Surge for Pipe2010

Users Guide

Proven … Solutions … Software … Support

October 2009

LICENSE AGREEMENT

This is a legal agreement between the user and KYPipe LLC. By accepting, using or installing any portion of this software the user agrees to be bound by the terms of this agreement.

SOFTWARE LICENSE

GRANT OF LICENSE: For each license purchased from KYPipe, LLC, or one of its authorized distributors, KYPipe LLC grants to the user the right to use one copy of the software program(s) on a single terminal connected to a single computer (i.e., with a single CPU). The user may not network non-network versions of the software or otherwise use single user versions on more than one computer terminal at the same time. Network versions are only to be used with one physical site (buildings at the same mailing address) and are not to be used in a WAN environment. The number of network licenses purchased for a network version is the maximum number of users permitted to run the software concurrently. If granted for an evaluation period by KYPipe LLC, user agrees not to use the software beyond the evaluation period specified by KYPipe LLC. The user agrees not to utilize features, options, or number of pipes beyond the license the user has purchased.

COPYRIGHT: The software and the documentation are owned by KYPipe LLC and are protected by United States copyright law and international treaty provisions. The user must treat the software like any other copyrighted material except that the user may make one copy of the software solely for backup or archival purposes or may transfer the software to a single hard disk and keep the original disk(s) sole for backup or archival purposes. The user may not copy the written materials accompanying the software without explicit written permission from KYPipe, LLC.

TRANSFER BY USER: The user may not rent, lease, assign or permit others to use the software but may transfer the software and accompanying materials on a permanent basis provided the user retains no copies and the recipient agrees to the terms of this agreement. As a condition to permit the recipient use the software under this License Agreement, when such a transfer is made, KYPipe LLC must be notified, in writing, of the transfer, including the identity and address of the recipient, and the agreement of the recipient to the terms of this License Agreement.

OTHER RESTRICTIONS: The user may not modify the software. The user may not reverse engineer, decompile, disassemble, or otherwise attempt to determine the source code of the software. The user shall protect the software from unauthorized use, and shall protect the software and the intellectual property from infringement by others. The user shall notify KYPipe, LLC, in writing, immediately upon receiving any information that would indicate that the software is being used in an unauthorized manner or the intellectual property is being infringed.

DISCLAIMER: Although every reasonable effort has been made to ensure that the results obtained are correct, neither the author(s) nor KYPipe LLC assumes any responsibility for any results or any use made of the results obtained with these programs. THE SOFTWARE IS SOLD AS IS WITH NO IMPLIED WARRANTIES, INCLUDING WARRANTIES OF MERCHANTABILITY AND FITNESS FOR ANY PARTICULAR PURPOSE. NO EXPRESS WARRANTY EXISTS EXCEPT AS SPECIFICALLY SET FORTH IN WRITING BY KYPIPE, LLC. IN NO EVENT, REGARDLESS OF THE NATURE OF ANY CLAIM, WILL KYPIPE, LLC, ITS MEMBERS OR AFFILIATES, BE LIABLE FOR ANY LOSS FOR PERSONAL INJURY, BUSINESS INTERRUPTION, LOST PROFITS, OR INCIDENTAL OR CONSEQUENTIAL DAMAGES, AND ITS LIABILITY, IF ANY, SHALL BE LIMITED TO THE PURCHASE PRICE OF THE SOFTWARE.

USE OF THE DOCUMENTATION AND PROGRAM: The documentation is provided for the use of individuals or companies which purchase it from KYPipe LLC. Except for back-up copies, the program disks or documentation may not be copied, marketed, or distributed without explicit written permission from KyPipe, LLC. For users who wish to use the programs on networks or multiple computers or different locations, network copies and multiple copy discounts may be obtained. Please contact KYPipe LLC for details.

GOVERNING LAW AND VENUE FOR ENFORCEMENT AND DISPUTES: This Agreement will be governed by and construed in accordance with the substantive laws of the Commonwealth of Kentucky, and, to the extent federal law applies, to the laws of the United States. The state and federal courts of Fayette County, Kentucky, shall have exclusive jurisdiction over any claim brought against KyPipe, LLC, and the user agrees to submit to the jurisdiction of the state and federal courts of Fayette County, Kentucky, in the event any claim is brought against the user, and user waives all defenses to jurisdiction and inconvenience of forum.

Chapter 1: Introduction to Pipe2010: Surge

Pipe2010: Surge is the Pipe2010 Series Transient Flow Model. Some System Data input and other modeling variations differ from the Pipe2010: KYPipe water model. These differences are discussed in Chapters 2-6.

An Examples Manual is available for download on the KYPipe web site, www.kypipe.com. This contains a section with numerous examples of surge analysis,

To view **Audio/Video Tutorials for Surge**, insert the **Pipe2010 CD** and select **Start Tutorial**. Select **Surge**. Then the user may either play the videos in order (this includes basic getting-started and modeling information) or click **Select Video** to choose from the following Surge-specific tutorials:

Intro to Surge Analysis – 1

Intro to Surge Analysis – 2

Pipe2010 : Surge Geometric Requirements

Pipe2010 : Surge Components

Converting KYPipe files to Surge

Surge Control Devices

Features for Pipe2010 : Surge Components

Surge Control Components

Variable Input Data

Surge Analysis Example

Adding Surge Protection to a Model

See also the Pipe2010: Surge Demonstration files, DEMOSURG.p2k and SURGTANK.p2k, in the Demo folder in the Pipe2010 directory.

Appendix 1 provides an overview of the Wave Method used for transient analysis and provides demonstrations of the accuracy and computational advantages of this approach,

Appendix 2 provides some useful tips and procedures for surge modeling using Pipe2010 : Surge

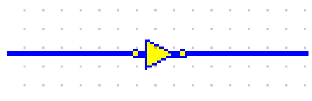
Chapter 2: Pipe2010: Surge Modeling Requirements

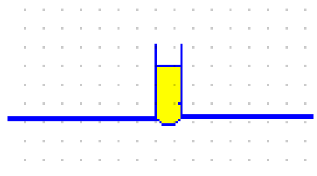
Using Pipe2010: Surge

Modeling Requirements

A Surge model is comprised of pipe sections connecting junctions, components, and surge control nodes. The following restriction applies:

Only two pipe connections are allowed for regulators and surge control devices. These include side discharge orifice's (SDO's), surge tanks, relief valves, sprinklers, surge anticipation valves and air/vacuum valves.





The Surge error check will check these requirements and produce an error message for any requirements which are not met.

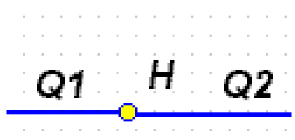
Chapter 3: Pipe2010: Surge Calculations and Sign Conventions

Calculated Parameters

For surge analysis, calculations (heads, flows, volumes) are made at each element for each time step. This produces very large files of results. Elements consist of junctions (including reservoirs and dead ends), components (usually pumps and valves) and surge control devices. For each time step the following values are calculated for each element. Note that only flow results are presented for pipes. This represents flow at the midpoint of the pipe. This is because flow varies along the pipe.

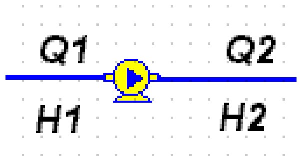
junction: head/pressure (one value)

flow (each connection)



component head/pressure (each side)

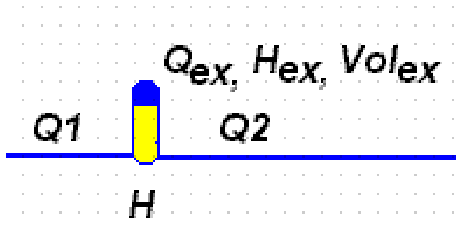
flow (each side)



surge control (SDO) head/pressure (inside and external)

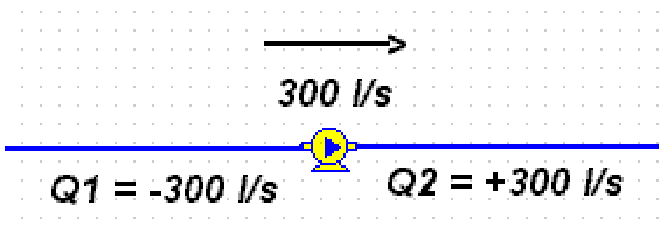
flow (side 1, side2, external)

air or cavity volume (external)

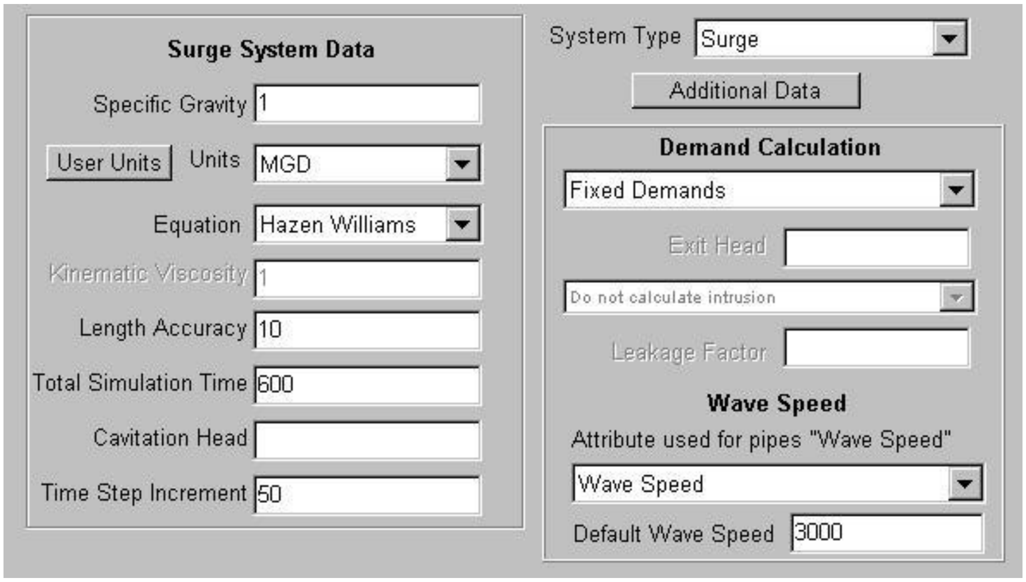


Sign Convention for Flow

Flow away from a node is positive and flow toward the node is negative. For example, if 300 l/s is flowing in the normal direction through a pump the outlet flow is +300 l/s while the inlet flow is -300 l/s. Therefore, when plotting and tabulating results you can choose to utilize the outlet value to get positive flow. See Surge Node Results



Chapter 4: Pipe2010: Surge - System Data / Simulation Specs



Specific Gravity - ratio of density of liquid to density of water. See also Specific Gravity.

Units - Flow Units CFS (cubic feet/second)

GPM (gallons/minute)

MGD (million gallons/day)

Liters/Sec (liters/second)

CMS (cubic meters/second)

Liters/Min (liters/minute)

Lb/s (pounds/second)

BPH (barrels/hour)

kg/s (kilograms/second)

USER (user defined units)

Equation - Head Loss Equation. Choose between Hazen Williams, Darcy Weisbach and Constant Resistance. The appropriate roughness must be entered (Pipe Information - Data) for the equation chosen. See Hazen Williams Table and Darcy Weisbach Table. For Constant Resistance the resistance is defined as the head drop (ft or m) divided by the flow squared (CFS or CMS)

Kinematic Viscosity - Required only for Darcy Weisbach equation (ft^2/s or m^2/s).

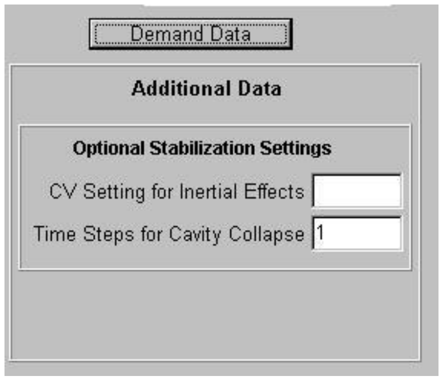
Length Accuracy - This input defaults to 10 and represents the maximum differences between the actual pipe lengths and the ones chosen for the model. Note the calculation time increment and required computational time are affected by this selection and decreasing the length accuracy by a factor of two will double the required computational time.

Cavitation Head - This defaults to values applicable at standard pressure and can be entered here to override the default.

Time Step Increment - This must be an integer and will result in a smaller time increment being used. New time increment = calculated time increment / time step increment. Hence the input 5 will result in a new time increment on 1/5 of the original and will increase the required computational time by a factor of 5. This input may be required for high frequency periodic disturbances.

System Type - Should display 'Surge' as selection

Additional Data - This will bring up the following box.



CV Setting for Inertial Effects - CV Setting is a number between 0 and 1. The default is 0 and will result in no inertial effects. If the CV is in the closing mode (reverse flow through the element) and the CV open ratio is greater than CV Setting, then we continue to close the CV until it is completely closed EVEN if the flow changes sign (flow in the forward direction). In other words, we are assuming that the CV continues to close once it reaches certain closing position due to the inertial effects of the disc or flap. The use of this setting will reduce pressure spiking due to check valve action.

Time Step for Cavity Collapse - Time Step for Cavity Collapse is an integer representing the number of time increments for complete collapse of a vapor cavity. The default is 1, implying that the cavity collapses in one time increment, however small the time increment is. The purpose of this setting is to “soften” the effects of cavity collapse. Since air is normally released when cavitation occurs a softening effect is expected. Without allowing for this excessive pressure spiking may occur.

Demand Calculation - Choose between Fixed Demands and Pressure Sensitive Demands. For Fixed Demands the demands remain constant throughout the transient analysis and no additional input is required. For Pressure Sensitive Demands the demands vary with the pressure difference in the pipe and the exit region.

Exit Head - The assumed head in the exit region for pressure sensitive demands. This defaults to zero (atmospheric pressure).

Calculate Intrusion - If Pressure Sensitive Demands are selected, three options are available:

Do not calculate intrusion

Calculate intrusion using a leakage factor

Calculate intrusion using a leakage constant.

If one of the last two are selected then intrusion of liquid back into the pipe system will be calculated when the exit head exceeds the head inside the pipe. The user will need to then enter the Leakage Constant or Leakage Factor for that calculation. See below.

