS8 Quick Start.pdf" in the same folder where the executable CS8.exe is located.

2.5.3 About CS8 Interface

This displays the revision number and date of the graphical user interface of CS8.

2.5.4 About CS8 Compute Engine

This displays the revision number and date of the compute engine portion of CS8. The compute engine is a file named CS8engine.exe. It is interesting to note that the compute engine is an independent executable. If you start it on its own, say by double clicking CS8engine.exe, it will execute and let you open case files to run. This would be useful if you want to write your own case files or write some sort of batch routine that modifies a case file and runs it over and over.

3. LIBRARY COMPONENTS

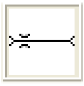
You build up cases for simulations by using library components in conjunction with source, line and discharge devices. You must define components for the library before you can build up cases for simulation. This section covers building library components using the Project Manager.

3.1 Project Comments

At the top of the Project Manager window is the selection to Manage project comments and engineering units. Double click the icon that looks like a sheet of school notebook paper. The primary purpose of this is to set default output units of measure for all calculated quantities. The units selected here are used throughout the project unless a particular device has its own selection of output units.

You may right click individual units to change them. You can also click the All Metric, or All English buttons to set up a set of default units.

3.2 Resistance Element

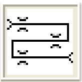


This is the simplest fluid resistance device in CS8. It consists of a simple orifice element in series with a section of tubing. You must have at least one tubing or hose defined before you can define a resistance element.

Property	Description
Name	Name or identifier for this component. The name is used in cases to select which library component is to be used. Each component of this type in the library must have a unique name. NOTE: changing the name creates a new copy of the component. Choose the name carefully because changing it later is tedious. You have to create a new one, find and change all places the old one is used, and change them all to the new name.
Cv Forward	This defines a flow coefficient for the simple orifice element for flow in the forward direction (normally left to right on the screen).
Cv Reverse	This defines a flow coefficient for the simple orifice element for flow in the reverse direction (normally right to left on the screen).
Tubing/Hose name	Select type of conduit and then select the desired component from the library. The selected library tube or hose sets the internal diameter and internal surface roughness for the tube in this resistance element.

Total length	This is the length of the tube. It can be zero, in which case, the resistance comes only from the orifice element. If the length is not zero, then pressure drop in the tube is computed using normal steady state conditions for laminar, transitional, or turbulent flow, as appropriate.
90 degree bends	This allows you to approximate the pressure drop due to bends and obstructions in the tube. It is based on rough estimates of K factor for valves and fittings as defined in Crane[1] page A29. K is based on a multiple of f_T (friction factor in full turbulent flow) with K for a 90° bend being $30f_T$. The number of 90° bends may include fractions. Ex: entering 2.5 will cause the program to add a resistance of $75f_T$ in series with the orifice and the tube. Click the Help with Bends button for suggestions on other components.

3.3 Resistance Path

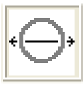


A resistance path is a collection of one or more Resistance Elements in series. The resistance path is a convenient way to define resistances out of commonly used elements. You must have at least one resistance element defined before you can define a resistance path.

[1] Flow of Fluids Through Valves, Fittings, and Pipe, Crane Technical Paper No./ 410, 1981, Crane Co.

Property	Description
Name	Name or identifier for this component. See the description of the name for the Resistance Element in section 3.1 for more details and the effect of changing the name.
Add an Element	To add elements to this path, select the element from the library using the Element to Add pull down list (the element must already be defined in the library). Then press the Add button as many times as you wish.
Delete an Element	Click the element in the list that you want to delete. When the element is highlighted, press the Delete button.
Move Up and Move Dn	To change the position of an element in the list, click the element in the list (to the right of the Move Up button) that you want to move. When the element is highlighted, press the Move Up or Move Dn button to move the element as desired. Note, the pressure drop in the Resistance Path does not depend on the position of the elements in the list; this is only for your visual satisfaction.

3.4 Tubing



Tubing refers to relatively stiff conduit with linear elastic expansion when under pressure. Tubing is used in long umbilicals and can be used to define Resistance Elements.

Property	Description
Name	Name or identifier for this component. See the description of the name for the Resistance Element in section 3.1 for more details and the effect of changing the name.

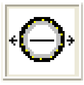
3.4.1 Dimensions Tab

Property	Description
Outer Diameter (OD)	Nominal outer diameter of the tube including the walls.
Wall	Thickness of the wall. Internal diameter is OD – 2*Wall
Roughness of internal wall	This is the absolute roughness of the internal surface of the tubing.
Young’s modulus	This is the modulus of elasticity of the pipe wall material. For tubing it is assumed that the stretch of the wall is elastic and linear.
Poisson’s ratio	This is used in computing the speed of sound in the tube. For steel 0.3 is typical.

3.4.2 Comments Tab

The comments section can be used for any purpose you find useful.

3.5 Hoses



Hoses are conduits that have relatively large expansion and the expansion is nonlinear and time dependent (it creeps). Virtually any conduit that has plastic elements is a hose by this definition.

The creep of the hose wall refers to a viscoelastic deformation of the plastic and is described by several parameters. When you apply pressure to a hose, it expands a certain amount immediately (elastic expansion). It then expands some more over time (viscoelastic expansion). The viscoelastic expansion is modeled by a fast and a slow part, each having its own time constant. These various elements are explained quite well in a paper that experimentally examined the effects[2].

Property	Description
Name	Name or identifier for this component. See the description of the name for the Resistance Element in section 3.1 for more details and the effect of changing the name.

3.5.1 Dimensions Tab

Property	Description
Internal Diameter (ID)	Nominal inner diameter of the hose from inside wall to inside wall.
Internal wall roughness	This is the absolute roughness of the internal surface of the hose.
Fast time constant	This describes the speed of the fast viscoelastic expansion component. Unless you know better, 35 sec is a reasonable value for common hoses.
Slow time constant	This describes the speed of the slow viscoelastic expansion component. Unless you know better, 400 sec is a reasonable value for common hoses.

[2] *The Dynamic Response of Thermoplastic Hoses*, P. S. McCarthy and P. H. Knight, Umbilicals – The Future, Proceedings of an international conference organized by the Society for Underwater Technology, London, 14 December, 1995, pp 35-49.

Ratio of elastic to total expansion	This is the portion of expansion that is elastic (that occurs immediately). This will always be less than 1 for a thermoplastic hose. This portion of the expansion determines the speed of sound in the hose. Unless you know better, 0.4 seems to be a good number for common hoses. If you are not sure, note the calculated speed of sound. Typical hoses run from a minimum of 600 to 1000 ft/sec at low pressure to a high of 2500 to 3000 ft/sec at high pressures.
Ratio of fast to total viscoelastic expansion	The portion of viscoelastic expansion that is fast. Unless you know better, 0.55 seems to be a good number for common hoses.

3.5.2 Expansion Table Tab

The expansion table defines the way the internal volume of the hose varies with internal pressure. Measured correctly, the data you normally get from hose manufacturers is the total expansion, viscoelastic plus elastic. That is what CS8 expects to be entered into this table.

Zero pressure is assumed to result in zero expansion. Do not enter a point for zero. Pick enough points on the expansion curve to describe it accurately and enter the pressure/expansion values into the table in order, from low pressure to high, as you go down the table.

To enter a well-behaved table, first, look at your hose data and consider the following. Logically, the expansion should increase smoothly with pressure. It should not have bumps or wiggles or any place where the slope is down rather than up. You may want to draw a smooth curve through the available data and use points from your curve rather than entering test results directly.

If you have more than 8 points, use the scroll bar on the right to access parts of the table that are not visible.

Finally, include data in your table that covers the entire pressure range that will be used in the simulations. The program will extrapolate to higher pressures, but that may not be the right thing to do for a given hose.

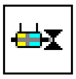
Property	Description
Unit of measure	Right click the boxes at the top of the pressure and expansion columns to select the unit of measure to be used in the column.

Add a point	Click the Add button to add a blank point at the bottom of the table (high pressure end). Then enter the pressure and expansion for the new point.
Insert a point	Click the line in the table where you want to insert the new point. Then click the Insert button. The line you clicked (and all lines below it) will move down, opening up a blank point for you to fill in.
Delete a point	Click the line in the table that you want to delete. Then click the Delete button. The line that was highlighted will be deleted and all lines below it in the table will move up.
Move up and Move Dn	To change the position of a point in the table, click the point that you want to move. When the point is highlighted, press the Move Up or Move Dn button to move the point as desired. Other points will move as needed.
Error check	Click this at any time to find mistakes such as pressures that do not increase as you go down the table, blank lines, etc. This same error check is called when you press Save.

3.5.3 Comments Tab

The comments section can be used for any purpose you find useful.

3.6 Valve

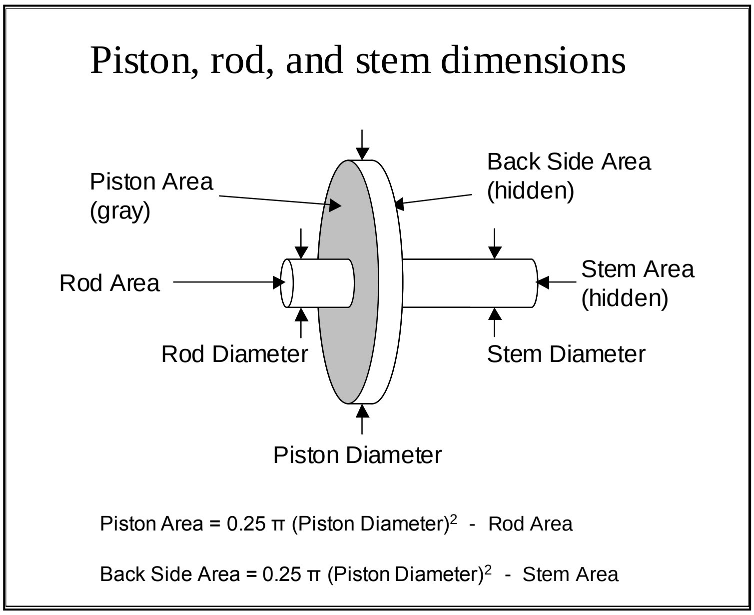


The valve is a very general-purpose device that can be used for gate valves, ball valves, BOP rams, and many other types of piston driven devices. By manipulating the volumes, pressures and forces, you can mimic a wide variety of functions.