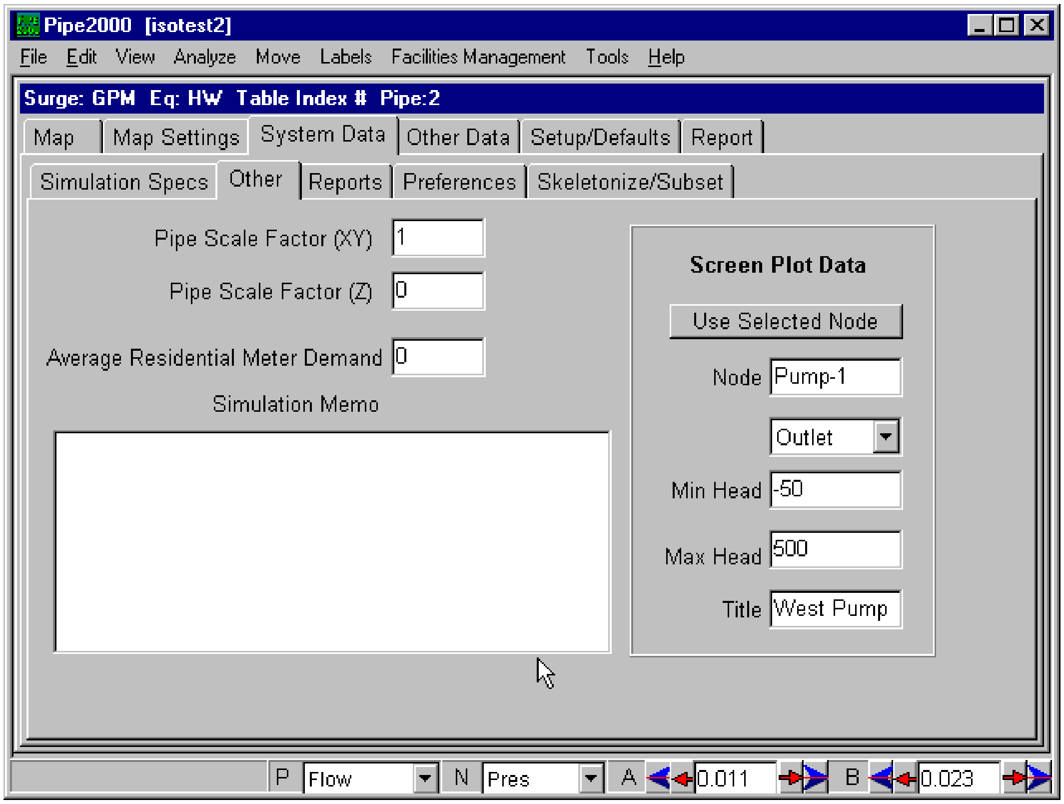
Leakage Coefficient - If intrusion is calculated, this input determines the ratio of flow which can intrude for the same differential head. For example, a Leakage Coefficient of 0.1 will calculate an intrusion rate 0.1 times as great as the corresponding outflow at the same head difference. The

Leakage Coefficient will correlate to the lost water percent.

Wave Speed Attribute - This should normally display the User Data "wave speed" although another User Data attribute can be used (but this is not recommended). For new files this should automatically appear. For imported files the "wave speed" attribute may not be defined and will not appear on the drop down list of User Data attributes. In this case the user should access the User Data and make a "wave speed" attribute and then select this in the drop down list.

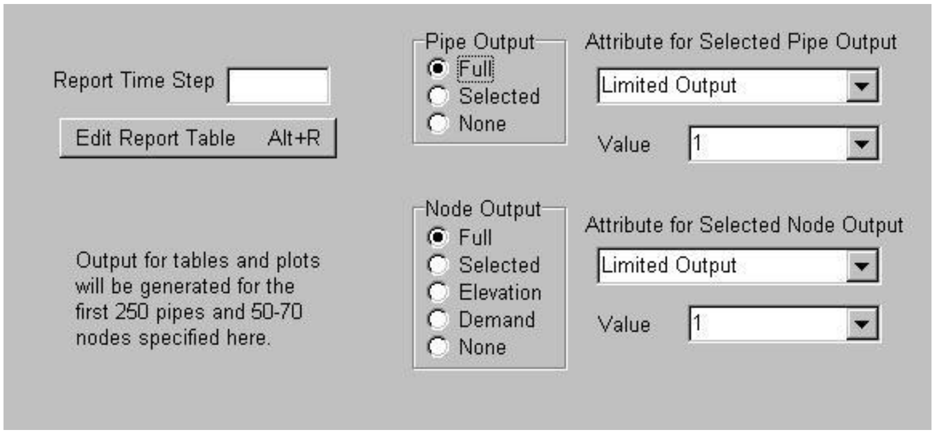
Default Wave Speed - If a value is entered here, this will be used for all pipes for which a wave speed is not specified in the Pipe Information window data box (Map Screen) or in the Wave Speed User Data item defined in the Wave Speed Attribute field above.

Chapter 5: Pipe2010: Surge - System Data / Other



Screen Plot Data - It is useful to designate a node for the screen plot which appears as Surge is executing. The best way is to select one node in Layout Mode and then click on **Use Selected Node**. Then fill in the rest of the data (Title is optional). This entry produces a real time plot on the screen as the surge analysis proceeds allowing the user to observe the progress of the analysis.

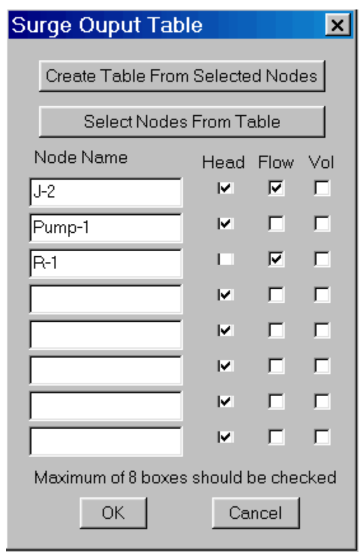
Chapter 6: Pipe2010: Surge - System Data / Reports



The Reports screen allows the user to designate desired outputs. The Report Table appears in the Surge Tabulated Output report and is limited to 8 items.

Report Time Step - this is the time step used in the Tabulated Output. This defaults to the computational time step which is usually very small. Using this default will result in a very long report.

Edit Report Table - Check on this to bring up the following screen. It is best to first go into Group Mode and select up to eight nodes for this report. Then click on **Create Table from Selected Nodes**. The nodes will appear and you can select the desired output parameters.

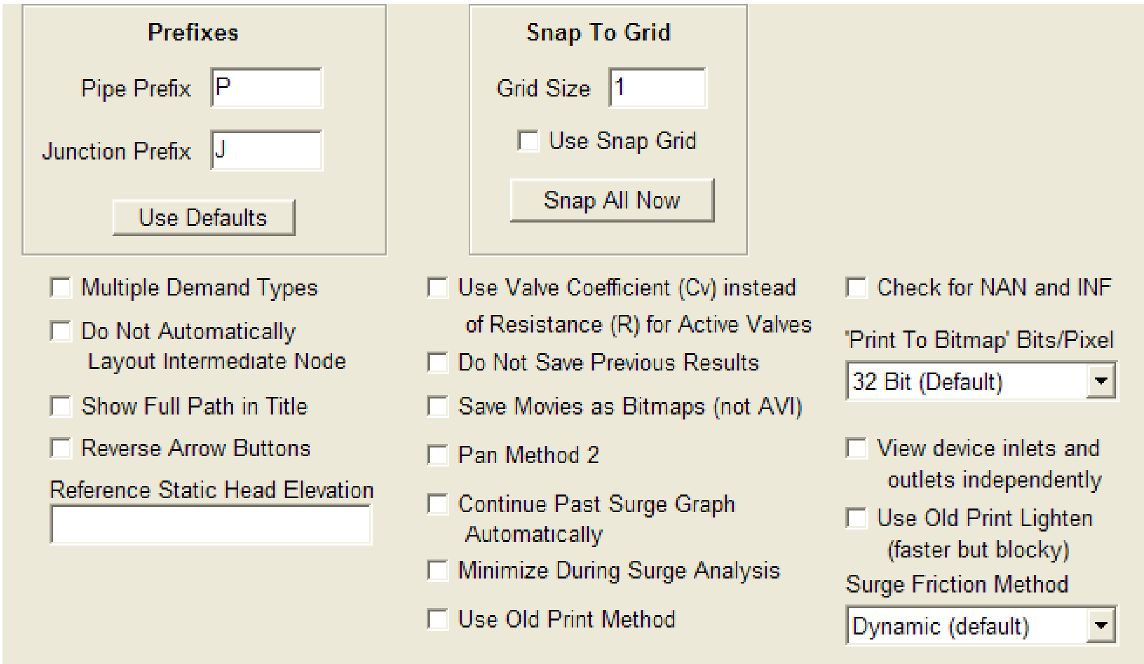


The Node Output options are used to select the nodes which can be accessed for customized plots and tables. Because of the very large amount of results data, Versions 1 and 2 output is limited to 70 nodes. Version 3 and beyond, however, provide all results up to 4000 nodes. For Versions 1 and 2 or if you have > 4000 nodes you should designate one of the following:

Full - all nodes are selected. If there are more than 70, the first 70 will be used.

Selected - only nodes selected using the designated User Data attribute will have output. Normally the attribute "Limited Output" is selected and the value 1 used to designate the included nodes.

Chapter 7: Pipe2010: Surge - System Data / Preferences



There are a few Preference items which are especially applicable to Surge models. They are as follows:

Use Valve Coefficient (Cv) instead of Resistance (R) for Active Valves

When this box is checked a valve coefficient, normally provided by the manufacturer, may be used. See Active Valves.

Do Not Save Previous Results

Previous results are automatically saved unless this boxed is checked. Saving or not saving previous results effects file size. It may be desirable to check this box in particular with Surge files which have a large amount of results data. If previous results are saved, they may be viewed in the Node/Pipe graphs or table. See Node Results Boxes or Pipe Results Boxes.

Continue Past Surge Graph Automatically

When a Surge analysis is conducted a screen plot is drawn. When the plot is completed the user is by default prompted to 'Click here to continue" with analysis. Checking this box will bypass the user prompt, closing the screen plot and continuing on with analysis.

Minimize During Surge Analysis

Certain graphics cards cause messages generated during a Surge analysis to be hidden behind the Pipe2010 window, giving the appearance that Pipe2010 has locked-up. Check this box to avoid this situation.

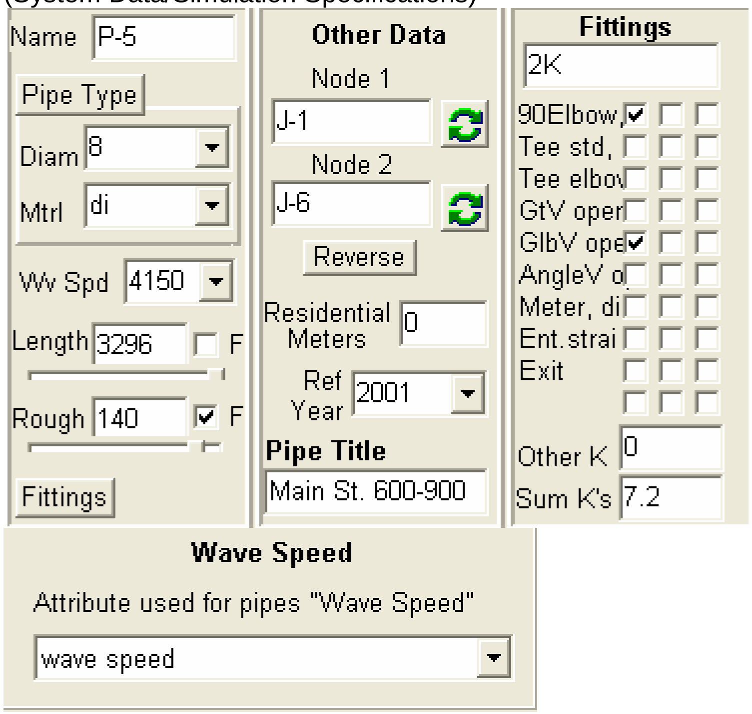
Surge Friction Method (V3)

For Version 3 the use of dynamic friction calculations were introduced and this selection is recommended. This approach will give better hydraulic results for final steady state conditions.

Chapter 8: Surge Elements and Data Requirements

Pipes

Set User Data item for wave speed and then select wave speed in the box shown below right (System Data/Simulation Specifications)



Pipe Data / Units
Diameter (Diam) – in. or mm
Wave Speed (Wv Spd)– ft./s or m/s.
Length – ft. or m.
Roughness – depends on HL equ.
Residential Meters – Number of users in the pipe.
Fittings data – sum of K's

Pipes connect two nodes and can have intermediate nodes for alignment changes. Because of the importance of the location of closed valves, **for Surge models pipes cannot be closed.** Closing pipes must be done using a closed Active Valve. Also for Surge models pipes do not incorporate check valves as can be done in steady state models. **For Surge models, Check Valves are designated at nodes** (such as Pumps and Active Valves).   See more below.

Check Valves

Pumps, active valves, and loss elements all can be equipped with a check (non-return) valve to prevent backflow. Either a normal check valve which will close or open depending on conditions or a non-reopening check valve which will close only once and remain closed can be designated. The closing time (CV Time) is the time it takes for the check valve to close once closing is initiated. The check valve resistance (CV Res) is the resistance (head/flow^2) when the valve is fully open. If a Check Valve is closed for the steady state then the element which incorporates the Check Valve will be initially closed. If conditions are appropriate the element Check Valve will open during the transient analysis.

Pipe2010: Surge does NOT permit modeling stand alone check valves - unlike KYPipe where a pipeline can have a stand alone check valve. When there is a standalone check valve on a pipeline, then the user must model that as an active valve with a check valve. An active valve by itself may or may not have a check valve. When a KYPipe model that has standalone check valves is converted to Pipe2010: Surge model, the program deletes the standalone pipe check valves and produces warning messages.

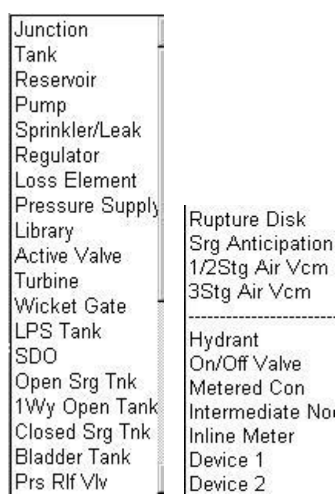
See Chapter 4: System Data / Simulation Specs for information on CV Settings for inertial effects

Bypass Lines

A pump bypass line is one which will open when the suction (upstream) head exceeds the discharge (downstream) head. The bypass line resistance (Byps Res) is the resistance of this pipe.

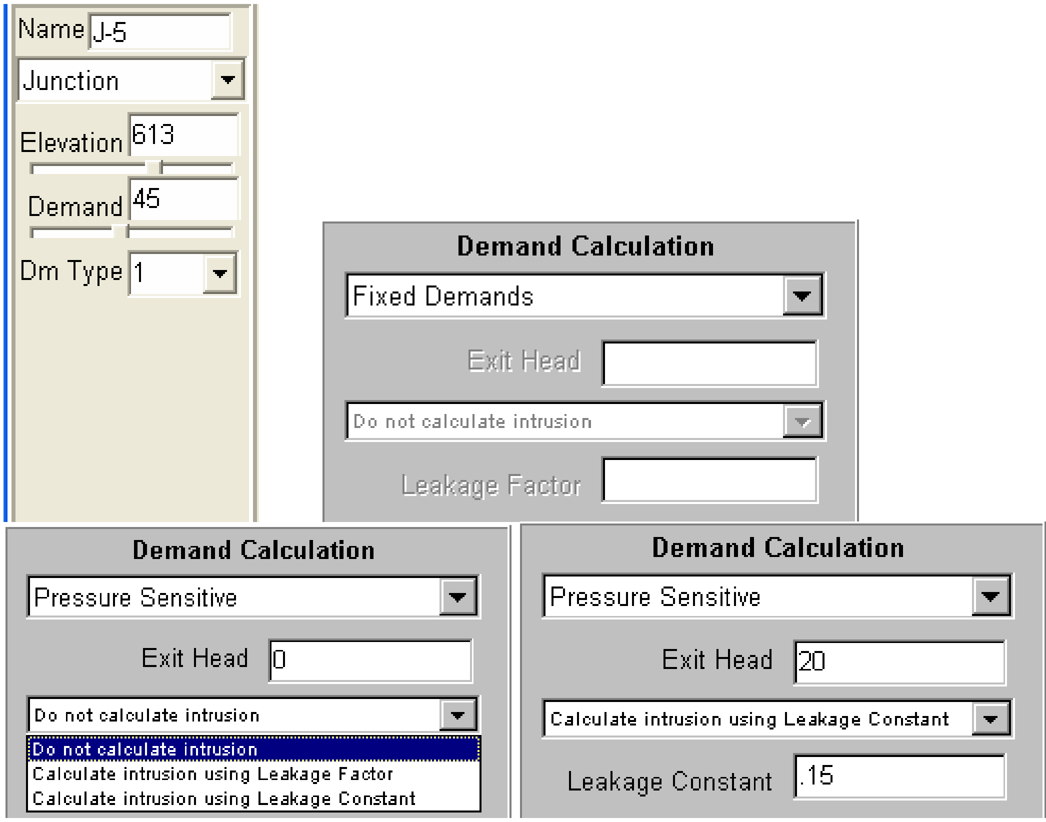
A valve bypass line is used to define a different resistance when flow reverses and is assumed to flow in the bypass line.