Property	Description
Name	Name or identifier for this component. See the description of the name for the Resistance Element in section 3.1 for more details and the effect of changing the name.

3.6.1 Valve – Dimensions Tab

The following figure shows the relationships between areas and diameters in the valve model.



IMPORTANT NOTE: If the Stroke field is entered as a linear measurement (such as inches), then the Piston Area must be correct for the valve. This is because the stroke and area of the piston are used to compute the total volume of control fluid needed to operate the valve. Entry of the Stroke as a volume makes it independent of the Piston Area.

Property	Description
Dp (or Ap)	Diameter or effective wetted area of the inlet side of the piston. Pressure on the inlet side of the piston tends to compress the spring and move the valve toward 100% of stroke. Be careful with this, because the interpretation of an area is quite different from the interpretation of a diameter. See the figure above for a detailed diagram and explanation.
Dr (or Ar)	Diameter or cross sectional area of the rod. The rod is acted upon by the sea static pressure. This area can be zero but must be less than Dp.
Ds (or As)	Diameter or cross sectional area of the stem. The bore pressure acts upon the stem. This value can be zero but must be less than Dp.
Cv	This is the flow coefficient of a leak path around the piston. Normally this will be zero. The most common use of this leak path is for failure modes and hydraulic motors.
Stroke (or volume)	Enter here the total inlet fluid volume or the total linear piston motion required to move the piston completely from 0% to 100% of stroke.
Stroke (or volume, or %) subject to high friction	Between 0% of stroke and some higher percent of stroke (typically about 15%), a gate valve can be subject to much higher friction than when the gate is open. Enter here the portion of the stroke subject to this increased friction. If entered as a percent, it is taken to be percent of the total stroke.

3.6.2 Forces Tab

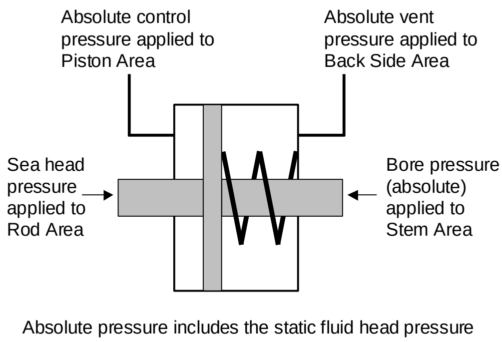
IMPORTANT NOTE: On this tab there are 5 friction and load fields for which the entry may be a pressure or a force. If any of them is entered as a force, the Piston Area must be correct. This is because the piston area and force applied are used internally for computing a pressure. If all are entered as pressures, then the simulation forces are independent of Piston Area.

Property	Description
Reference bore pressure	There are two kinds of friction defined for the valve model: sticking and Coulomb (running). Both are assumed to be directly proportional to absolute bore pressure. Minimum bore pressure is fixed at 0 psi (you cannot change it). Enter in the upper field an absolute reference bore pressure at which the maximum sticking and Coulomb frictions are valid. Note: if the Reference Bore Pressure is entered as 0 psi, then interpolation is impossible and the upper Coulomb and sticking friction are used.
Sticking friction	This friction is used by the model when the piston is in the portion of the stroke subject to high friction. In this field, enter the total friction in the mechanism when the Reference Bore Pressure (see Valve – Dimensions Tab) is present (maximum sticking). At zero bore pressure, the Sticking friction is always the same as the lower Coulomb friction. Sticking friction for the Reference Bore Pressure must be greater than or equal to the upper Coulomb Friction.
Coulomb (running) friction	The Coulomb friction is the normal friction that is always present in the valve. In the case of a gate valve, this is the friction with no pressure drop across the gate. Enter in the lower field the Coulomb friction for 0 bore pressure. Enter in the upper field the Coulomb for the Reference Bore Pressure. The upper field must be greater than or equal to the lower field.

Full Load	This is the load that must be applied to position the piston just barely to 100% of stroke. Normally, this is the load required to completely compress the spring. This only includes the spring. Do not include friction or any other forces.
Preload	This is the load that must be applied to the piston to position it barely to 0% of full stroke. Normally, this is the load required to just begin compressing the spring. This only includes the spring. Do not include friction or any other forces.
Stroke (or volume)	Enter here the total inlet control fluid volume or the total linear piston motion required to move the piston completely from 0% to 100% of stroke.
Preload and full load are already at depth. Disregard control head and water depth.	Check this box if the preload, full load, and all frictions are already corrected for depth. The valve model will only be valid for the specified depth. See diagrams at the end of this section to see how pressures are applied in this case.
Preload and full load already include bore pressure effects. Disregard bore pressure.	Check this box if the preload, full load, and all frictions are already corrected for bore pressure. The valve model will only be valid for the specified bore pressure and any bore pressure you enter in the model will be ignored and exert no force on the stem. Note: if this box is checked, then the Upper Coulomb friction value is used and bore pressure is ignored in calculating friction. See diagrams at the end of this section to see how pressures are applied in this case.

The effects of the check boxes to disregard depth and bore pressure are illustrated in the figures below. The normal, complete model is shown first.

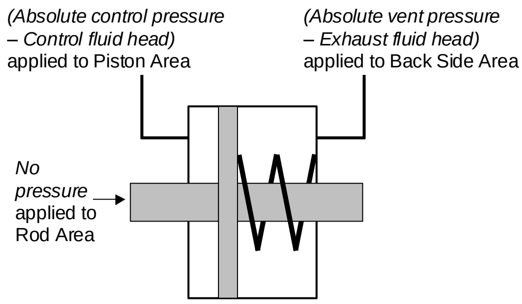
Pressure Forces on the Valve
Complete Model



In the complete model, the absolute fluid pressures exert forces on the piston, stem and rod. Absolute control pressure includes the control fluid head and any other applied pressures or drops in the supply system. The absolute vent pressure includes the static head on the vent system (which may be sea water or control fluid) and any other pressures due to flow in the vent system.

If the Disregard Depth box is checked, the only force on the piston from sea or control fluid comes from the gauge pressures on each side of the piston. This force is the same at any depth. Ex: a 5000 psi supply will apply a force of 5000 psi*Piston_Area to the piston, regardless of depth or vent fluid head.

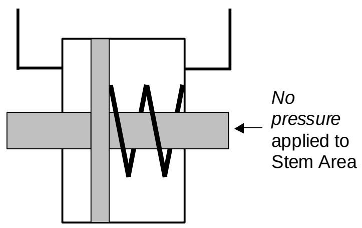
Pressure Forces on the Valve
Changes when "Disregard Depth" is checked



Note that pressure drops in the supply and vent systems still affect the piston. For instance, a restricted vent system can still apply a force to the back side of the valve pistons while there is flow in the vent.

If the Disregard Bore Pressure box is checked, then there is no force applied to the stem, regardless of the bore pressure entered into the model.

Pressure Forces on the Valve
Changes when "Disregard Bore Pressure" is checked



Note, that friction is affected by bore pressure. If this box is checked, then the model uses the upper friction values (the ones associated with the reference bore pressure).

3.6.3 Spring Curve Tab

If the spring is nonlinear, use this tab to approximate the curve of spring force vs position. As you fill in the table the plot shows you the resulting curve.

"Fraction of Stroke" denotes a set of 2 to 11 points along the stroke of the piston where you can define the spring force. You may put points at varying intervals, but the fraction of Stroke must increase as you go down the list. A sudden change in force requires two points with Fraction of Stroke values that are close together (like 0.4 and 0.405) but they may not be equal.

"Fraction of Force" is a factor that determines the spring force at that location. It is a linear scaling factor from 0 to 1 that is applied to full load and preload such that 0 = Preload and 1 = Full load. It works like this:

Spring Force = (Full Load – Preload) * (Fraction of Force) + Preload

For a normal, linear spring, Fraction of Force = Fraction of Stroke.

An annular blowout preventer is an example of a case where a nonlinear spring can be useful. I have also used it to simulate the cutting of pipe with a blind shear ram.

3.6.4 Comments Tab

The comments section can be used for any purpose you find useful.

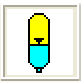
3.6.5 Special Cases

3.6.5.1 Spring acts in the same direction as control pressure

In the case where the spring is acting to aid the hydraulic pressure, enter a large magnitude negative force for the preload, and a smaller magnitude negative full load.

Ex: Preload = -750 psi Full Load = -500 psi

3.7 Accumulators



The accumulator component can be used for any kind of gas filled accumulator that is precharged with nitrogen or helium. The accumulator model can be adiabatic, nearly isothermal, or anything in between. The accumulator model uses a very accurate gas law, good up to about 15,000 psia and any temperature normally encountered in offshore use.

Property	Description
Name	Name or identifier for this component. See the description of the name for the Resistance Element in section 3.1 for more details and the effect of changing the name.

3.7.1 Parameters Tab

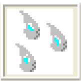
Property	Description
Select the precharge gas	Use this pull down box to select the gas used to precharge the accumulator. You will notice that nitrogen and helium appear twice in the list. This affects the gas law that is used. NIST is is the more accurate but is slower to execute. RK (Redlich-Kwong) is pretty good accuracy and faster. In terms of density for a given pressure and temperature, NIST is within about 0.6%, whereas RK is within 2 to 5%. "Ideal" is in this list. There is no good reason to use an ideal gas unless you are comparing results to a hand calculation. Real gasses are not ideal.
Precharge gas volume	When the accumulator is "empty" this is the volume occupied by the precharge gas. Note that in some cases, you might precharge the accumulator when it is truly empty of fluid, but then in operation a valve closes to stop the motion of the bladder or float before the accumulator is actually empty. In this case, use the smaller, limited volume and adjust the precharge pressure accordingly. CS8 only knows about one "empty" volume and that should be the volume at which it cannot supply more fluid.
Gas law exponent (Cp/Cv)	This selects the ratio of specific heats (Cp/Cv) for the gas. Normally you should check the box to use the built in tables. When the table is in use, CS8 tracks the pressure and temperature in the bottle and adjusts Cp/Cv as appropriate. To enter your own constant value of Cp/Cv, uncheck the box and enter your value in the box that appears. There is normally no good reason to do this except to compare to a hand calculation.