Nodes



The list above shows all the devices that can be included in a Surge model. Some of these devices are utilized in both the initial steady state and the surge analysis. These devices have specific modeling requirements (number of pipe connections) for Surge which are illustrated in the network shown below and discussed on the following pages. Each of these devices requires unique input data, which includes the elevation (in feet or meters). The additional data requirements for each device are presented on separate pages along with notes regarding their use.

Junctions



Demands – each junction may be assigned a Demand by the user or through meters connected to pipes connected to the junction. The demand type, **Dm Type,** is not utilized in the surge analysis.

Connection Restrictions: No restrictions for junctions

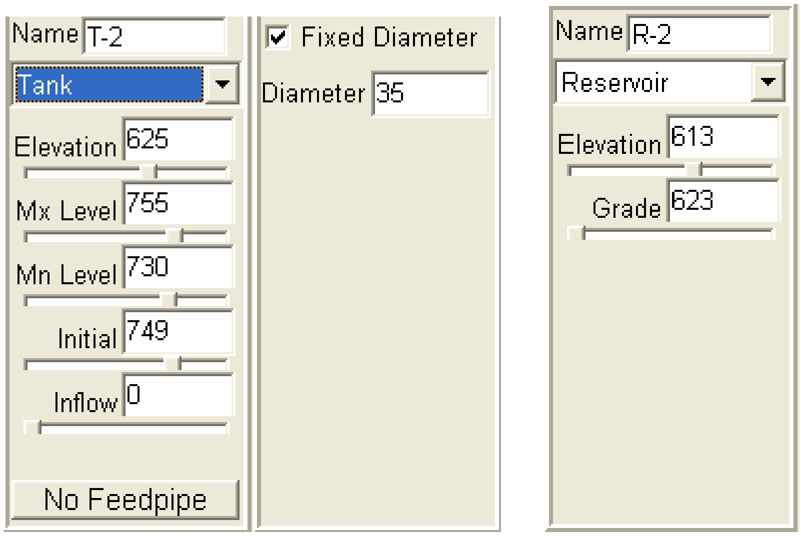
Under System Data | Simulation Specs:
Demand Calculation – select calculation method:

- **Fixed Demands** – demands remain constant throughout transient analysis
- **Pressure Sensitive Demands** – demands vary with pressure according to the orifice relation

Intrusion Calculation – select calculation method

- Do not calculate intrusion
- Calculate using Leakage Factor or Leakage Constant

Reservoirs and Tanks



Reservoir and Tank Data / Units
On/off switch
Grade – ft. or m.
Mx, Mn, Initial (level) – ft. or m.

Connection Restrictions: Only one pipe can connect these devices

Reservoir – A constant grade is maintained at a reservoir. The grade maintained is input. **Grade is defined as elevation plus pressure head (in feet or meters).**

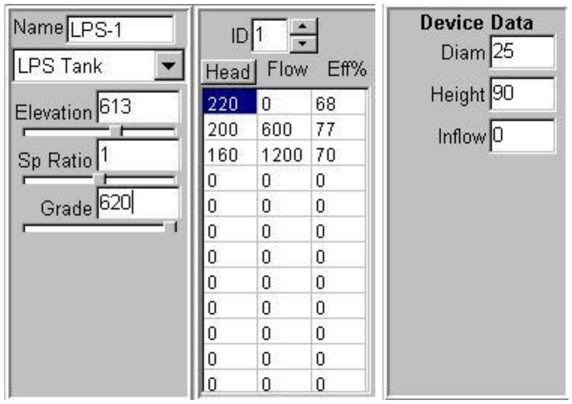
Tank – A supply tank is treated as a constant head reservoir for surge analysis. Besides the Elevation, the (Initial) grade (elevation of level) is the only input value required for a tank. A Feedpipe is a steady-state feature, which does not apply to surge analysis.

LPS Tanks are constant diameter tanks with an ID Pump (variable speed) situated in the bottom of the tank. This is a discharge-only element. When the element is selected, the results which are available include outlet pressure, head, and flow.

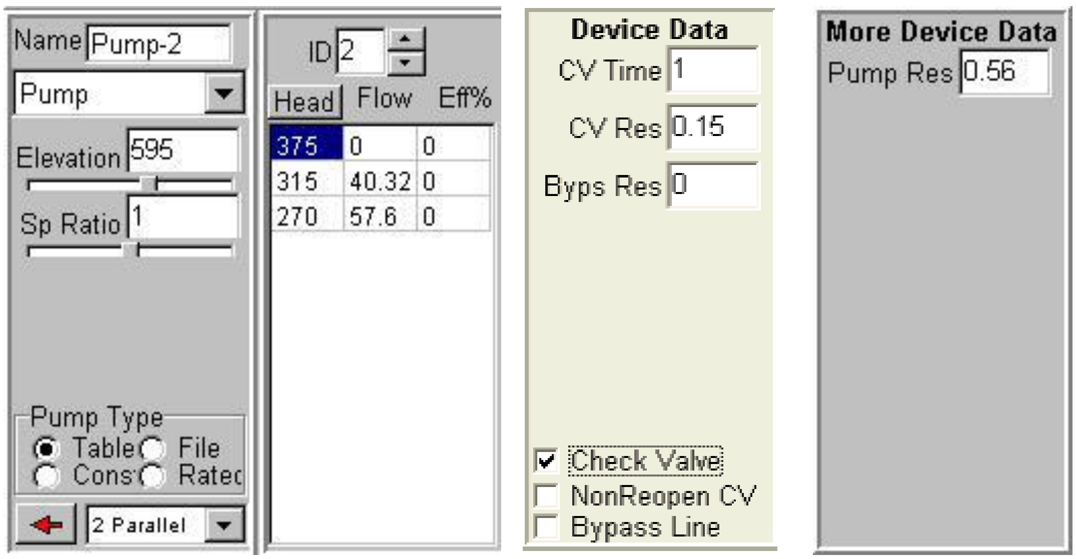
For the tank, instead of maximum and minimum elevations as with regular tanks, the height above the applied elevation is defined, along with diameter. Inflow may also be defined if applicable.

For the internal pump, the pump speed, the grade from which the pump is supplied, and the pump ID are all defined.

To view tank levels for LPS tanks, go to Report and view the Tank Report.



Pump Table (Head/Flow Data)



Pump Table Data / Units
On/off switch
Speed (ratio) – rpm/rated speed
Head – ft. or mt.
Or
Pressure – psi or kPa
Flow – specified flow units
Eff(iciency) - %/100 (not used in Surge)

Connection Restrictions: one pipe connected to each side or one pipe connected to one side and reservoir on other side

This Pump Type is selected as Table. The head (pressure)/flow table should include exactly 3 data points including cutoff head (at zero flow). If more data is provided or is brought in when a KYPipe file is imported to Surge, only the first three data points will be utilized in the surge analysis.

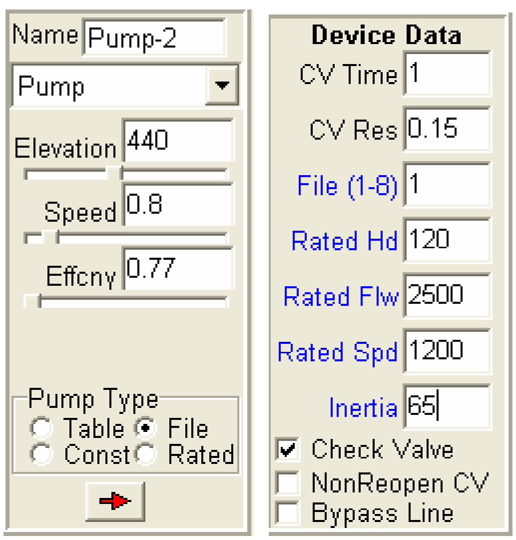
Pumps may be specified as multiple pumps (e.g. 2 parallel pumps). Note that results are for combined pump configuration, not for individual pumps.

All pumps can include a Check Valve, Non Reopening Check Valve and/or Bypass line. These options perform the following functions:

Check Valve – prevents flow reversal through the pump
CV Time = time it takes check valve to fully open (or close) after flow reversal initiates
CV Res(istance) = wide open resistance for the check valve
Non-Reopen(ing) CV – This check valve will not open once it has closed.
Bypass Line – This allows flow to bypass the pump when the suction head exceeds the discharge head.
Byps Res(istance) = resistance of the Bypass line
Pump Resistance - With parallel or series pumps, it may be desired to define a resistance to account for pump piping. For this purpose, Pump Res in the More Device Data box may be used (click the 'More' button or pointing hands to view, if necessary). A pump resistance for each pump may be defined in units of headloss/(flow)^2. A special tool is available to calculate the resistance, Resistance Calculations. In the Resistance Calculation Tool, specify 'Piping for Parallel and Series Pumps.'

Since only normal conditions (positive head, positive flow) are described by the data, transient analysis should be limited to situations where the pumps operate under normal conditions. If abnormal conditions occur (reverse flow, turbining, etc.) a pump file (below) should be used.

Pump File



Pump File Data / Units
On/off switch
Speed (ratio) – rpm/rated speed
Efficiency - %/100 (at rated flow)
File (number) – 1 to 8
Rated Head – ft. or mt.
Rated Flow – specified flow unit
Rated Speed – rpm
Inertia - # ft2 or nmt2

Connection Restrictions: one pipe connected to each side or one pipe connected to one side and reservoir on other side

The **Pump Type** is selected as **File**. This description should be employed if the pump will be operated abnormally during the transient (flow reversal, turbining, etc). The other three descriptions should be used for transient analysis only if the pumps always operate in the normal zone of operation (positive head-positive flow). To use this description, you must select file (1-8) based on the specific speed (use the **Select Pump File** tool) and provide the rated head (feet or meters), flow and speed. The motor and pump inertia is normally difficult to obtain from the manufacturer and a reasonable estimate can be calculated using **Inertia/Specific Speed** tool.

A pump file is a table of values defining head and torque as a function of flow and speed. These are based on experimental data and 8 files are available to use. The file selection is based on specific speeds and the following files are available.

File number	specific speed	
	English	Metric
1	1270	25
4*	3725	72
5*	4409	85
6*	5203	101
7*	6792	131
2	7600	147
8*	8764	169
3	13500	261

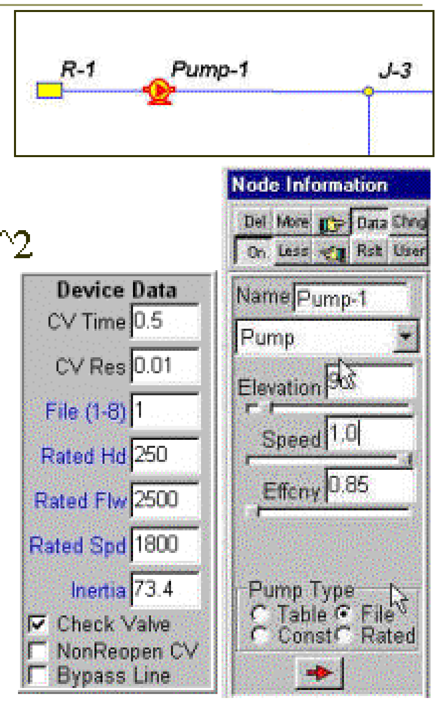
These files were selected to cover the range of specific speeds for which data is available. This specific speed is defined as

specific speed = (Nr (Qr^0.5))/(Hr^0.75)

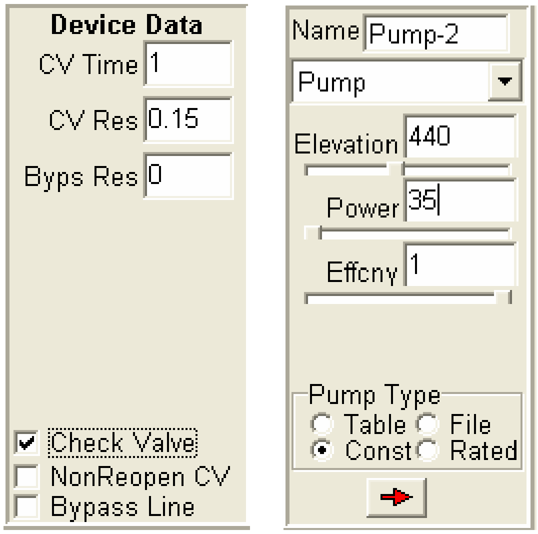
*files 1, 2, and 3 were available for versions of SURGE prior to SURGE 5 and files 4 - 8 were added to version SURGE 5.1.

The pump file is customized to your pump by providing the Rated Head, Rated Flow, Rated Speed and Inertia. See the Surge Tools for help in calculating the specific speed and inertia. The use of this tool is illustrated below for this example.

- File number = 1 (for 9991)
- Rated Head = 250ft
- Rated Flow = 2500gpm
- Rated Speed = 1800rpm
- Pump+Motor Inertia=73.4 lb-ft^2
- Pump Efficiency = 85%
- Initial Pump Speed Ratio = 1.0
- Elevation = 90feet
- Check valve resistance=0.01
- CV open time = 0.5sec



Pump (Constant Power)

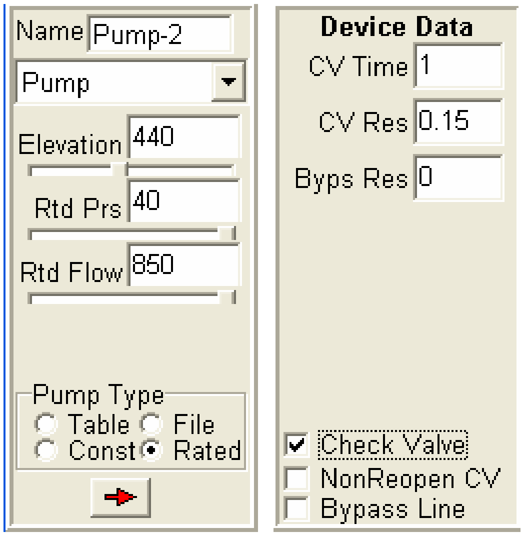


Constant Power Pump Data / Units
On/off switch
Power – horsepower or kilowatts
Efficiency - %/100 (at rated flow)
(not used in Surge)

Connection Restrictions: one pipe connected to each side or one pipe connected to one side and reservoir on other side

This **Pump Type** is selected as **Constant.** All the user is required to enter is the useful power (horsepower or kilowatts). For the surge analysis a head/flow curve is generated based on using the steady state operating point as rated conditions employing the same procedure as that described for the Rated Pump. The constant power pump description is not recommended for Surge Analysis because of the assumptions required to apply it. This description also cannot be used for pumps that are initially off since the appropriate rated conditions cannot be determined.

Pump (Rated)



<u>Rated Pump Data / Units</u>
On/off switch
Rated pressure – psi or kPa
Rated flow – specified units

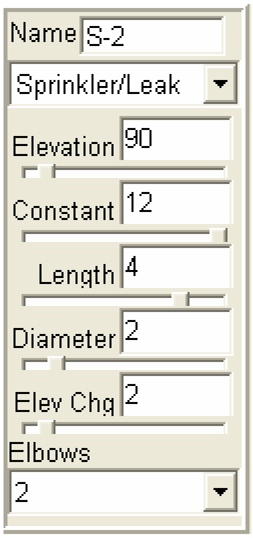
Connection Restrictions: one pipe connected to each side or one pipe connected to one side and reservoir on other side

This **Pump Type** is selected as **Rated**. The user is required to enter the Rated Pressure (in psi or kPa) and Rated Flow (selected flow units). A normal operating range pump curve is generated using the following pressure/flow data:

Pressure, P_r	Flow, Q_r
1.4 P_r	0
P_r	Q_r
.65 P_r	1.5 Q_r

where P_r is the rated pressure and Q_r is the rated flow.

Sprinkler/Leak



<u>Sprinkler Data / Units</u>
On/off switch
Constant – flow in gpm for 1 psi
or
flow in l/s for 1 kPa
Length – (of connector) ft. or m
Diameter – (of connector) in. or mm
Elevation Change – ft. or m
Number Elbows in connector

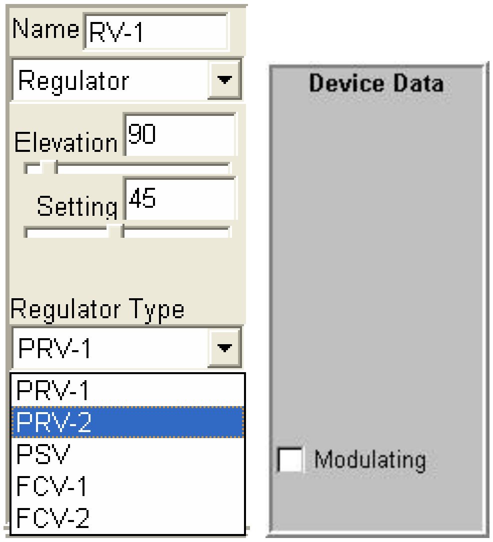
Connection Restrictions: one pipe connected to each side, cannot be at the end of a line.

This element models flow through a sprinkler orifice. The flow obeys the classic sprinkler relation where Q is the flowrate (in gpm or liters/s), K_s is the sprinkler constant and ΔP is the pressure difference (in psi or kPa) between the inside and exit of the sprinkler.

$$Q = K_S\sqrt{\Delta P}$$

A connecting pipe between the pipe in the model and the sprinkler orifice can be modeled by inputting data for the characteristics of the connections (length, diameter, elevation change from pipe centerline to orifice (negative if sprinkler is above pipe centerline) and number of elbows in the connection.

Regulators



Regulator Data / Units

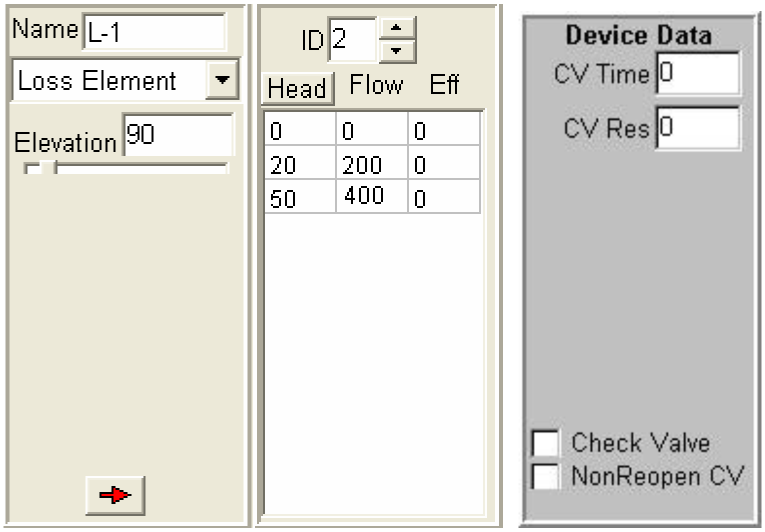
Setting	
PRV	psi or kPa
PSV	psi or kPa
FCV	specified flow

Connection Restrictions: one pipe connected to each side

This element models flows through regulating valves (pressure regulating, pressure sustaining and flow regulating). You can select PRV-1 (pressure regulating), PSV (pressure sustaining) and FCV-1 (flow regulating) from the drop down list. The other two (PRV-2 and FCV-2) should not be selected since they model abnormal conditions.

For transient analysis, the regulating valves are modeled as a constant resistance based on the steady state conditions. If the regulator is wide open or closed for the steady state it will remain in the same condition for the transient analysis. If you wish to model a regulating valve, which changes its stem position during the transient analysis, you need to use an Active Valve. *** modulating regulator, fix in Help also.

Loss Element



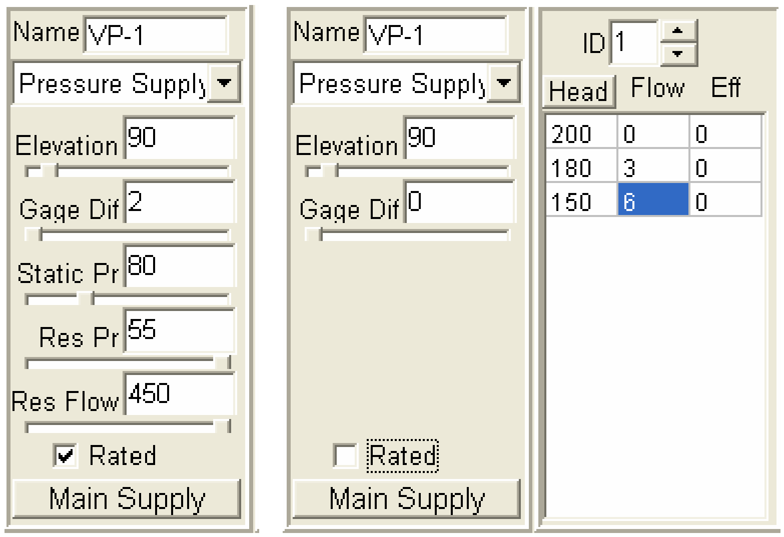
Loss Element Data / Units
On/off switch
Head loss – ft. or m
or
Pressure loss – psi or kPa
Flow – specified units
Eff(iciency) – not applicable

Connection Restrictions: one pipe connected to each side

This element models a device for which head loss/flow data is available. The modeling is identical to a pump described by head/flow data except that a head loss (instead of a head gain) occurs across the element. The Loss Element can have a Check Valve to prevent flow reversal.

See Pumps for information on using check valves.

Pressure Supply

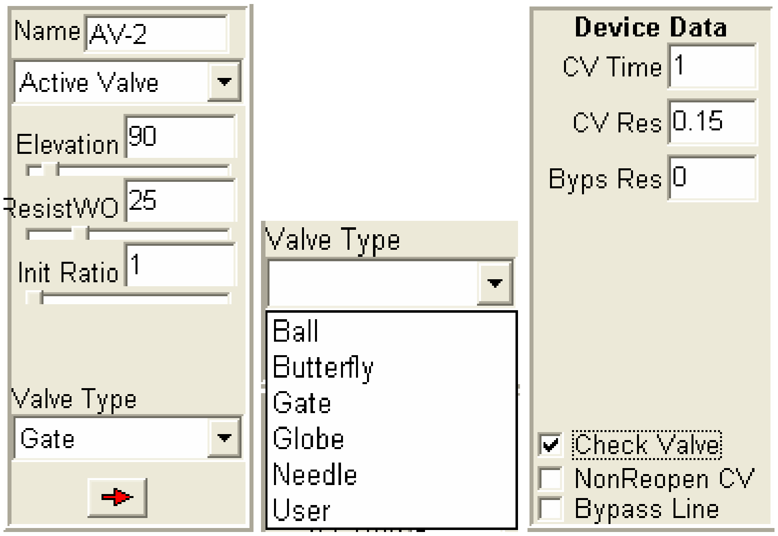


<u>Pressure Supply Data / Units</u>
On/off switch
Gage Dif(ference) – distance gage is
above CL pipe (ft. or m)
Static Pressure – psi or kPa
Residual Pressure – psi or kPa
Residual Flow – specified flow units

Connection Restrictions: only one pipe connected to this device

This element models a connection to a supply where the available pressure depends on the supply flow. This applies to a connection to an existing distribution system. The data required to describe the pressure/flow relation for this supply may be provided by a head (or pressure)/flow table or by hydrant test data (static pressure and residual pressure and flow) for a hydrant close to connection. For Surge analysis the same pressure/flow relation is maintained.

Active Valve



SRatio	0	.1	.2	.3	.4	.5	.6	.7	.8	.9	1
Ball	0	.043	.1136	.2031	.3062	.4188	.5371	.6578	.7773	.8924	1
Butterfly	0	.0123	.0489	.1090	.191	.2929	.4122	.546	.691	.8436	1
Gate	0	.1271	.2529	.3762	.4954	.6090	.7152	.8119	.8959	.9626	1
Globe	0	.1	.2	.3	.4	.5	.6	.7	.8	.9	1
Needle	0	.19	.36	.51	.64	.75	.84	.91	.96	.99	1
User	0	.1	.2	.3	.4	.5	.6	.7	.8	.9	1

Active Valve Data / Units
On/off switch
ResistWO (wide open resistance) - H/Q2
Initial Ratio (stem) - % open/100%
Valve Type – select from drop down table

Connection Restrictions: one pipe connected to each side or one pipe connected to one side and reservoir on other side

This element models a variety of valves. A valve is described by the Wide Open Resistance (ResistWO = head loss/flow2) and the data in the Active Valve Table shown above (Other Data | Active Valves). This table gives the open area ratio (open area/fully open area) as a function of ratio of the stem movement to the full movement (S(tem) Ratio). This table shows, for example, a ball valve where the stem position is half way to full amount (0.5) the open area is 41.88% of the fully open area. Users can create their own valves. For Surge analysis, valve closings and openings are modeled by providing the time history of the stem movement. For example, a Ball Valve can be linearly closed in 4 seconds, which means that the stem movement is linear.

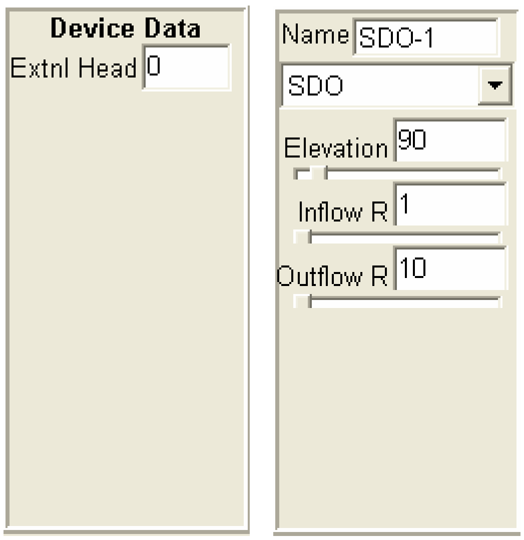
Init Ratio is the ratio of current stem position to the fully open stem position. For ball and butterfly valves the stem position is expressed as an angle. For all other standard valves, the stem position is expressed at the translational distance of the valve stem. For partially closed valves, specify the appropriate stem position ratio at the beginning of the simulation, e.g. if the stem position is half way the Init Ratio = 0.50, if the stem position is 3/4 closed then Init Ratio = 0.25, if the stem position is completely closed, then Init Ratio = 0.

See **Check Valve** for information on using an Active Valve to model a Check Valve.

Surge Control Devices

Note on Surge Control Devices: For surge tanks and relief valves the input data includes the inflow and outflow resistance. As always, the resistance is defined as the head drop (ft or m) divided by the flow squared (cfs or cms). The resistance depends on the type of connection (pipe section or orifice) and can be calculated from the characteristics of the connection. See the Surge Tools – **Resistance Calculation Tool** for help with the calculations.

Side Discharge Orifice (SDO)



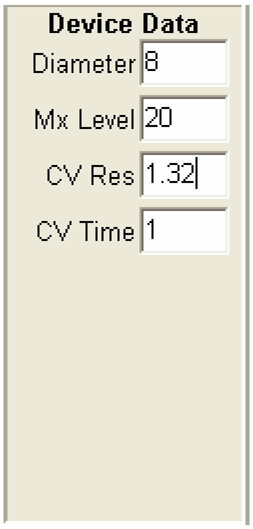
SDO Data / Units
On/off switch
Inflow R(esistance) - H/Q2
Outflow R(esistance) - H/Q2
External Head – ft. or m

Connection Restrictions: one pipe connected to each side

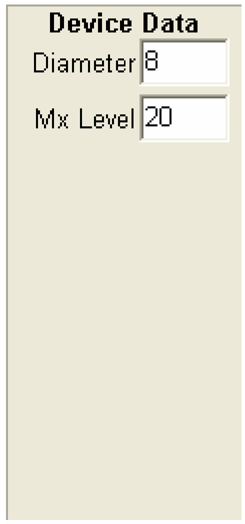
This device represents an orifice in the pipeline where flow enters or exits the pipeline based on the orifice relation. The inflow and outflow resistances can be different (inflow refers to flow into the pipeline and outflow is out of the pipeline).

This device also is an integral part of a number of surge control devices including surge tanks, pressure relief valves and surge anticipation valves.