Thermal time constant	This determines how fast heat is transferred into and out of the accumulator. You can force the accumulator to be adiabatic by checking the adiabatic box. You cannot make the accumulator truly isothermal, but entering your own small time constant like 1-2 seconds will approximate an isothermal condition. The number is complex in its meaning and not much data is available. In general, it can be thought of as a time for the temperature to return back to ambient. That is a qualitative statement. Quantitatively, the action is more complex. Going on available test data, for short stubby accumulators (length/dia = 4-6), a value of 40-45 sec is reasonable. Long skinny ones (length/dia = 20-30) is more like 15 sec. I do not know of a good way to calculate this number.

3.7.2 Comments Tab

The comments section can be used for any purpose you find useful.

3.8 Fluids



Fluids in CS8 can be any Newtonian fluid. Viscosity is allowed to vary with pressure and temperature.

Property	Description
Name	Name or identifier for this component. See the description of the name for the Resistance Element in section 3.1 for more details and the effect of changing the name.

3.8.1 Parameters Tab

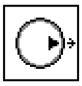
Property	Description
Bulk modulus	Defines the compressibility of the fluid. Enter the bulk modulus at atmospheric conditions.
Specific gravity or density	This defines the density of the fluid at atmospheric conditions. This can be entered as a specific gravity, a mass density, or a weight density.

Viscosity at the indicated temperature and pressure	Viscosity is defined by 4 points representing two different pressures, and two different temperatures. The temperature for a given viscosity point is determined by the row it is in, while the pressure is determined by the column. If you only know one viscosity, enter it in all four places. If you only know how the viscosity changes with temperature, then make the two columns the same.
Low Pressure	At the top of this column, enter the pressure at which the viscosities in the left column are valid.
High Pressure	At the top of this column, enter the pressure at which the viscosities in the right column are valid.
Low Temperature	At the left end of this row, enter the temperature at which the viscosities in the top row are valid.
High Temperature	At the left end of this row, enter the temperature at which the viscosities in the bottom row are valid.

3.8.2 Comments Tab

The comments section can be used for any purpose you find useful.

3.9 Pumps



The pump is a simple device that can approximate electric and air pumps. Internally it is implemented as a pressure source in series with a flow restriction. By manipulating the pressure and restriction, a variety of output curves can be approximated.

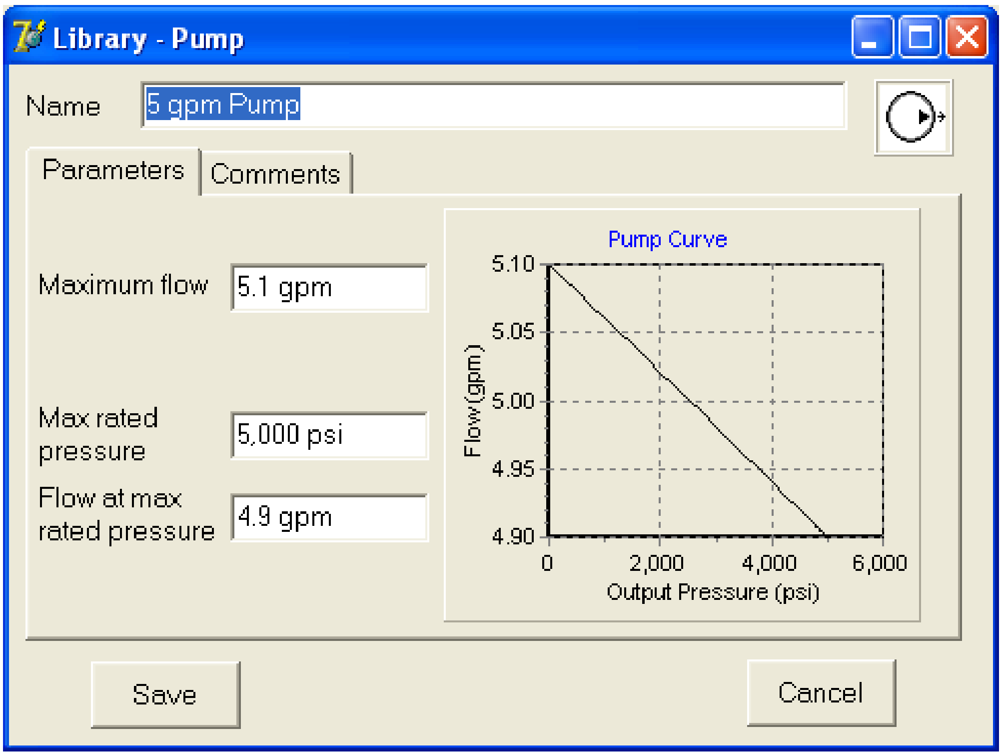
Property	Description
Name	Name or identifier for this pump. See the Valve component for more details on the name.

3.9.1 Parameters Tab

Property	Description
Maximum flow	This is the full flow that will occur if there is no pressure or restriction on the outlet of the pump.
Max rated pressure	This is an arbitrary upper pressure for defining the load curve of the pump. It will usually be the rated pressure of the pump. The model may be able to put out pressures above this value.
Flow at max rated pressure	This defines the high pressure point on the pump load curve. This is the flow that results when the pump is discharging into "max rated pressure".

3.9.2 Pump Curve

The Pump Curve is the load curve (flow vs pressure) for the pump. It is computed and redrawn as you change the parameters. The only purpose for the curve is to let you easily see the pump response given the parameters you have entered and to compare it to pump curves in a commercial specification sheet. An example is shown below:



3.9.3 Comments Tab

The comments section can be used for any purpose you find useful.

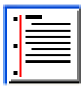
4. DEVICES FOR USE IN CASES

The project toolbar contains icons representing devices you can add to a case to do a simulation. To add a new device to a case, simply drag it from the toolbar onto the case window.

4.1 Standard devices for all cases

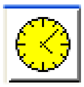
These standard devices are automatically included in new cases. You do not need to drag them onto the case and they are not on the toolbar.

4.1.1 Documentation



The documentation device simply has a place for making comments on the specific case. It does not affect the simulation.

4.1.2 Timing

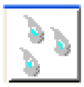


The timing device controls the simulated run time, how the lines are broken into pieces for solution and controls the print interval for time samples of the output.

Property	Description
End Time	Unless ended by a trigger, the simulation ends after simulating this amount of run time.
Minimum # reaches	A reach is a segment of a long line. This field tells the program a minimum number of reaches to divide the shortest line up into. In general, accuracy improves as this number is increased. But computer run time goes up roughly as the square of this number, so there is good reason not to make it too big. Usually 25-50 is plenty. Sometimes the output can contain ringing or other instabilities that can be fixed by increasing the number of reaches beyond 50. If you increase this number and the answer does not change appreciably, then you may safely use the lower number.

Print interval	Enter the time you would like to have between printed time samples in the output file. CS8 will get as close as possible to it. Note that given the minimum number of reaches and the length of the shortest line, CS8 computes a time step to use. The print interval must be a multiple of this time step, so you will not match the print interval exactly, but it will usually be close.
Print Ratio	Setting the Print Ratio to something other than zero allows the print interval to get longer as the simulation proceeds and results is a semi-logarithmic spacing of time samples. The actual print interval is calculated as: “PrintInterval” or “SimulatedTime * PrintRatio” whichever is larger. Example: Print interval = 0.1 sec, and Print Ratio = 1%. In this case, samples will print every 0.1 second up to 10 seconds. Then the interval will start to increase until it is 1 second between points at 100 sec, 10 seconds between points at 1,000 sec, etc.
Fine Printing Start	You can define an interval in which to print more often than normal. In this field, enter the simulated time at which you want to start the fine interval printing.
Fine Printing Duration	Enter the number of simulated seconds you want fine printing to last. If you set this to 0 sec, fine printing is disabled.
Fine Printing Interval	Enter the interval between time samples during fine printing. If you enter 0 sec, you will get every computed value.

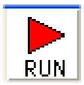
4.1.3 Fluid



This selects the fluid and the ambient temperature for use with all of the lines. This fluid is assumed to be used throughout the entire project.

Property	Description
Name of Fluid	Select the fluid to be used from the library using the pull down list.
Ambient temperature	This is an ambient temperature for use throughout the project unless some device has its own ambient temperature selection (such as the HPU).

4.1.4 Run

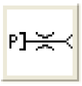


Run is not really a device. Click the Run icon when you want to run a simulation. When you do so, the Run icon will get a “clock” on its face that runs until the compute engine is finished with the simulation.

While running, the compute engine puts up a window showing its progress. If you click somewhere in the project, this progress window may disappear behind the project. You can get it back easily enough by looking at the bottom of the computer monitor and finding the icon for CS8engine and clicking it. It is an independent program like any other Windows application.

You can have more than one case running at the same time. They will not interfere with one another. On a computer with a multiple core processor, if you start a simulation while another simulation is running, it may actually run on its own core to run sort of in parallel.

4.2 Simple Pressure Source



The simple Pressure Source gives an unlimited amount of fluid supplied from a constant pressure source through a flow restriction. This is useful for a variety of purposes where pump rate and accumulator size in the HPU are not an issue.

The simple pressure source can cycle pressure on and off to the line. When on, it supplies fluid to the line, and when off, it drains fluid from the line.

Property	Description
Name	Name or identifier for this Pressure source. Each device used in a Case has a name that is always used in the same way. This name appears above the icon in the source window and is used to name the output files. You should make it unique in a Case, but this is not required. CS8 always adds a unique device number into the file name to make sure no two devices try to write to the same file. You may change this name at will, but realize that the plot refers to the output files. If you change the name of a file used in a plot, you have to edit the plot parameters to set the new file name or the trace will disappear from the plot and you will get an error message when you display the plot.

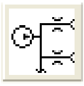
4.2.1 Source Parameters Tab

Property	Description
Resistance path or element	First choose whether you want to use a path or a single element. Then use the pull down box to select the resistance from the library. The path or element must already be defined in the library.
Source Pressure	Enter the gauge pressure that the source is to put out when it is ON. The source may be ON or OFF. When OFF, the gauge pressure is zero.
Height of HPU above the water	This affects the total control fluid head for the subsea components. Gauge pressure for this device is relative to the atmosphere.
ON Time	This is the amount of time that the pressure source will spend in the ON state before turning OFF. This time cannot be zero.
OFF Time.	This is the amount of time that the pressure source will spend in the OFF state before turning back to ON. Set this time to zero to have the source continuously ON.
At start of simulation, pressure is…	Use the pull down list to set the initial state of the source. Depending on this selection, the simulation will start at the beginning of an ON or OFF time. Thereafter, it will toggle between ON and OFF.
Unit to use for output pressures	Enter here a gauge or absolute pressure unit to be used for all computed outputs from the pressure source.

4.2.2 Comments Tab

The comments section can be used for any purpose you find useful.

4.3 HPU Pump/Manifold



The HPU pump/manifold is a very versatile source. You can add multiple pumps with independent start and stop pressures, multiple output lines with restrictions and regulators, and an optional accumulator bank.

The HPU has a lossless central supply header that connects all of the components together. The pumps discharge into the header, an optional accumulator is connected to the header through a restriction, and each line connects to the header through a restriction and an optional regulator.

The HPU keeps track of fluid flow into and out of a virtual reservoir as follows:

- Pumps always remove fluid from the reservoir, allowing no back flow.
- Fluid can flow either direction through the discharge ports feeding the umbilicals.
- If a regulator is not used, or the header pressure is below the regulator pressure, then any fluid flowing back toward the HPU from the line goes into the accumulator or another line.
- If a regulator is used and the header pressure is above the regulated pressure, then any fluid flowing back toward the HPU from the line goes into the reservoir. There is no check valve.
- Setting a regulator to a lower pressure than the header (like 0 psi) is a way of draining a line to the reservoir.

The icon shows a simplified schematic, not including the regulators. The regulators for the output lines go between the header and the restriction.

Property	Description
Name	Name or identifier for this HPU. For more details and information on the Name property, see the simple pressure source in section 4.1

4.3.1 Pumps Tab

Property	Description
Number of pumps	CS8 will allow up to 10 different pumps, each with its own library pump component, functional name, and start/stop pressures. Set the number of pumps before you enter the parameters for the pumps.
Function	Each pump has a functional name that is used only for legends in the output. This name will be something like main pump, HP Pump 1, Backup Air Pump, etc.
Initially Running	Put a check in this box if the pump is to start the simulation running. If unchecked, the simulation will start with the pump off. This sets only the initial state. After the simulation starts, the running state is controlled completely by the header pressure and the Start and Stop pressures.
Library pump selection	Beside the Initially Running check box is a pull down list for selecting the pump component from the library. You must define at least one pump in the library before you can set up an HPU device.
Stop	If the header pressure goes above this gauge pressure, a running pump will stop running.
Start	If the header pressure goes below this gauge pressure, a stopped pump will start running.

4.3.2 Restrictions Tab

Property	Description
Number of main umbilicals	CS8 will allow up to 20 outputs that can each supply fluid to a line (main umbilical).
Resistances from the header to the main umbilicals	Each line is connected to the supply header through a resistance. Those resistances are defined in the table below. Choose the number of main umbilicals, then fill in the parameters below.
Element or Path	Select the type of resistance you want to use and then select the resistance from the library using the pull down box at the right.
Use pressure regulator	You can place a pressure regulator between the supply header and the resistance by checking this box.

Regulated Pressure	If "Use Pressure Regulator" is checked, then you need to enter the fixed regulated gauge pressure in this field. If the supply header pressure goes below the regulated pressure, then that line functions as if the regulator did not exist and any back flow from the line will go into the supply header. If the line pressure goes above the regulated pressure so that fluid flows backwards, that fluid is treated as if it flows to the reservoir.
Get Pressure fcn from file	If you use a pressure regulator, you have the option of getting the regulated gauge pressure from a file that defines pressure as a function of time. See section 4.6.3 for the file structure and location. But do not use an absolute pressure unit. This file must be gauge. This overrides any constant regulator pressure you may have entered.

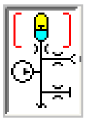
4.3.3 General Tab

Property	Description
Height of HPU above the water	This affects the total control fluid head for the subsea components. Gauge pressure is relative to the atmosphere for this device.
Pressure unit to use in computed outputs	Enter here a gauge or absolute pressure unit to be used for all computed outputs from the HPU.
Ambient temperature surrounding this device	If the HPU includes accumulators, then the accumulator gas will attempt to stabilize at this temperature.
Disable all pumps after this amount of time	The pumps normally sense the header pressure to determine when to run and when to stop. You can use this field to stop the pumps completely after a certain simulated period of time.
QaEpsilon	This determines the error tolerance of the equation solvers in the HPU. 10e-6 gpm seems to be a good number. A smaller number may improve accuracy but will take longer to run. However, if you make it too small, the solvers may have trouble converging on an answer. Reasons to mess with this number may be raggedy output where you would expect smooth, or virtual leaks, where a flow is present or a pressure changing with no reason. Normally, leave it alone.

4.3.4 Comments Tab

The comments section can be used for any purpose you find useful.

4.4 HPU with optional Accumulator

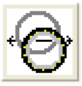


An HPU can include an accumulator bank. To do so, drag an accumulator from the toolbar onto the HPU pump/manifold icon. The HPU with accumulator has all of the properties and behaviors of the HPU Pump/Manifold (see section 4.3) to which it adds the accumulator properties listed below.

To edit the accumulator parameters, double click the accumulator symbol in the upper part of the icon. To edit the HPU Pump/Manifold parameters, double click the pump symbol at the bottom of the icon.

Property	Description
Accumulator	Use this pull down list to select an accumulator bottle from the library.
Number of bottles in the accumulator bank	The library device can be a single bottle. Selecting the number of bottles allows you to determine the total accumulator volume.
Accumulator Resistance	This is the resistance that is between the supply header and the accumulator bank. Select the type of resistance you want to use and then select the resistance from the library using the pull down box at the right.
Precharge Conditions	Type in the pressure and temperature of the precharge gas as it was precharged. This temperature and pressure are only used to determine the mass of gas that is in the accumulator. Note that this can be quite different from the pressure and temperature when the accumulator is actually used.
Initial Conditions	Enter here the state of the accumulator at the start of the simulation. You can enter a starting pressure or a starting volume of fluid. Normally, for a charged accumulator you should enter the starting pressure. If you want to start with the accumulator empty of fluid, enter 0 gal, not a pressure.

4.5 Lines



A primary design goal of CS8 and its predecessors is to model the transients due to wave motion, line packing, water hammer, etc. on long hydraulic supply and control lines. A line in CS8 is a tube or hose that is long enough that the dynamics matters.

In addition to the dynamic effects and normal frictional losses, CS8 includes provisions for handling additional pressure drop due to curvature from being on a reel or helically wound in a bundle.

Property	Description
Name	Name or identifier for this pump. See the Valve component for more details on the name.

4.5.1 General Tab

Data on the General tab refers to all types of lines.

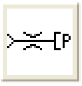
Property	Description
Tubing or Hose	Select the type of line and then select the specific conduit defined in the library using the pull down list.
Length of the line or umbilical	This is the straight end-to-end length of the umbilical. Do not include a correction for helical bundling. That will be handled automatically by describing the bundling on the Reel and Umbilical tab.
Number of lines in parallel	If you enter more than 1 in this field, then the line functions as if "N" identical lines are connected together at each end by a lossless "T".
Initial gauge pressure in the line	This is a pressure relative to the hydrostatic head in the line. It is assume that the whole line is initially at the same gauge pressure along its entire length.
Unit to use for the computed wave speed.	Speed of sound is an important computed quantity. This is the unit used to display it.
Comments	The comments section can be used for any purpose you find useful.

4.5.2 Reel and Umbilical Tab

Data on the Reel and Umbilical tab refers to the effects of curvature in the line.

Property	Description
Check this box if all or part of the line is on a reel	With this box unchecked, the line is assumed to be completely unreeled, laid out flat and straight. If you check the box, you will have to enter a description of the reel.
Average diameter of the reeled portion	Take the average of the inner hub of the reel, and the diameter to the outer surface of the outer layer of umbilical coils actually on the reel. The diameter to the outer surface of the umbilical coils is probably smaller than the maximum diameter that the reel will hold.
Portion of the line that is on the reel	The line is divided up into a number of reaches (short segments) for the computation. This entry determines how many of the reaches are considered to be curved and how many are straight for the purpose of calculating friction factor.
Check this box if the line is bundled inside of an umbilical.	The lines in a bundle are wound into a helix to allow the umbilical to bend easier. The helical shape causes additional pressure drop due to additional length and increased turbulence. If you check this box, you will have to describe the helix in the following two fields.
Distance from the center of the umbilical	Take the distance from the centerline of the umbilical (the axis of the helix) and the centerline of this tube or hose.
Distance to make one turn around the umbilical	This is the pitch of the helix. If the line makes one turn around the umbilical in 5 ft, then enter 5 ft. To calculate pitch from lay angle, use this formula: pitch = 2 * pi * (distance from center) / tan (lay angle)
Make a Guess	If you do not know the pitch of the helix, you can click this button. It calculates the pitch resulting from a 10 degree lay angle.

4.6 Simple Pressure Discharge



The simple pressure discharge is simply a fluid restriction discharging into a back pressure. The back pressure may be constant, or any desired function of time. It always terminates a line; no branching is possible from this device. It is useful for modeling a chemical injection case. It can be used to approximate a blocked line by making the restriction Cv very small (it cannot be zero), but the Manifold Discharge is actually better for simulating a blocked line.