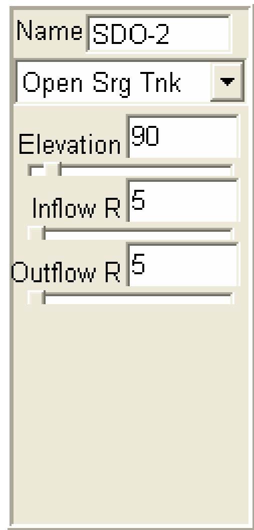
Open Surge Tank (Spilling Tank and One Way (feed) Tank)



One Way Tank



Open/Spilling Tank



Open Surge Tank Data / Units
On/off switch
Diameter – ft. or m
Maximum Level – ft. or m (spilling tank)

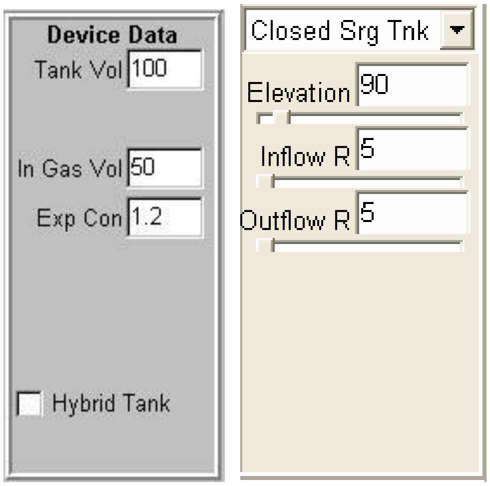
Connection Restrictions: one pipe connected to each side

This device models the following:

open surge tank (non-spilling) This is a tank connecting to the pipeline which is open to the atmosphere. A maximum level (Mx Level) of zero defaults to this NON spilling tank.
spilling surge tank If the tank becomes full where water spills over, the maximum level (Mx Level) should be input and the tank is modeled as a spilling tank.
one-way open surge tank This tank allows flow only from the tank into the pipeline and requires a check valve to be defined at the entrance which prevents flow from entering the tank. This is useful for controlling down surges at a point where the static head is large and a normal open surge tank would not be practical.

Note: If the tank is not a vertical cylinder, calculate the equivalent diameter of a cylinder with the same total volume and same vertical draft. For a horizontal cylinder of diameter D and length L, the equivalent diameter is De = (D*L)^0.5. The expansion constant varies between 1 (isothermal) and 1.4 (adiabatic) and a value between these limits should be used.

Closed Surge Tank



Closed Surge Tank Data / Units
On/off switch
Tank Volume - ft3 or m3
Initial Gas volume (In Gas Vol) – ft3 or m3
Expansion constant

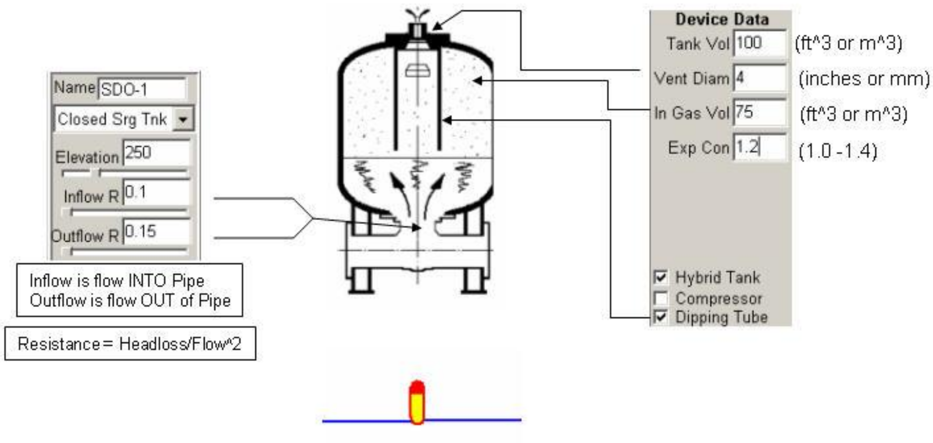
Connection Restrictions: one pipe connected to each side

What is a closed surge tank? A closed surge tank has a volume of gas, usually air, in the space above the liquid. This is the initial gas volume (In Gas Vol). The tank will be initially pressurized to the starting pressure which is determined by the initial steady state analysis. The gas expands and compresses as flow leaves and enters the tank.

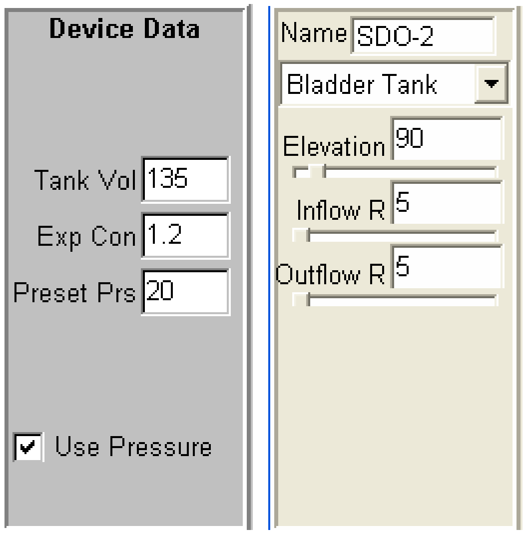
Note: If the tank is not a vertical cylinder, calculate the equivalent diameter of a cylinder with the same total volume and same vertical draft. For a horizontal cylinder of diameter D and length L, the equivalent diameter is De = (D*L)^0.5. The expansion constant varies between 1 (isothermal) and 1.4 (adiabatic) and a value between these limits should be used.

Hybrid Tank –

Hybrid Surge Tank
(closed surge tank)



Bladder Surge Tank

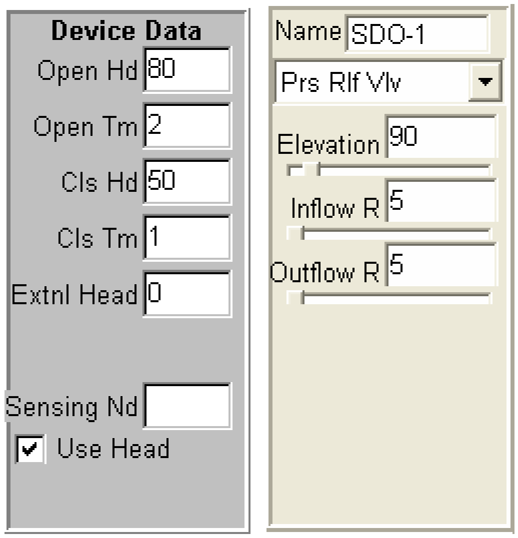


Bladder Surge Tank Data / Units
On/off switch
Tank (bladder) volume (Tank Vol) – ft3 or m3
Expansion constant
Pre-Set pressure (Preset Prs) – psi or kPa or Head – ft. or m

Connection Restrictions: one pipe connected to each side

This is a closed surge tank with a gas-filled bladder. This tank will activate at the set pressure for the bladder and will act as a closed surge tank for pressures above the set pressure. The pre-set pressure (Preset Prs) is the pressure set in the gas in the bladder prior to opening the valve connecting the tank and the pipeline. At any pressure lower than the Pre-set pressure the bladder fills the tank, there is no flow into the pipeline, and the tank has no effect. This is an undesirable condition. The initial gas volume (Vi) is determined by the program using the initial line pressure (Pi) at the location of the bladder tank. This volume is calculated assuming isothermal compression (PiVi = PsVb) For example, for the data displayed above if the initial calculated line pressure at the Bladder Tank is 80 psi, the initial volume is Vi = (20+Pa)*135/(80+Pa) = 49.5 cubic feet. Pa is atmospheric pressure (14.7 psi)

Pressure Relief Valve

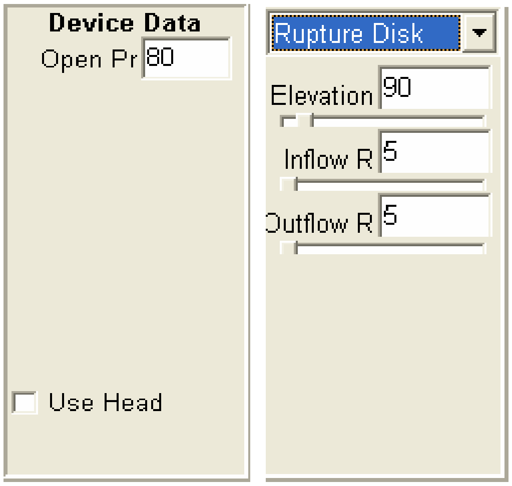


<u>Pressure Relief Valve Data / Units</u>
On/off switch
Opening pressure – psi or kPa
or Head - ft. or m
opening / closing time - sec
closing pressure – psi or kPa
or Head – ft. or m
External Head – exit head for relief valve – ft. or m

Connection Restrictions: one pipe connected to each side

A Pressure Relief Valves begins to open when the pressure at the sensing node exceeds the opening pressure. The opening time is the response time for the valve to go from the start to the fully open position. The valve closure is initiated when the pressure (head) drops below the closing pressure, which is lower than the opening pressure. Because of pressure fluctuation at the sensing node, this valve can remain open for longer periods. If the valve exits to a pressurized region (tank etc.) the External Head should be input. The pressure to activate the valve is generally sensed at the valve but any node can be used for this purpose. If a node is not specified, the sensing is assumed to occur at the valve location.

Rupture Disk

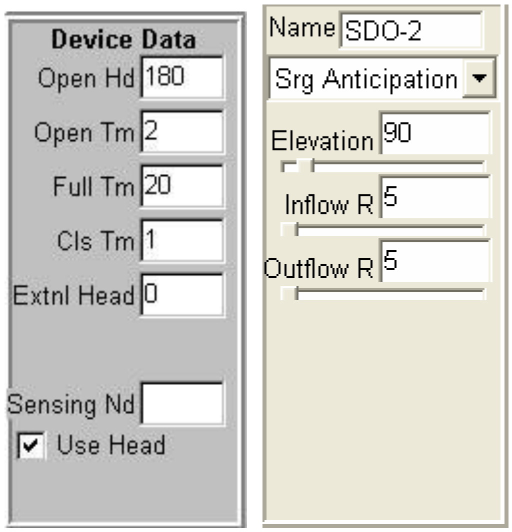


<u>Rupture Disk Data</u>
On/off switch
Opening pressure – psi or kPa
or Head – ft. or m

Connection Restrictions: one pipe connected to each side

This device opens very quickly (ruptures) when the pressure exceeds the opening pressure.

Surge Anticipation Valve



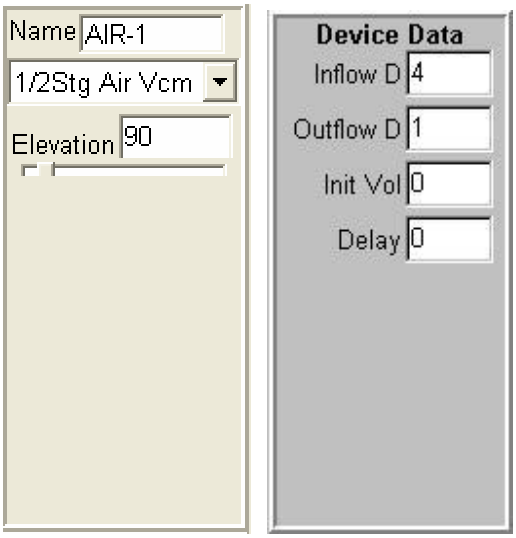
Surge Anticipation Valve Data
On/off switch
Opening pressure – psi or kPa or (head) – ft. or m
opening time – sec.
Fully open time – sec.
Closing time – sec.
External Head – ft. or m

Connection Restrictions: one pipe connected to each side

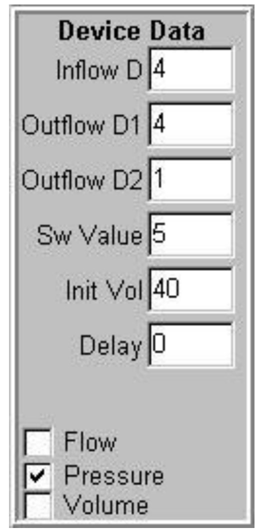
A surge anticipation valve is a device normally located at a pump discharge. This device activates on a down surge when the pressure at the sensing node drops below the opening pressure to provide protection for a subsequent upsurge. Once activated this device works on a timed cycle. After opening is initiated the valve completes a cycle where it opens fully in the opening time, remains fully open for the fully open time and closes completely in the closing time. If the valve exits to a pressurized region (tank etc.) the External head should be input. The pressure to activate the valve is generally sensed at the valve but any node can be designated as the sensing node if the opening is to be activated by the pressure at a different location.

May specify and external head if applicable.

Air Vacuum Valve –1, 2 and 3 Stage



1 and 2 Stage



3 Stage

Air/Vacuum Valve Data
On/off switch
Inflow diameter – in or mm
Outflow diameter (D1) – in or mm
Outflow diameter (D2) – in or mm
Switching value -
Initial Air Volume – ft3 or m3
Switch type – flow, pressure or volume

Connection Restrictions: one pipe connected to each side

Single stage (or nominal) air/vacuum valve - This device has a single orifice which takes air in when the pipeline pressure drops below atmospheric and releases air when the pipeline pressure exceeds atmospheric.

Two-stage air/vacuum valve - This device has a second (smaller) air outflow orifice to reduce the rate of air expulsion and the "air slam" which occurs when all the air is expelled and the liquid columns rejoin.

Three-stage air/vacuum valve – This device has a third orifice that is utilized for outflow if the flow pressure or air volume exceeds the corresponding switching value. For this valve the same orifice can be used for inflow and outflow prior to the switch (D1=D2). The transition from the primary outflow orifice (diameter D1) to the second orifice (D2) usually occurs at a specified pressure (psi or kpa) but can be modeled to switch on flow (cfs or cms) or volume (ft^3 or m^3).

For all three types an initial air volume can be designated at the valve. This feature allows the modeling of start-up with air in the pipeline.

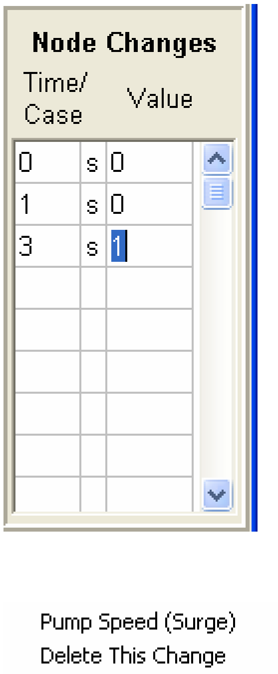
Creating a Transient

What is Variable Input Data? This is data that describes the cause of a transient. A series of pairs of data points are used. These are for time and for the value of a parameter related to the cause of the transient. Periodic changes can also be specified to describe the cause of the transient.

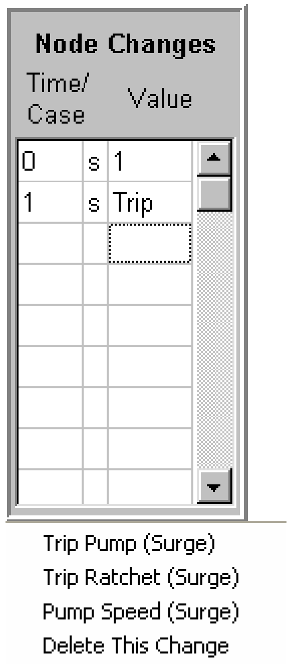
Providing Variable Input Data

- Use change data to specify changed parameter vs. time
- Pumps
    - pump speed ratio (speed/rated speed) vs. time
- Valves
    - valve stem position vs. time
    - effective open area ratio vs. time
- Rapid demand changes
    - demand vs. time
- Rapid pressure changes
    - hydraulic grade vs. time

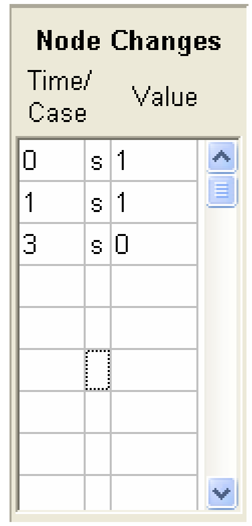
Pumps



1) All Pumps (startup)



2) Pump File (trip)



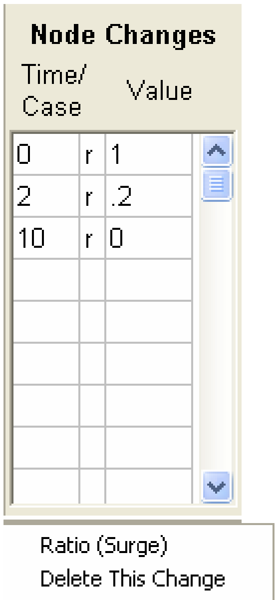
3) All Pumps (speed changes)

Operating speed changes for pumps produce transients. A time dependent change in the speed ratio(s) (operating speed/rated speed) may be defined for all types of pumps as shown above (left). Pumps described by a pump file also can be tripped (lose power) and the resulting rundown calculated based on the pump and motor inertia and dynamic characteristics described in the pump file. A trip can be specified with a ratchet, which will present reverse rotation.

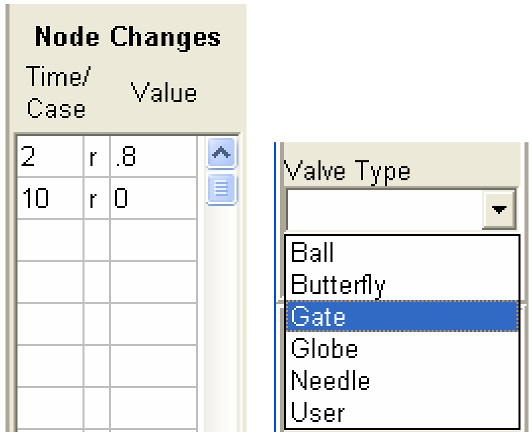
Several scenarios are shown above:

1) The pump is off (speed = 0) for 1 second and then the speed ramps up to full speed (1) over the next 2 seconds.
2) The pump trips (loses power) at 1 second.
3) The pump speed ramps down from 1 to zero in 2 seconds.

Valves



1) User Valve



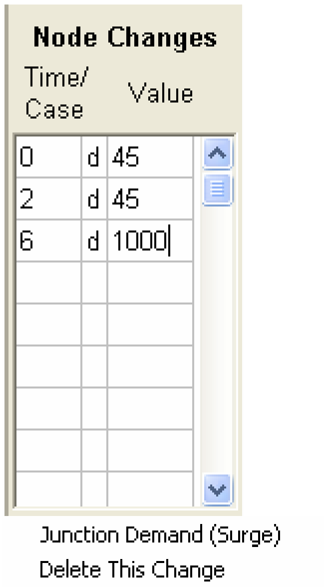
2) Standard Valves

A change in the ratio of the stem position (r) for a valve will produce a transient. A ratio of 1 means the valve is fully open, 0.5 means the stem has turned 50% of fully closed and 0 means the valve is fully closed. For User Valves, any number of pair of time/ratios can be specified to define a closure, opening, or a combination of these two. For other types of valves, only two pairs of values are utilized. The first pair is the time and ratio at the beginning of the valve action and the second pair defines the values at the end. It is assumed that the valve stem moves linearly during the action period.

Two scenarios are shown above:

1) The User Valve is fully open at time – 0 and 20% open at time = 2. The valve is fully closed at time = 10 seconds.
2) The Gate valve is 80% open at 0 seconds, remains steady at 80% until 2 seconds and is fully closed at 10 seconds.

Junction Demand Changes

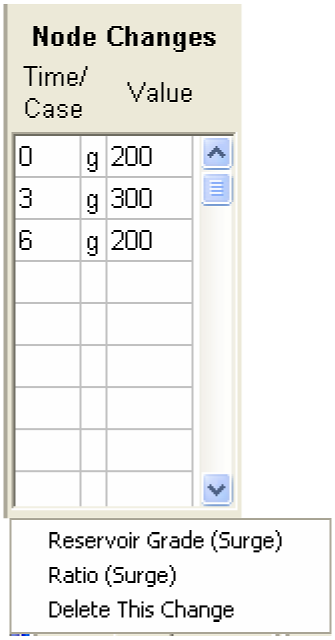


A change in the demand at a junction will produce a transient. This can be used to simulate a hydrant opening or any rapid change in demand. The setup for this as shown above is to provide time/demand pairs to define the demand variation. The demand is in user specified flow units.

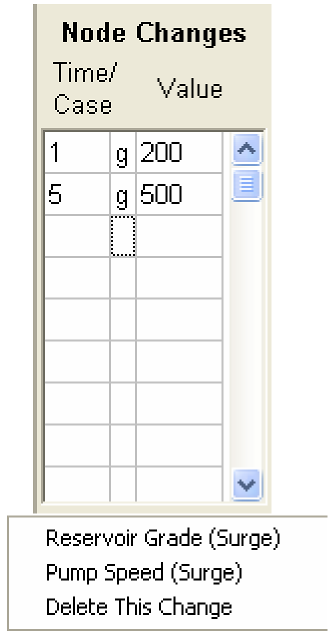
One scenario is shown above

1) A demand of 45 is held for 2 seconds and then ramps up to 1000 over the next 4 seconds.

Reservoir Grade Changes



Reservoir of a Valve



Reservoir of a Pump

A change in the grade (elevation + head) at a reservoir will produce a transient. This feature may be applied only at a valve or a pump, which connects a reservoir (when only one pipe is connected to a valve or a pump, the other side is a reservoir). The setup for this situation is shown above. Time/grade pairs are provided to define the head variation.

Several scenarios are shown above:

1) At a valve the grade ramps from 200 to 300 in the first 3 seconds and then back to 200 in the next 3 seconds.
2) At a pump the grade ramps from 200 at 1 second to 500 at 5 seconds.

Periodic Variations

Control Switches | Constraints | Calibration | Quality | Meters | Loss Element (BFP) | Periodic Input | Ac

Type: 0=Node 1=Connecting Rerservoir

NODE	TYPE	PERIOD	PHASE SHIFT	AMPLITUDE	AVERAGE
AV-2	0	.02	0	.25	.5
Pump-1	0	.02	3.14159	.2	1
AV-2	1	.02	0	50	100
J-3	0	.02	0	10	50

Periodic variations in valve stem position ratios, pump speed ratios, grades at valves (at pumps) and junction demands may be specified. This is done as shown above in the Other Data/Periodic Input screen.

Several scenarios are shown above:

1) A valve operates at 50 cps (period = 0.02 sec) with stem position ratio from 0.25 to 0.75 (average = 0.50).
2) A pump operates at 50 cps with a speed variation from 0.8 to 1.2. This oscillation is 180° (3.14159 radians) out of phase with the other shown.
3) The grade at a valve varies from 50 to 150 feet (m) at 50 cps.
4) The demand at junction J-3 varies from 40 to 60 at 50 cps.

Note: For grade variations at a valve or pump, use Type = 1. For all others, use Type =0.

Node - the node where the parameter change occurs

Type - this only applies to a change in the hydraulic grade at a reservoir attached to a valve or pump. For this application the Type = 1.

Period - the period in seconds (1/frequency).

Phase Shift - the phase shift in radians is required only if there are 2 or more periodic inputs and there is a phase difference between the different inputs.

Amplitude and Average - These are ratios for each of the Variable Inputs

pump: amplitude (average) / rated speed

valve: amplitude (average) / 1.0

demand: amplitude (average) / initial value

grade: amplitude (average) / initial value

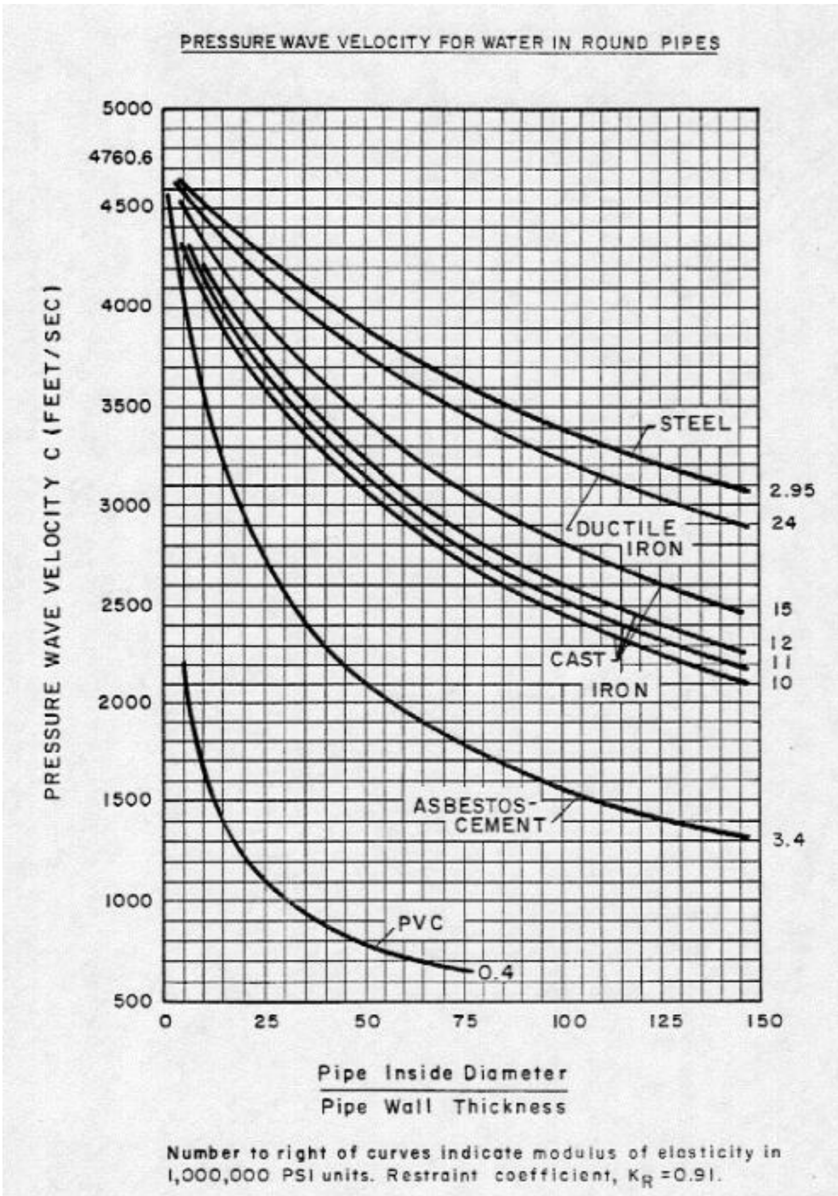
Chapter 8: Pipe2010: Surge Pipe Data (Wave Speed)

What is Wave Speed?
Wave Speed Calculation Tool
Displaying Wave Speed Data.
Defining Wave Speed - Pipe Type
Defining Wave Speed - Pipe Data Box
Wave Speed User Attribute

With the exception of Wave Speed, the Pipe Data is same as required for steady state.

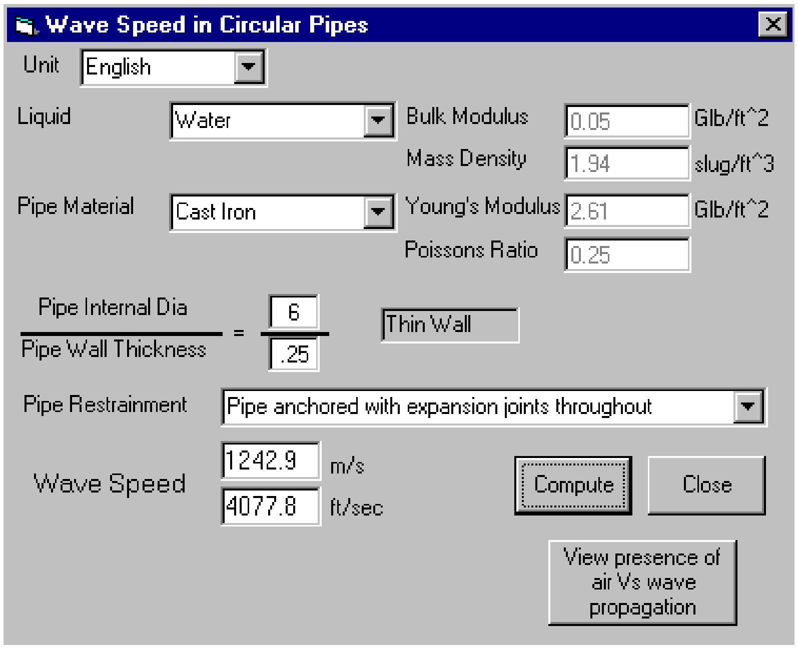
What is Wave Speed?

The wave speed is the speed for pressure wave propagation in a pipe and is an essential data item for transient analysis. The wave speed has a very significant effect on the magnitude of the pressure surge generated by a transient event. The wave speed depends on the properties of the pipe and liquid. The diagram below shows values for some standard pipes.

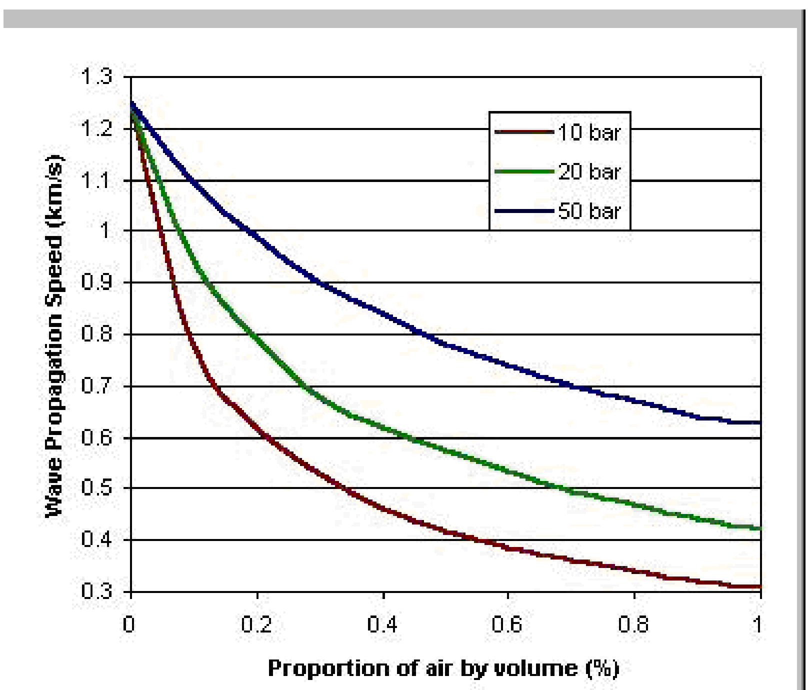


Wave Speed Calculation Tool

Pipe2010: Surge provides a tool for calculating the wave speed. You can select the liquid, pipe material and type of restraint and provide the diameter and wall thickness and calculate the wave speed as shown below.