Property	Description
Name	Name or identifier for this device. For more details and information on the Name property, see the simple pressure source in section 4.1

4.6.1 Restrictions Tab

Property	Description
Resistance from the source line to the discharge port	To flow into or out of the discharge end of the line, fluid must flow through this resistance.
Element or Path	Select the type of resistance you want to use and then select the resistance from the library using the pull down box at the right.
Vertical water depth above the discharge device	This height plus the height of the source above the water determines the absolute head pressure at the discharge end of the line.
Read opposing pressure function from a file	This check box controls how the back pressure is determined. When unchecked, the pressure is constant and entered in the "Opposing Pressure or Head" field. When checked, you are allowed to choose a file containing a set of points defining the pressure vs. time function. Pressure can be gauge or absolute, determined by the unit in the file. See section 4.6.3 for the file structure and location.
Browse	Click this button to choose your file from a list. This will only do something if "Read opposing pressure function from a file" is checked. If not checked, clicks are ignored.
(*back pressure file name*)	To the left of the Browse button is a text field that holds the name of the file that describes the back pressure as a function of time. The contents are important only if "Read opposing pressure function from a file" is checked. Clicking the field will allow you to choose a file just like the Browse button.

Opposing Pressure or Head	If, "Read opposing pressure function from a file" is not checked, then this field holds the constant back pressure that will be applied to the line. If not checked, then this field is ignored. If this is entered as a gauge pressure (like psi), then it is interpreted as being relative to **the sea water static pressure**. If it is entered as a head, the head is internally converted to an absolute pressure (like psia) based on the specific gravity of sea water (1.025).
Pressure unit for use in computed outputs	Enter here a gauge or absolute pressure unit to be used for all computed outputs from the simple pressure discharge.
Ambient temperature surrounding this device	Nothing in the simple pressure discharge uses this. It is included for future use.
QaEpsilon	This determines the error tolerance of the equation solvers in the device. 10e-6 gpm seems to be a good number. A smaller number may improve accuracy but will take longer to run. However, if you make it too small, the solvers may have trouble converging on an answer. Reasons to mess with this number may be raggedy output where you would expect smooth, or virtual leaks, where a flow is present or a pressure changing with no reason. Normally, leave it alone.

4.6.2 Comments Tab

The comments section can be used for any purpose you find useful.

4.6.3 Pressure vs. Time File Structure and Location

The file of pressure vs time file allows you to have any desired variation of back pressure over time. You must create this file yourself; at the moment, CS8 does not provide an editor for this file and it is separate from the project file.

4.6.3.1 Structure and file name

The pressure vs. time file is a "comma separated value" file. This normally has the file extension ".csv" and is easily written and maintained using Excel. This description will refer to the file as if it is being created with Excel, with rows and columns.

If you create the file with Excel, you write the file by choosing SaveAs, and then choosing csv from the "Save as type" drop down box.

Alternatively, you may create the file using Notepad with the file extension ".txt". Either file name extension is acceptable and, with one exception, the content is interpreted exactly the same way for both. The exception is for files created with the older version of Control Simulator called CSI, which makes an output file with the ".txt" extension. CS8 will recognize those files because of a line beginning with "Control Simulator Innovations" and another line beginning with "{-". Make sure that your file does not have both of these markers and it will be interpreted as described below.

4.6.3.2 Location

The file must be located in the same folder as the project file. The project file will have the name you gave it with the extension ".cs8proj".

4.6.3.3 Header section

The first several rows of the file may contain anything you want. Use these to describe the data, what it represents, assumptions made, etc. Anything another person would need to know to understand the data should be placed in the first several rows.

4.6.3.4 Units section

The last row before the data points must contain only the units of measure to be applied to the data points. It must be of the form

"unit of time", "unit of pressure" for example sec, psi

The comma is part of the csv file structure and is required. A space is allowed but is neither required nor sufficient. Do not put anything else on this line.

To recognize the units row, CS8 starts at the end of the file and looks for the first values that are not a valid numeric data point. So, do not put any comments or other text after the units row.

The units of measure may be any units recognized by CS8. To see a list of allowable units, right click any time or pressure field. Three useful time units definitely available are sec, min, and hr.

The pressure unit may be a gauge pressure or an absolute pressure. Be careful, because they are used quite differently. The absolute pressure (like psia) is well defined as the pressure relative to vacuum and includes any static head components. ***The gauge pressure (like psi) is a pressure relative to the sea water static pressure.*** Note that most gauge pressures in CS8 are relative to the control fluid static head, so this one is different.

To see the available pressure units, right click the “Pressure unit for use in computed outputs” field. You will see gauge pressures in the first column, and absolute pressures in the column to the right.

4.6.3.5 Data section

The data section consists of time,pressure ordered pairs of numbers, separated by a comma. For instance, assume that the units row is “sec,psi”. Then data in the the row

120,2500

means “at 120 seconds the pressure is 2500 psi above the sea water head pressure”.

The first time in the data section does not have to be 0. If it is greater than zero, then the pressure of the first data point is used for all earlier simulated times. If the simulation goes beyond the time in the last data point, then the pressure in the last data point is used for all later simulated times. See the example in section 4.6.3.6.

Blank lines are allowed. Do not put anything else in the data section.

The time must increase as you go down in the list. Spacing between time points can vary as you wish. CS8 will draw a straight line between time points when interpolating, so you only need to include the start and end points of a linearly changing pressure. Curves can be specified by as many points as you wish to get the curve “smooth”.

Minimize the number of points to minimize run time. This should not be a problem unless you get many hundreds or thousands of points.

4.6.3.6 Sample file

An example of a properly structured file follows.

```
Demonstration file for injection using CS8,

This represents a pressure vs time at the output of an injection line.,
Column 1 must be the time, and column 2 is the pressure",
Time points must be monotonically increasing,
You can put anything you want at the top of the file,
The last row before the data points must be the units,
Units like psi and psig mean pressure relative to the sea water static pressure,
Units like psia are absolute pressures,

sec,psi
50,1000
100,2000
101,1000
150,1000
151,2000
250,1000
```

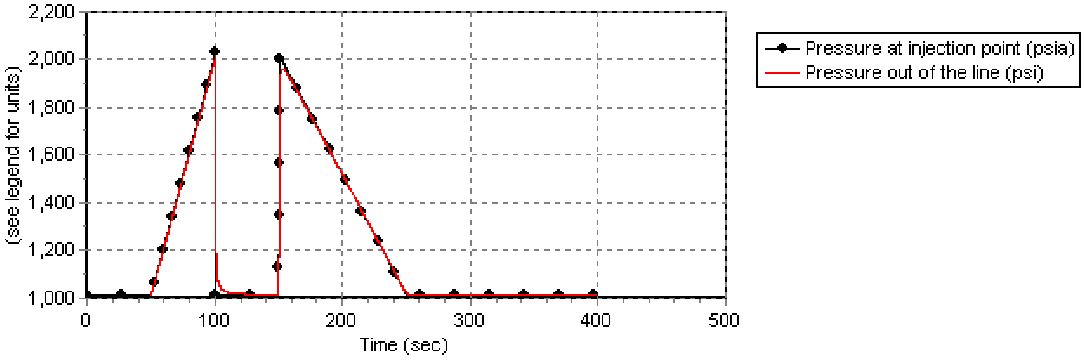
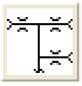


Figure 1 – Plot of the preceding data file as used in a simple simulation on the surface (no static pressure due to depth).

4.7 Manifold Device



The manifold device provides a junction point between two and is also the basic device from which to construct branched systems and multiplex control systems with accumulators and valve modules.

The manifold device consists of a central, lossless header connecting together a number of lines. There is one or more source lines, and zero or more discharge lines. Connecting source line(s) to a number of discharge lines is fundamental to modeling branched systems.

You can add a valve module and an accumulator to the manifold device, giving it great flexibility. The accumulator and valve module are described in following sections.

Property	Description
Name	Name or identifier for this device. For more details and information on the Name property, see the simple pressure source in section 4.1

4.7.1 Source Restrictions Tab

Property	Description
Number of source (inlet) connections	Every discharge device has to have at least one source line. When set to 1 or more, a table will appear below allowing you to select a resistance between the header and each of the source lines. Resistance number 1 corresponds to the line on the top left connector of the icon and the number increases in order as you go down.
Resistances from the source lines to the header	CS8 will allow up to 20 source connections that can each supply fluid to the header. The schematic inside of the icon shows how the lines are connected to one another.
Element or Path	Select the type of resistance you want to use and then select the resistance from the library using the pull down box at the right.

4.7.2 Disch Restrictions Tab

Property	Description
Number of discharge (outlet) connections	If set to zero, then there will be no discharge lines. If there is only one source connection, this manifold will be blocked, allowing no fluid to flow into or out of the source line. If there is more than one source line, the fluid can flow between source lines as well as the discharge lines. If set to 1 or more, then a table will appear below allowing you to select a resistance between the header and each of the discharge lines. Resistance number 1 corresponds to the line on the top right connector of the icon and the number increases in order as you go down.
Resistances from the header to the discharge lines	CS8 will allow up to 20 discharge connections that can each supply fluid to a line. The schematic inside of the icon shows how the lines are connected to one another
Element or Path	Select the type of resistance you want to use and then select the resistance from the library using the pull down box at the right.

4.7.3 Depth Related Tab

Property	Description
Vertical water depth above this discharge device	This height plus the height of the source above the water determines the absolute head pressure at this manifold
Pressure unit for use in computed outputs	Enter here a gauge or absolute pressure unit to be used for all computed outputs from the manifold and any associated devices
Ambient temperature surrounding this device	If the manifold has an accumulator, then the accumulator gas tries to stabilize at this temperature.
QaEpsilon	This determines the error tolerance of the equation solvers in the device. 10e-6 gpm seems to be a good number. A smaller number may improve accuracy but will take longer to run. However, if you make it too small, the solvers may have trouble converging on an answer. Reasons to mess with this number may be raggedy output where you would expect smooth, or virtual leaks, where a flow is present or a pressure changing with no reason. Normally, leave it alone.

4.7.4 Triggers Tab

Triggers are binary flags that can be set or cleared. In the manifold and devices associated with a manifold, certain conditions can set or clear the flags, while other devices can monitor flags to control their behavior.