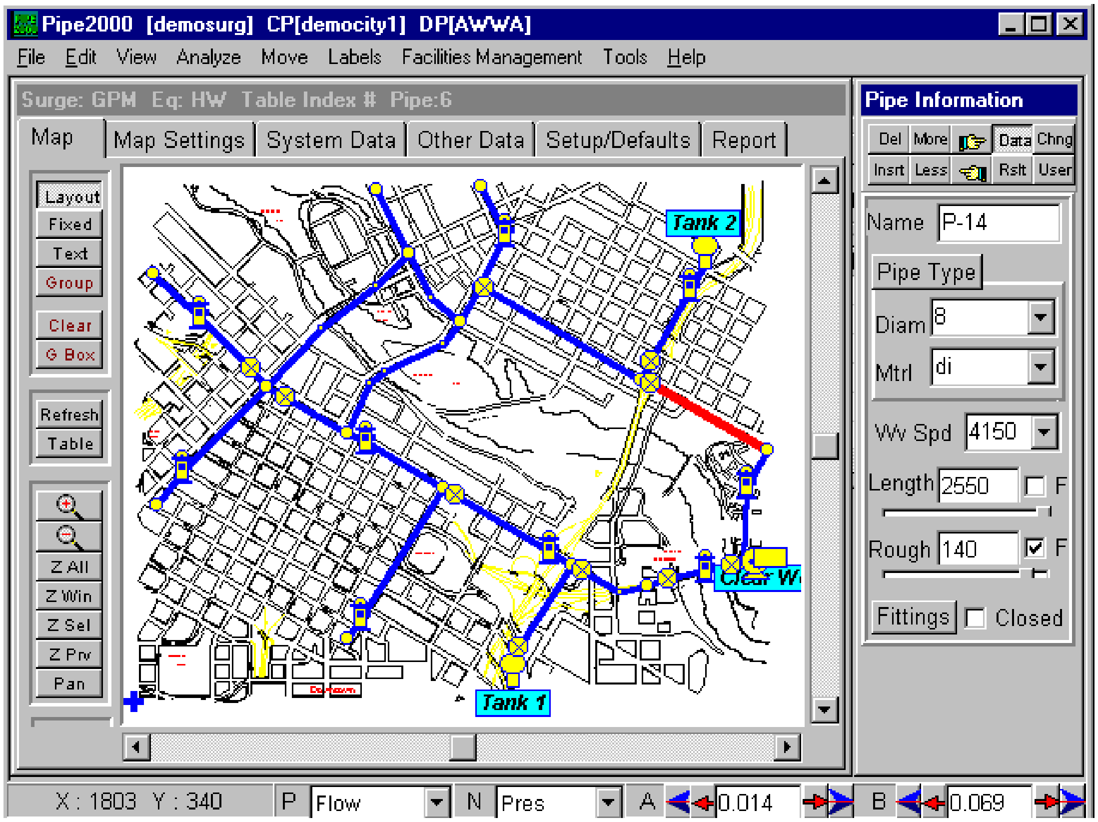
Small amounts of air entrapped in the liquid can greatly affect the wave speed as shown in the diagram below.



The Presence of even small quantities of air can significantly reduce the wave propagation speeds in water in pipes and tunnels. The above graph is for water in a steel pipe having a Diameter/thickness ratio of 20. Ref: Thorley

Displaying Wave Speed Data.

The value of the wave speed is displayed on the main Pipe Data box and the appropriate value can be entered in that location as shown below.



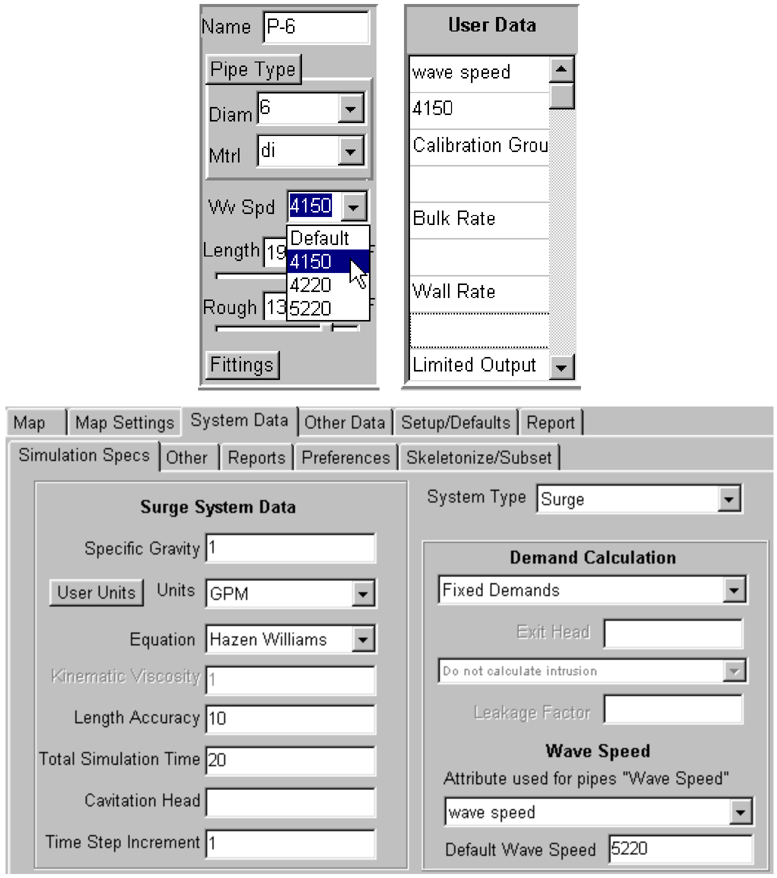
Defining Wave Speed

There are several ways to define the wave speed. They are, in order of precedence:

1. Pipe Data Box / Pipe Attribute

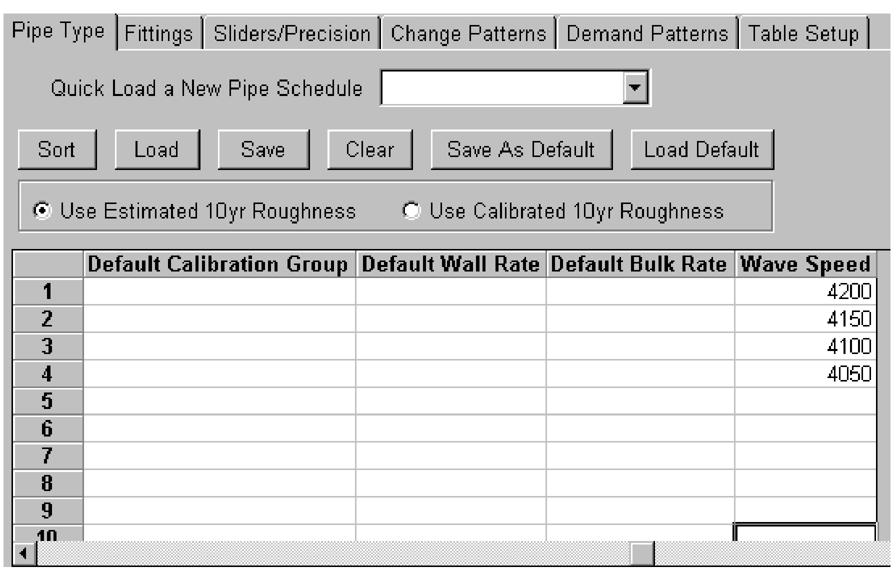
You can define wave speed by entering its value in the location labeled Wv Spd provided on the Pipe Data box as shown. If a value is keyed in here it will be added to the drop down list which can be used to select the appropriate wave speed entry for other pipes. After it is keyed in, that value will also appear in the User Data box under the wave speed user data item. By selecting the word "Default" in the drop down list, any keyed-in wave speed value is erased and the wave speed is defined by one of the other methods described.

For new files, this data item is automatically created and called "wave speed". It is specified under System Data | Simulation Specs in the Attribute used for pipes "Wave Speed" box. However, if you import an older Pipe2010 file, this attribute may not have been defined. If the Attribute used for pipes "Wave Speed" box is blank, you will need to create a User Data item called "wave speed" (see User Data / Adding User Data Items).



2. Pipe Type

This is a very efficient way to provide wave speed data as a system is being created. Under Setups/Defaults | Pipe Type the wave speed may be defined in the Pipe Type Data table as shown below. It will then be part of the data included as pipes are created using the Pipe Type button. Values assigned this way are over-ridden by wave speed data that is keyed directly into the Pipe Data box or in the User Data box under "wave speed".



	Default Calibration Group	Default Wall Rate	Default Bulk Rate	Wave Speed
1				4200
2				4150
3				4100
4				4050
5				
6				
7				
8				
9				
10				

3. Overall Default Wave Speed

Under System Data | Simulation Specs, a box for Default Wave Speed is provided. If there is no data in the Pipe Data box under Wv Spd or in the User Data box under "wave speed", or no wave speed has been assigned using the Pipe Type table then the Default Wave Speed defined here will be used.

4. Pipe2010 internal default

If no wave speed is defined by methods 1, 2 or 3, then Pipe2010 assigns a default wave speed internally.

Chapter 9: Pipe2010: Surge - Reviewing and Presenting Results

All the Pipe2010: KYPipe capabilities for showing results are available for Pipe2010: Surge (Tabulated Reports, Labels, Contours, Profiles, etc.). This section covers some special considerations for reviewing and presenting Surge results.

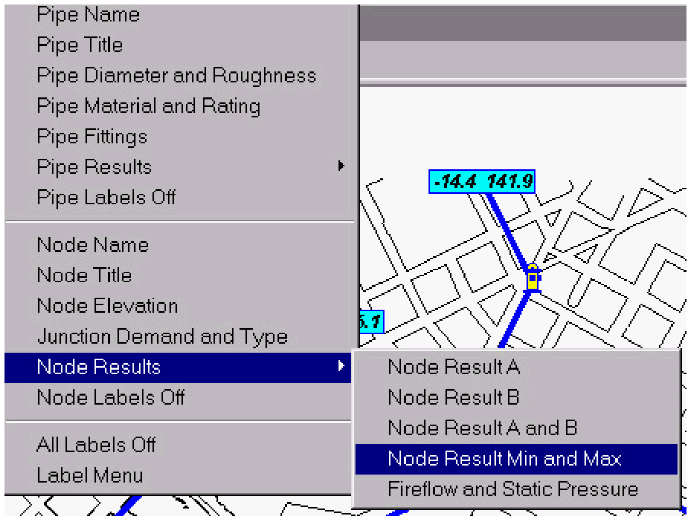
Because a surge analysis produces so many results, it is not practical to store every result for review. To do so may cause memory problems and excessive computational delays. In addition, every result is not normally required to evaluate the overall results. Surge can currently store approximately 6000 sets of results. The following discussion details how large numbers of results are handled in Surge.

For files over 6000 pipes, use Former Method of Analysis.

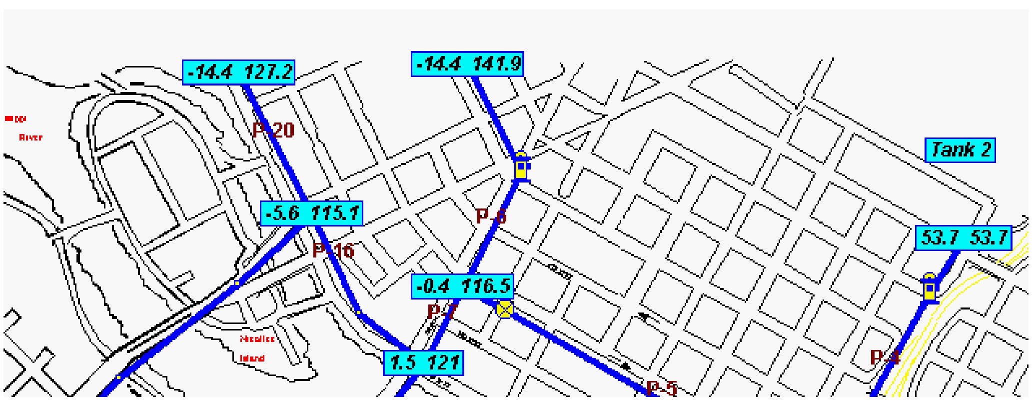
If using the Former Method of Analysis you may reduce the amount of results stored, the number of nodes for which results are stored is limited for larger systems as discussed in a subsequent section (see Selecting Nodes for Limited Output).

Max/Min Heads (Pressures) at Nodes

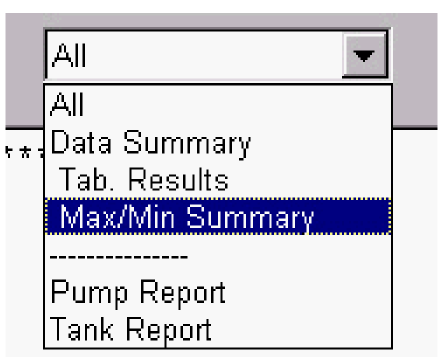
This is very useful information which is always available following the Surge analysis. You can display the max/min heads (pressure or HGL) at all nodes by selecting this label using the label button as shown below.



This will produce a display such as the one shown below.



Also the tabulated report for Surge always includes a Max/Min Head Summary which can be selected from the Report section selector as shown.



This produces the report shown below.

SUMMARY OF MAXIMUM AND MINIMUM HEADS:

POSITION NO.	MAXIMUM (ft)	MINIMUM (ft)	Frac. Time reverse grad.
J-1	237.56	42.82	0.000
J-10	275.62	-12.96	0.049
J-11	324.49	-33.21	0.130
J-12	225.90	38.87	0.000
J-13	293.53	-33.21	0.114
J-14	268.22	6.54	0.000
J-15	249.73	-13.95	0.034
J-16	227.24	10.17	0.000
J-2	379.86	-33.21	0.123
J-3	242.83	-8.15	0.051
J-4	279.31	3.41	0.000
J-5	328.29	-33.21	0.073
J-6	271.72	-1.03	0.006
J-7	265.45	-13.03	0.040
J-8	243.66	29.83	0.000
J-9	287.28	-33.21	0.098

If negative pressures occur this report also presents the fraction of the simulation time at each node at which pressures are negative (and reverse leaks (intrusion) are possible).

Graphs and Tables

For Surge, analysis graphs and tables of pressures and flows at selected nodes are results which are commonly reviewed and presented. For each node up to six sets of results are generated and of these the following are available for review if the node is selected for output. Because of the excessive amount of results generated for even moderately sized systems, it is desirable to designate the nodes to be selected for output (***up to 30 nodes can be selected - see Selecting Nodes for Limited Output). If you do not designate node, the first 30 or so (in the node table) will be used. For small systems all nodes will be automatically selected. For a selected node the following values are displayed.

junction nodes

pressure (or head or HGL) (1)

(or) demand flow (net) in first connecting pipe (1)

component node (pump, valve etc.)

pressure on each side (2)

(or) flow on each side (2)

SDO node

pressure internal and external (2)

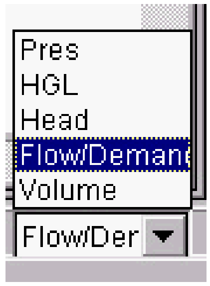
(or) flow side1, side2 and external (3)

(or) volume (air valves and air vessels) (1)

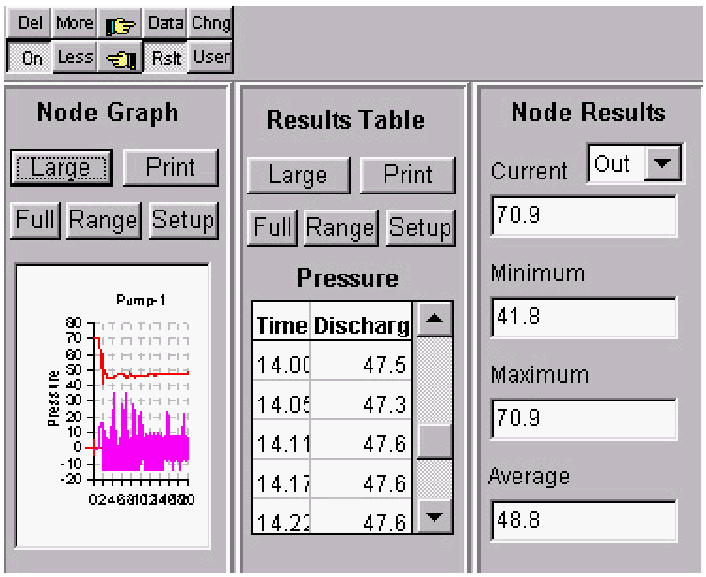
You can generate graphs and plots for one node at a time (layout or fixed mode) or for groups of nodes (group mode).