The flags are "owned" by the manifold. Flags must be defined in the fields below for the manifold before they can be used by the manifold or is associated optional devices (like valves).

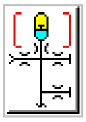
Property	Description
Name of new Trigger - Add	To define a new trigger (one not already listed in the table below), enter a trigger name that is not already in the list. Then press the Add button.
Triggers available for this manifold	This is a list of triggers that have been defined for use by this manifold (and its valves, if any).
Delete Selected Trigger	To delete a trigger name from the list, click the trigger name to highlight it, then press the Delete button at the bottom of the list.
Special Triggers	Four triggers are used by the manifold. The manifold can set two based on manifold header pressure. The others cause an action when they are set.

Two triggers are available to trigger on low manifold pressure (relative to sea).	If the pressure in the manifold goes below a preset level (see next fields), a trigger can be set.
Trigger Pressure 1	Normally used for DCV dropout functions. This field is grayed out unless the pull down box above it has a trigger selected. If you select a trigger, then a field becomes visible into which you type the trip pressure (gauge relative to sea). The trigger is false at the start of the simulation. When the header pressure goes below your trip pressure, the trigger will be set to true. Once true, the trigger stays true for the rest of the simulation.
Trigger Pressure 2	This works just like Trigger Pressure 1, except that it allows you to have two different trigger pressures.
Assign a trigger to simulate DCV dropout.	Select at least one trigger for low manifold pressure above. Then select that same trigger here. When this trigger becomes true, it causes all valves on this manifold to stop moving[see note]. This stops them from discharging any fluid into the lines and has the same effect on the supply system as the dropout of all DCV's in a real subsea control module (namely local venting of all of the valves). NOTE: If you select a trigger in this box, it will cause the "dropout" of all valves on the manifold ***unless the specific valve has its own dropout trigger specified***. (see DCV Dropout Trigger for the valves, section 4.9.1, for more details)
Assign a trigger to end the simulation	If you select a trigger here, the simulation will be terminated if this trigger gets set to true, or the time limit is reached, whichever happens first. Use this to stop the simulation based on valve movement or low manifold pressure.

4.7.5 Comments Tab

The comments section can be used for any purpose you find useful.

4.8 Manifold with optional Accumulator



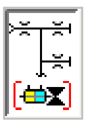
You may drag a discharge accumulator from the tool bar onto a manifold device. This connects an accumulator to the manifold main header through a restriction. A manifold with an accumulator could be used by itself, or it could become the supply system of a multiplex control pod.

The manifold with accumulator retains all of the properties of the manifold device (see section 4.7) but adds the following accumulator parameters to it. To edit the accumulator parameters, put the mouse on the accumulator symbol in the upper part of the icon and double click. To edit the manifold parameters, double click the manifold symbol main header.

.

Property	Description
Accumulator	Use this pull down list to select an accumulator from the library.
Number of bottles in the accumulator bank	The library device can be a single bottle. Selecting the number of bottles allows you to determine the total accumulator volume.
Accumulator Resistance	This is the resistance that is between the supply header and the accumulator bank. Select the type of resistance you want to use and then select the resistance from the library using the pull down box at the right.
Precharge Conditions	Type in the pressure and temperature of the precharge gas as it was precharged. This temperature and pressure are only used to determine the mass of gas that is in the accumulator and does not necessarily have anything to do with the subsea conditions.
Initial Conditions	Enter here the state of the accumulator at the start of the simulation. You can enter a starting pressure or a starting volume of fluid. Normally, for a charged accumulator you should enter the starting pressure. If you want to start the accumulator empty of fluid, enter 0 gal rather than a pressure.

4.9 Manifold with Valve Module



You may drag a valve module from the toolbar onto a manifold device. This connects a DCV header to the manifold main supply header through a restriction. The DCV header can supply one or more valves. With or without an optional accumulator, this forms the basis of a multiplex control pod supplying several valve operators.

.

Property	Description
Name	Name or identifier for this device. For more details and information on the Name property, see the simple pressure source in section 4.1

4.9.1 Valves Tab

Property	Description
Valves on this manifold	This is a list of all of the valves defined as part of this module. Each valve has a number of parameters associated with it. Click one of the valves. It will become highlighted and all of its associated parameters will appear in the fields to the right of this list for you to view or edit.
Add	Click this to add a new valve to the bottom of the list
Insert	First, click a location in the list where you would like a new valve to appear. The valve will become highlighted. Click Insert. The valves at that location and below will move down one space and a new valve will appear at that location.
Delete	First, click the valve that you want to delete. The valve will become highlighted. Then click delete. ***You do not get a warning or a second chance and you cannot paste it back.***
Clone	This adds a new valve to the bottom of the list that is an exact copy (except for the name) of another valve that is already in the list. First, click the valve you want to make a copy of. Then click Clone. Finally, change the name of the new valve to something meaningful.
Move Up Move Dn	Lets you change the order of the valves in the list. Click on a valve you want to move, then click Move Up or Move Dn to change its position in the list. Valves are not added or deleted.

The commands above manage the list of valves. The parameters below are all associated with a single valve. Click a valve in the list. It will become highlighted and all of its parameters are displayed in the fields to the right in the window. For the fields described below, keep in mind that they all apply to the valve that is highlighted in the list. For a schematic of the valve module, see the General Tab.

Functional Name	Type a name that identifies the function of this valve within the context of the manifold or the project. An example of a proper name would be “Upper Master”. It serves no

	purpose other than identification.
Inlet resistance	Select a resistance from the library that goes between the DCV header and the inlet of the valve. Normally, this will have at least an orifice representing a DCV in series with some length of connecting tubing.
Valve	Select a valve from the library using this pull down list.
Outlet Resistance	Select a resistance from the library that goes between the vent header and the outlet of the valve. Normally, this will have at least an orifice representing a DCV in series with some length of connecting tubing. Note that the "reverse" direction of this restriction is used when flow is positive. That is because in a normal MUX system, flow out of the "spring" side of the valve operator may flow to the vent system through a DCV, but in its reverse direction. Keep this in mind when selecting an outlet resistance.
Start Time	The valve is not allowed to move before this time in the simulation. This, in conjunction with the start time on the General tab, determines when the valve is allowed to move. Once it is allowed to move, the valve will move or not based on the pressures in the system. Note that the Open Trigger is used in place of a starting time if it a trigger is selected.
Start Position	This sets the position of the valve in percent of its stroke at the beginning of the simulation.
Repeat Count	This is the number of **additional** valve operations that will be done. If this count is greater than zero, then as soon as the valve is able to move for the first time, a repeat timer is started. When the repeat timer times out, the valve is returned to its Start Position. NOTE: no fluid is consumed when the valve is returned to its starting position and that motion is not actually simulated. The model piston position is simply reset to the starting state regardless of where it had moved to. If the valve movement was not complete before the repeat starts, then your answer may not be what you really want. This is normally fine for simulating transients and fluid draw from the supply system.

	Example 1: Start Time = 10, Repeat Count = 1, Repeat Period = 5. In this case, at 10 seconds, the valve will begin moving. Then, 5 seconds later, the valve is immediately returned to its Start Position and allowed to move again. Example 2: For a choke requiring 100 steps to open, Repeat Count would be set to 99 and Repeat Period would be set to the amount of time between stepping pulses.
Repeat Period	The amount of time between repetitions of the valve movement. This is ignored is Repeat Count = 0. See Repeat Count for more details.
Bore Pressure	This is an absolute pressure that is applied to the stem of the valve. It is constant and is applied for the duration of the simulation.
Use DCV Regulator	Check this box if there is a regulator between the DCV header and the inlet restriction of the valve. If this box is checked, then you must enter a regulated pressure.
DCV Reg Pressure	Enter here the regulator set point of a regulator connected to the DCV header. This regulator set point is a gauge pressure relative to the sea water head. If the absolute DCV header pressure is at or above the absolute regulator set point, then the regulated pressure will be applied to the valve inlet restriction. If the absolute DCV header pressure is below the absolute regulator set point, then the DCV header pressure will be applied directly to the valve inlet restriction.
Start Trigger	If a trigger is selected here, the valve is not allowed to move until this trigger is true, regardless of the timing. If a trigger is selected, the start time becomes invisible since it is ignored.
DCV Dropout Trigger	When this trigger becomes true, only this particular valve stops moving, simulating a DCV dropout in this one valve. On the output, this valve goes directly to 0% of stroke, but with no fluid movement. If a trigger is selected here, this valve ignores the general DCV Dropout trigger for the manifold (see section 4.7.4).
On full stroke, set (a trigger)	If a trigger is selected, it will be set when the valve reaches 100% of stroke. Common uses are to start another valve or to end the simulation early.

On 0% stroke, set (a trigger)	If a trigger is selected, it will be set when the valve reaches 0% of stroke. Common uses are to start another valve or to end the simulation early.
On stop, set (a trigger)	This trigger is set the first time the valve stops moving.

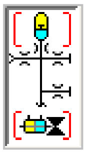
4.9.2 General Tab

Property	Description
Add this time to all valve start times	This field is used to delay the entire sequence of valve operations for this manifold. The intended purpose of this parameter is the following: Define a manifold with valves that represents a tree. Use individual valve start times to set the opening sequence for the valves on this tree. Use Copy/Paste to create one or more duplicate trees. Change their names and change this time so that the duplicates go through their opening sequence later and do not overlap earlier tree openings.
Check to print details of resistance paths	Use this to turn on or off the printing of various detailed data about each restriction in this manifold.
Resistance from supply node to DCV header	Use this to select from the library a resistance that goes between the manifold supply header and the DCV header. In a subsea control module, this would represent filter, selector or shuttle valve, dump valve, and anything else that is between the accumulator connection and the DCV's.
Check this box if the valve module vents to sea.	Normally this box will be checked. In an unusual situation such as venting into a column of mud in a riser, uncheck this box and enter the height and density or specific gravity of the fluid in the column.
Resistance from the vent header to the fluid column	All valve outlets are gathered together and the combined fluid has to flow through this restriction. If the individual valves all actually vent directly to the sea, then make this Cv large enough to be a negligible restriction.

4.9.3 Comments Tab

The comments section can be used for any purpose you find useful.

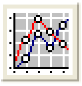
4.10 Manifold Device with Accumulator and Valve Module



You may drag an accumulator and a valve module onto a manifold. This creates a discharge device that has all of the parameters and behaviors of the three parts. To review those please see section 4.7 for the manifold, section 4.8 for the accumulator, and section 4.9 for the valve manifold.

To edit data, this icon has three regions. Double clicking the accumulator on top edits the accumulator data. Double clicking the manifold header in the center of the icon edits the manifold device parameters. Double clicking the valve symbol on the bottom of the icon edits the valve module.

4.11 Plot (and Tabular Output) Device



The plot device is a powerful tool for visualizing the output from CS8. It makes it easy to plot computed results and to compare the computed results to test data or to output from earlier versions of the Control Simulator. You can easily show dissimilar data such as pressure and flow on the same graph by scaling the data before it is plotted. The data plotted in the traces is also available in a file as columns of numbers in the same format as the raw output from CS8.

You use the plot device by dragging it onto the case window, then setting its name, adding traces, and setting parameters for each trace. You may have as many plots in a case as you wish.

Property	Description
Name	Name or identifier for this device. For more details and information on the Name property, see the simple pressure source in section 4.1

4.11.1 Source Data Tab

Property	Description
Traces on the plot	This is a list of the traces that are on this plot. The order that they appear in this list is the order that they will appear in the legend of the plot.
Add	Click this to add a new trace to the bottom of the list

Insert	First, click a location in the list where you would like a new trace to appear. The line will become highlighted. Click Insert. The traces at that location and below will move down one space and a new trace will appear at that location.
Delete	First, click the trace that you want to delete. The line will become highlighted. Then click delete. ***You do not get a warning or a second chance and you cannot paste it back.***

The commands above manage the list of traces. The parameters below are all associated with a single trace. Click a trace in the list. It will become highlighted and all of its parameters are displayed in the fields to the right in the window. For the fields described below, keep in mind that they all apply to the trace that is highlighted in the list.

Legend	This is the text that will appear in the list and in the legend of the plot for this trace.
Unit from column header	With this checked, CS8 will look at the top of each column in the data file for a unit of measure and insert that unit of measure into the legend.
File Name	This is the name of the data file that is the source of the data being plotted. Double click the field or click the Browse button to get a list of available files. You can also paste the file name from another trace. The list of files includes all files with extensions .txt and .csv that are in the case folder or in the folder with the project file.
View File	Press this button to bring up the selected file in Notepad. This allows you to view the headings and numeric data to see which columns contain the data you want.
X Column	This is the number of the column containing the X coordinate of the points to plot. The left column is column 1. Select the desired column from the pull down list. The normal plot in CS8 is a plot as a function of time, in which case X Column is normally 1. But it is useful to plot other things such as pressure vs. flow, or pressure vs. valve position.
Y Column	This is the number of the column containing the Y coordinate of the points to plot. The left column is column 1. Select the desired column from the pull down list.

Select X by Symbol Select Y by Symbol	These buttons bring up a list of the column headers for the selected data files. Once you are familiar with the data in the file, this is a quicker way of selecting the X and Y columns.
Line: Visible, Color, Weight	When Visible is checked, CS8 will draw a line from data point to data point in the selected color. Weight controls the thickness of the line.
Symbol: Visible, Style, Color, Size, Spacing	When Visible is checked, CS8 will mark data points by the selected symbol in the selected color. The size of the symbol is selectable. The spacing of symbols deserves a comment. If you set spacing to zero, then you get a symbol at each data point. Usually, this is too many. Select a non-zero spacing to limit how closely symbols will be plotted. The spacing is the minimum space between symbols measured in pixels. Symbols are still at data points, but not all data points. You can choose different spacings for curves that are close together to insure that the symbols are not always on top of each other.
Y Scale	Apply this scaling factor to the Y data before it is plotted. This allows you to have different kinds of data plotted together. For instance, you may plot flow and pressure on the same plot, or pressure and valve position. Plotted value = (Yscale * f(Y)) + YOffset The scale factor appears in the legend along with the unit of measure. Ex: a Y Scale of 10 causes valve position to plot in the range of 0 to 1000 (for 0% to 100%).
Y Function	After the Y Scale field there is a X and a pull down list. Normally you want to plot Y. But this list lets you plot other functions of Y. The function f(Y) is applied before the scaling factor (see Y Scale for more information). **Y**: simply use the value from the file **± sqrt(|Y|)** : If Y >= 0, use sqrt(Y). If Y < 0, then use –sqrt(-Y) **±Y^2**: If Y >= 0, use Y^2. If Y < 0, use –Y^2 |**Y**|: use the absolute value of Y.
Y Offset	This offset is added to the value before it is plotted. see Y Scale for more information

X Scale, X Offset	See Y Scale and Y Offset for more explanation. These are seldom used but can be useful. The X coordinate plotted is found by (Xscale * X) + Xoffset
Limit the range of the X Axis	You may want to only plot a portion of the available data. This allows you to set a start and end value for the X coordinate. The limits are checked after applying the scale and offset. **This applies only to this particular trace**.

4.11.2 Plot Area Tab

Property	Description
Title of Plot	This title goes on top of the plot. It may be multiple lines long.
Vertical Axis Label	In the case of plotting mixed units, it may be best to have no label on the vertical axis, or simply refer the reader to the legend.
Horizontal Axis Label	Use this label on the X axis
Limit the range of the X Axis	You may want to only plot a portion of the available data. This allows you to set a start and end value for the X coordinate. The limits are checked after applying the scale and offset. **This applies to all traces**.

5. TECHNICAL NOTES

5.1 Combining Long and Short Lines (flying leads)

There is no restriction on the length of a line. But in a Case where several Lines of different lengths are included in the model, problems can occur if the longest line is extremely long compared to the shortest line.

5.1.1 What is "extremely long"?

By "extremely long", I mean a ratio of 100:1 or more. CS8 will allow the ratio of the longest line length to the shortest line length to be as much as about 200:1 before really bad things start to happen. Remember, this has nothing to do with the actual length of any particular line; it is the ratio of the longest to shortest line length[3].

This is often a problem if you try to model a short flying lead or jumper as a line. Your main umbilical may be 20 km long and your flying lead is 200 m long, a ratio of 100:1.

5.1.2 What are the symptoms of the problem?

The most common symptom of this problem is that the simulation will run very slowly. If you do not get a warning message, the answers will be OK, but you may have to wait many minutes or even hours to get them.

If the ratio of line lengths is too great, CS8 will give you a warning message like the following:

"Number of segments revised…" means that CS8 cannot properly prepare the line for simulation. There is a good possibility that the simulation will become unstable and the

[3] To be precise, it is not the length in feet or meters, but the length in terms of time of travel for sound. In this respect a 1000 ft hose is actually about twice as long as a 1000 ft steel tube. because sound travels slower in hose. But this is a detail that does not change the basic point of the discussion above.

Compute Engine may even crash. The resulting output, even if the simulation runs successfully, is highly suspect. If you get a message like this, you must fix the problem.

5.1.3 The Fix… maybe

5.1.3.1 Make the shortest line a bit longer

The longer lines usually control the overall response of the system, so you want them to be as accurate as possible. But the shortest lines often can be doubled (or more) in length without changing the answer much at all. Try doubling the length of the shortest line. If this helps, double it again and see if the answer changes significantly. If it does not, then you have a "good enough" simulation.

5.1.3.2 Model the shortest line as a resistance element

Suppose that you have a manifold with several wells connected with flying leads to a central node. Model this as if all of the valves are on the same manifold. The resistance of the flying leads will be included as a length of tubing in the appropriate resistance element. You may have to get creative about how you model the various resistances of the system to make this work. Be aware of which resistances are subject to flow from only one valve vs. those that pass fluid for several valves if multiple valves will be moving at the same time.

5.1.4 Why This Happens

The mathematical model used in The Control Simulator requires a well-defined relationship to exist between the number of segments (reaches) a given line is divided into, the size of the time steps, and the speed of sound on the line. Using more reaches (see section 4.1.2) or shorter lines result in shorter time steps.

The number of reaches must be greater than 25 because simulation accuracy suffers with fewer reaches. But requiring 25 reaches or more in the shortest line sets the required time step to a small value. When you apply that small time step to a long line, it results in a large number of reaches for the long line. However, the maximum number of reaches is limited to 4998. If the compute engine cannot satisfy the relationship between reaches and time step on all of the lines, then the timing relationship for one or more lines will be wrong and an accurate simulation is impossible.

5.1.5 How can you tell how many segments (reaches) are being used?

Dividing a line up into a large number of segments usually increases the accuracy of the simulation, but the run time goes up roughly as the square of the number of segments. So past a certain number of segments you increase run time but make no practical difference in the answer. Information on the number of segments and the size of the time steps being used can be found in two places:

- Right click the Timing component on the Case (the yellow clock icon), and view the header file. It contains information on how the time step is decided upon and the time step actually selected to use.

- Right click any line and view its header. At the bottom you will find the number of segments the line is divided up into. If this number is 4998 then it is extremely likely that the long/short line length ratio is too large and the answer is possibly quite wrong.

5.2 Multiple Identical Valves

Putting lots of valves on a manifold results in extended run times. If two or more identical valves move at exactly the same time, or they do not overlap at all, you can save run time and data complexity by using these tricks.

5.2.1 Model as one big valve

If you have more than one of the same type of valve on a manifold and they will always be moved together, the simulation will run faster and give the same answer if you model them as one valve. Assume you have N identical valves moving together. Instead of putting N valves in the model, put one valve. In the library, define a valve with N times the volume or stroke but with all other valve parameters the same as the actual valve. Use that valve in the model instead of the actual valve.

But note: If you do this, you must consider that N times as much fluid will be flowing through the Inlet Resistance and Outlet Resistance as would be the case if it was one valve. So you have to use altered resistance elements or paths in those locations. See section 5.4.

5.2.2 Model using the Repeat feature

A way to do multiple identical valves that to not overlap (only one is moving at any one time) is to use the Repeat feature. This was introduced in Release 11 and allows one valve to be started over multiple times. See section 4.9.1 for more details. This is simpler than multiple valves and runs faster.

5.3 Modeling a stepping choke

In many ways, a choke is just a valve operator that you operate 100 to 200 times. But it does have a special set of problems because the cylinder is normally so tiny. If you are not careful, you can end up with a model that is highly unstable. There are two ways to approach a choke.

5.3.1 Gross behavior model

The fluid used by any one pulse is minuscule and usually negligible. If you are willing to ignore the details in favor of overall system response, construct a single valve with a volume equal to the volume of the choke cylinder time the number of pulses. Then adjust the series resistance so that the valve moves in the time number of pulses times pulse period. The “choke” will consume the same amount of fluid in the same amount of time, and the long lines will only see the effect of the that total fluid draw from the system.

For the valve on the tree, set Repeat Count = 0. The model is simple and will run rapidly.

5.3.2 Detailed behavior model

Construct a valve that models the choke cylinder as accurately as possible in terms of volume, spring force, friction, etc. Then, when included on the tree, set Repeat Count = desired number of pulses, and Repeat Period = time between pulses. This should also give the same flow from the system supply over time, but will also show the response to each tiny pulse of fluid. With no further attention, it will probably also become quite crazy.

The problem is that the volume is generally so small that the valve model will be slammed from 0% to 100% of travel in only 1 or two time steps, causing huge, spurious pressure excursions and instabilities. To minimize this, you will have to go to the Time icon and force a much larger “Minimum # reaches” than normal, maybe a couple of thousand. You may also need to divide your main umbilical up into 2 or more segments to help get the time step small enough that the valve (choke cylinder) strokes in 10 or more time steps.

You may also need to increase the resistance in series with the valve (choke cylinder) so that fluid flow rate is limited to make the cylinder stroke in a longer time.

To see what is really happening with this tiny cylinder, use the print interval or Fine Print interval to print the valve operation in extreme detail. You can see if the time step is small enough to even see this mind of detail by looking at the header for the timing icon (right Click the Clock icon on the case window and look at the header). You will be looking for the last line that says something like “Time step is set to ?????? sec per iteration…”.

NOTE: This model does not simulate the return of the piston by the spring.

Example: Set up the End Time to run only the first 10 steps or so. Pick a large number of reaches, like 500 and a Print Interval of say 0.001 sec. Then run the simulation, and immediately terminate it. Right click the Clock and see what the time step is. If it is long, like 0.5 sec, increase the number of reaches and try again. When it is something small like 0.03 or 0.01 seconds or less, run the simulation and plot the pressure in the inlet cylinder. If it oscillates wildly, increase the number of reaches and try again. Once it is reasonably stable, and the waveform does not change appreciably when you double the number of reaches, you are good to go.

If you try this, let me know how it goes, and if you have big problems, send me your model so I can see what is going one.

5.4 Multiple Resistance Elements in Parallel

Suppose you have a circuit that has multiple, identical resistance elements in parallel. How do you model that parallel combination with the resistance elements in CS8?

Assume you have N identical elements in parallel and you know the tubing type and length and the Cv of the elements. Here is how to define a new element that is the exact equivalent of N elements in parallel so that the pressure drop matches for any flow rate, regardless of Reynold's number.

Parameter being modeled	**Actual element, of which you have N in parallel**	**Parameter for the equivalent single element**
Orifice flow coefficient	Cv	N * Cv
Pipe internal diameter	D	N * D
Pipe roughness	e	N * e
Number of bends	b	N^2 * b
Pipe length	L	N^3 * L

Example: You have 4 resistance elements in parallel. They each consist of a coupler with Cv=0.5 and a ½" ID pipe with 2 bends, roughness of $5x10^{-6}$ ft and length 6 ft. What is a single element that is equivalent to the 4 elements in parallel?

Cv'	= 4 * 0.5	= 2
ID'	= 4 * 0.5"	= 2"
e'	= 4 * $5x10^{-6}$ ft	= $20x10^{-6}$ ft
b'	= 4^2 * 2 bends	= 32 bends
L'	= 4^3 * 6 ft	= 384 ft

6. ERROR MESSAGES

6.1 Number of segments revised from …

Your Case includes lines that are extremely long relative to the shortest lines. See section 5.1.

Believe it or not, this is the end of the section. Most error messages are self explanatory. At least, that is my goal. If you have a question about an error message, email me with your question.
www.mcadamsengineering.com, Contact.

APPENDIX 1 - DEFINITION OF TERMS

Absolute – An absolute pressure is one that is relative to a vacuum.

Gauge – A gauge pressure is measured relative to some other reference point. In surface equipment, the reference is usually 1 atmosphere. For most subsea equipment in CS8, the reference pressure is usually relative to the static head of the control fluid.

Component – A functional building block used to construct devices from which to build cases. Components are kept in a library that is part of a project. The properties used to describe a particular component are predefined and fixed, no matter how or where the component is used. Ex: the diameter of tubing is a component property.

Device - A predefined hydraulic circuit that can be added to a case to build up a circuit to simulate. Example: a line or manifold.

Discharge – The right side of a device as you view its icon on the screen. See outlet..

Discharge Device – A device that goes on the end of a line that is farthest from the master control point.

Double Click - With the mouse cursor over an object, rapidly press and release the left mouse button twice.

Drag and Drop – Position the mouse cursor over the object to drag. Press and hold the left mouse button while you move the mouse cursor to where the object needs to be. Then release the left mouse button.

Hover – Position the mouse cursor over an object.

Icon – the picture representing a device.

Inlet – A connection point on the side of a device or long line that is closest to the master control point (source side). Fluid may flow into or out of an inlet, with fluid flowing into the device being defined as positive flow. In CS8, inlets are normally on the left side of the icon.

Library – The part of a project file that holds components for use in building cases. Components in the library are only defined once when you put them in the library. Thereafter, the components are referenced by name to build devices and cases.

Line – A relatively long conduit for conveying hydraulic fluid. In CS8, lines refer to long lines where the dynamics of pressure waves, compression of fluid, and expansion of the line are important to the simulation. Short pieces of tubing or hose should be modeled as resistance elements or paths.

Master Control Point – See Source Device

Outlet – A connection point on the side of a device or long line that is farthest from the master control point (discharge side). Fluid may flow into or out of an outlet, with fluid flowing out being defined as positive flow. In CS8, outlets are normally on the right side of the icon.

Path – A number of resistance elements in series.

Project – A single file containing a library of components and any number of cases to be simulated.

Resistance Element – A simple flow restriction combined with a relatively short section of tubing. The length of the tubing is short enough that it can be assumed to always be in steady state flow, with the dynamics and transients being negligible.

Right Click – With the mouse cursor over an object, press and release the right mouse button once.

Source – The left side of a device as you view its icon on the screen. See inlet..

Source Device – The device in a case that serves as the master control point. A case can have only one source device with at least one line coming out of it to feed the rest of the system. Source devices have no inlets.

APPENDIX 2 – FILES AND NAMING CONVENTIONS

The only files required for running CS8 are its graphical user interface, the compute engine, and the owner license file. They may be put in any folder or even on a USB memory stick and executed from there. CS8 makes no use of the registry and does not require a formal install process.

CS8.exe - The graphical user interface for CS8.

CS8engine.exe – The compute engine for CS8. You can run multiple copies of compute engine at the same time as long as they are operating on different case files. This file must be in the same folder as CS8.exe.

***ownername*.cs8lic** – The CS8 license file. Neither the graphical user interface nor the compute engine will run properly without it. This file must be in the same folder as CS8.exe.

CS8 Quick Start.pdf – The file containing the quick start manual, which has tips on installation and setting up basic projects. It should be located in the same folder as CS8.exe, but other copies may be placed anywhere that is convenient.

CS8 Technical Manual.pdf – The file containing this manual. It should be located in the same folder as CS8.exe, but other copies may be placed anywhere that is convenient.

***projectname*.cs8proj** – All projects have names of this form. I highly recommend that you put a project in a folder all to itself. CS8 creates a lot of files related to the project, and it can get confusing if you have more than one project in a folder. But you are free to put any files you want in the folder with the project. It would make sense for instance to put test data, background data, valve and pump specifications, and other reference data in the folder with the project file.

All of the information needed for the project is contained in the one project file. All of the other files are either temporary files or output files. To send a particular case to a colleague, you only need to send the one file: *projectname*.cs8proj. But, if you zip and send the whole project folder, the recipient will be able to look at output files without rerunning the simulations.

(*projectname*) *casename*.cs8case – When you run a simulation of a case, the graphical user interface makes a file with this name structure to be read by the CS8 compute engine. It is recreated each time that the case simulation is run. These files may be deleted without harm.

(*projectname*) *casename*.cso – When the compute engine runs a given case, it creates a folder with this name structure to contain all of the CS8 computed output files. With one exception (see _Ref_*filename.xxx*), all files in this folder are erased and recreated each time the case simulation is run. These files may be deleted without harm.

_Ref_*filename.xxx* – Files with names of this form (beginning with _Ref_) are not erased when the case simulation is run even when they are in the case folder. These files are for use as reference curves in plots, for instance when you want to plot test data against a simulation. The two possibilities for "*xxx*" are "csv" in the case of a file in the CS8 output format, and "txt" if the file is in the format produced by earlier versions of the Control Simulator (CS4 and CSI).

***devicename (N)-function*.csv** – Within the case folder *(projectname) casename*.cso, the compute engine creates a number of output files with names of this structure. "*devicename*" is the name of the device as it appears above the icon in the case window.

The device names are not necessarily unique (though you should strive to make them so). In order to insure that each device has a unique output file name (and a unique section in the cs8case file), CS8 assigns a unique serial number to each device as it is created. That number is the "*N*" in the file name and is permanently assigned to that particular device.

Each device has its own "*function*" names for its output files. Function names identify what part of the device this output applies to, such as "Accum".

***devicename (N)*-Header.csv** – Each device may have a header file that is used for echoing input data and for displaying other computed output that is not a function of time.