Surge Node Results

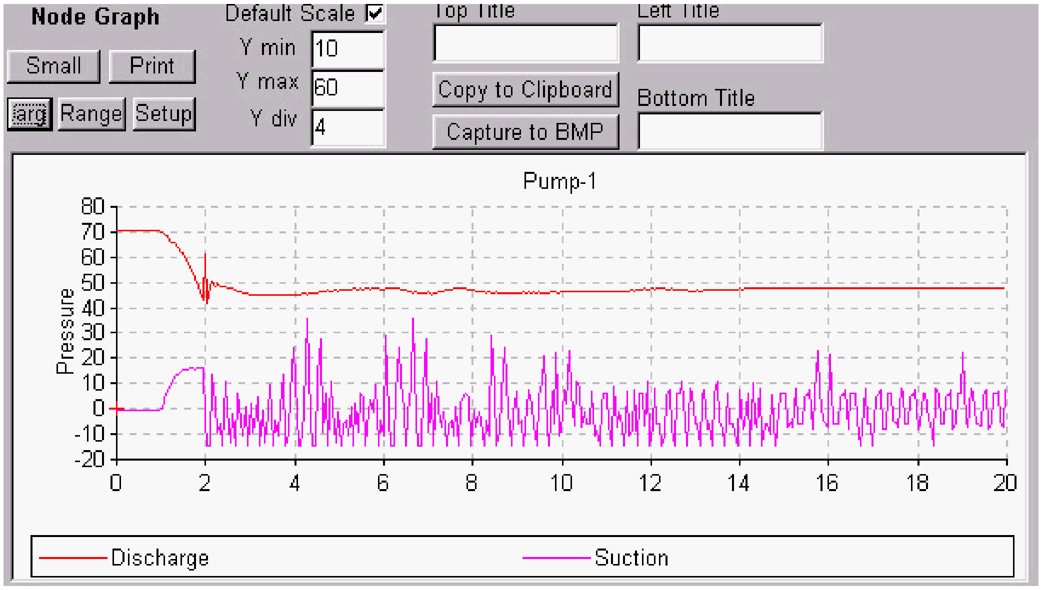
To see the results for a particular node, turn on the Results (Rslt) button and select that node and select the parameter to display from the Node Results popup list as shown below.



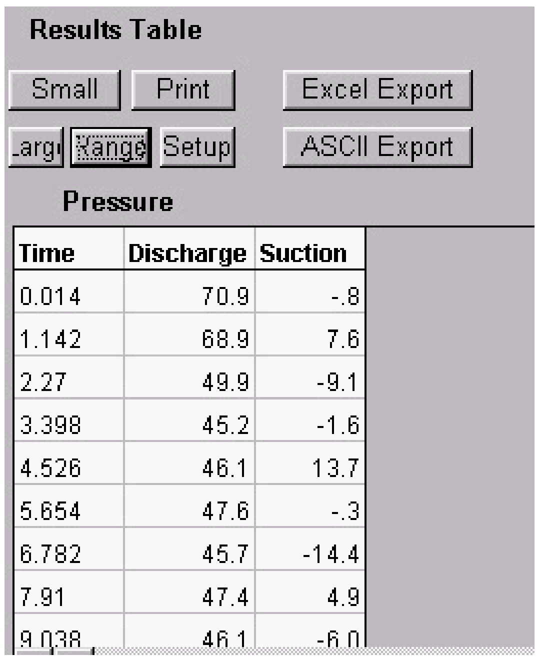
This will produce the Node Information Results boxes shown below which will include a graph or table which will have 1, 2, or 3 sets of results displayed as noted above. The picture below is for pump pressures which has 2 sets of results (2 sides)



The graphs can be expanded as shown below and printed.

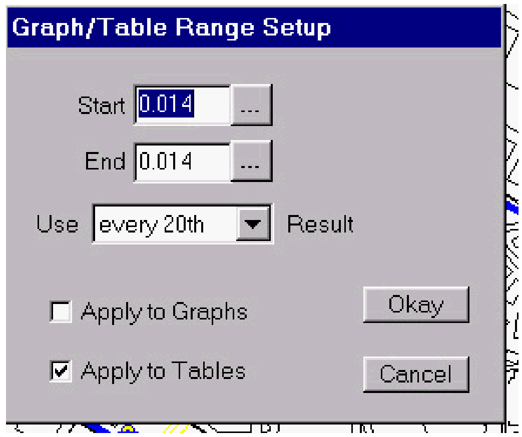


The tables can also be expanded as shown below



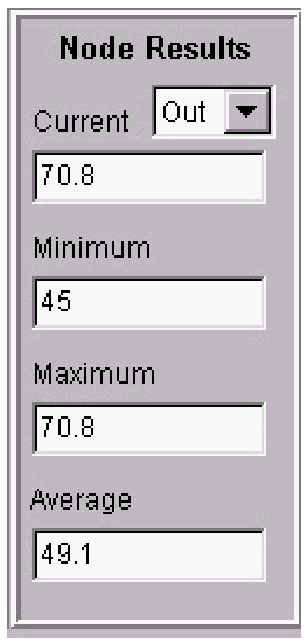
Time	Discharge	Suction
0.014	70.9	-.8
1.142	68.9	7.6
2.27	49.9	-9.1
3.398	45.2	-1.6
4.526	46.1	13.7
5.654	47.6	-.3
6.782	45.7	-14.4
7.91	47.4	4.9
9.038	46.1	-6.0

For both graphs and tables, the range and frequency of values displayed can be controlled by clicking on the Range button and providing data in the locations indicated as shown below. This is particularly useful for tables since the small time increments used in the Surge analysis will result in very long tables.



Group Node Results

Graphs and Tables showing multiple Node Results can be produced in Group Mode. Just select the multiple nodes desired in Group Mode and a graph or table showing results for the selected nodes will be generated. Note that a maximum of three graphs can be displayed in this manner but the tables can include all the selections. For components and SDO results, only one pressure or flow per node will be displayed for Group displays and this selection is controlled by the In/Out button which appears in the Node Results box as shown below.



Other Considerations

Tabulated Reports

Surge performs both a steady state (to determine initial conditions) and a transient analysis when a surge analysis is selected. A tabulated report is generated for each analysis and can be accessed using the Report button and the Load/Swap button to go between the steady state and surge output reports. If the transient analysis is selected, the additional results (labels, graphs, tables, profiles, etc.) are for the transient results. If you wish to display steady state results you should select the Analysis/Analyze System choice: Steady State and perform the steady state analysis.

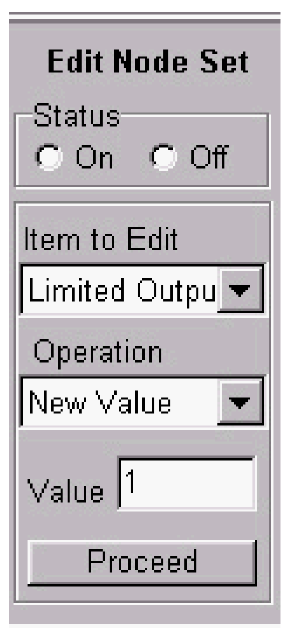
Profiles

The Pipe2010 profiles which show the Max/Min head envelope are a particularly useful tool for reviewing results. This envelope can be generated for any path within the piping system. The maximum/minimum envelope is developed using all results at all nodes. However, the profile for a selected time will use only results for selected nodes which are included in the profile.

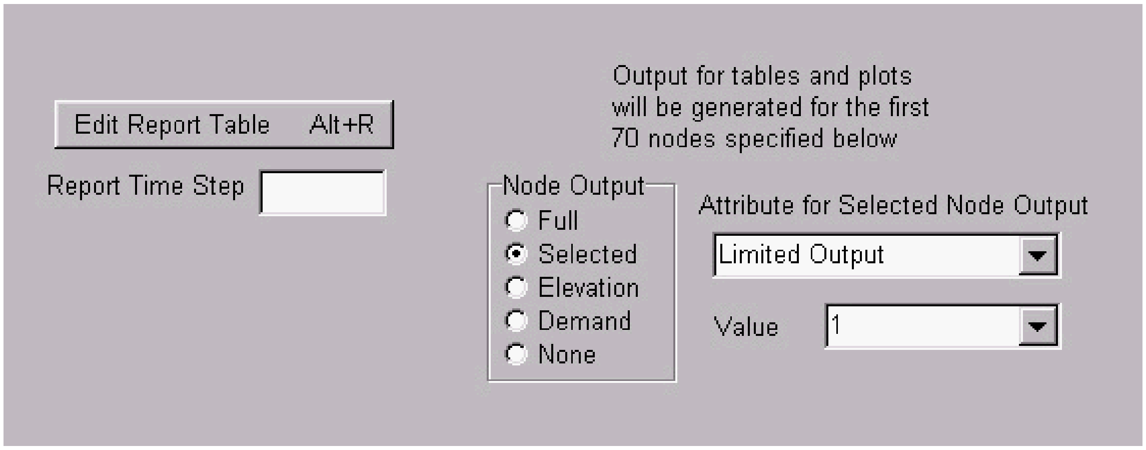
Selecting Nodes for Limited Output

For system with ***25 -30 or more pipes, selecting the nodes for which results are available is an important step to assure that you have access to the results required to evaluate the transient. The normal procedure for doing this is as follows.

1. Select Group Mode
2. Select up to ***25-30 nodes for detailed results
3. Turn on the Node Information Data box and select Limited Output as shown below. Then provide a New Value (usually 1) for the Limited Output attribute for the group of nodes. Make sure you have first set this attribute to zero or a different value for all the other nodes.



Finally under System Data/Reports you need to designate Node Output | Selected, the attribute (Limited Output) and value (1) used to designate the selected nodes as shown below.



Of course, you can use this procedure to select a number of groups with different Limited Output values (say 1, 2, and 3) and designate the desired value to use for a particular run.

Chapter 10: Pipe2010: Surge QuickStart Example

Step 1 - Initial Preparation
Step 2 - System Layout
Step 3 - Provide pipe and node data
Step 4 - Save data file
Step 5 - Analyze System and Review Results

See the Demonstration Examples videos on the Pipe2010 CD #2.

This will guide you through the complete layout development, data entry and hydraulic analysis of a simple pipe network. We recommend that you run Surge in as high a resolution as your monitor can display such that it can be comfortably read. We recommend the following Windows 95/NT settings:

Monitor Size	Display Setting
14" or 15"	1024 x 768
17"	1280 x 1024
21"	1600 x 1280

Step 1 - Initial Preparation

Initial steps include file selection, system data selections, background preparation and pipe type preparation.

a. file selection
You can access an existing data file or, as for this demonstration, create a new one. Click on File (top menu box) and select New.

b. system data selection
The New File setup screen appears. Select Surge - CFS and make sure Wave Speed is checked in the features list. Click OK and go to System Data **|** Simulation Specs. As a minimum you need to specify the head loss equation to use (Hazen-Williams), and total simulation time (10 seconds).

Click on MAP to return to the Surge map.

c. background preparation
You can import a background map, utilize grid lines or choose not to use a background. For this demonstration we will turn on a grid and use it to guide our layout letting Pipe2010: Surge calculate pipe lengths.

Click on Map Settings | Grids - The default grid settings of 1000 (major) and 100 (minor) are good for our demonstration so we will use them.

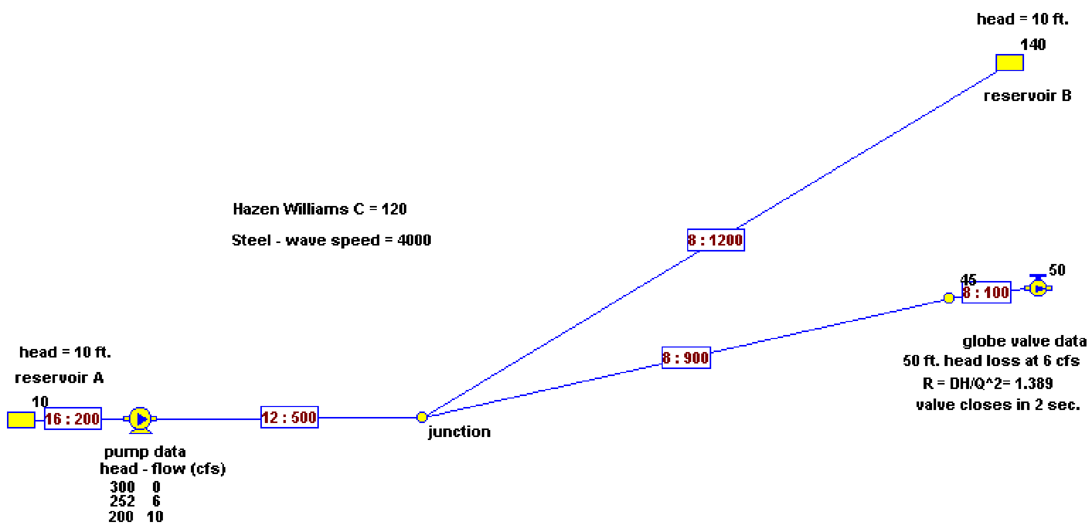
Click on Major Grid and Minor Grid check boxes. This will display background grid lines.

Click on Map to return to the Surge map.

d. pipe type list
Click on Setup/Defaults and Pipe Type and make sure that 16, 12, and 8 inch steel pipe, HW roughness coefficient = 120 and a wave speed of 4000 ft/s are included in the Pipe Schedule you select. This will simplify your data entry requiring only a single selection to define all the pipe data.

Step 2 - System Layout

The map area which appears on the screen will show a region approximately 1000 x 1000 feet with the 100 foot grid lines displayed. This area will be appropriate for the demonstration. A larger or smaller region can be displayed by clicking on the zoom in (+) or zoom out (-) button on the left side.



The system we wish to lay out is shown above. The node elevations are noted. The diameter and roughness are noted in a box for each pipe. The development of the pipe system model is accomplished in four steps. You should be in Layout mode (button - top left) to proceed.

a. layout pipes and nodes

The entire piping system can be laid out using the mouse and a right click (RC) to add pipes and nodes and a left click (LC) to select a node. The following operations will produce the system layout:

1) RC on gridline intersection near the left edge to make first node
2) move mouse 200 feet (2 blocks) to right and RC
3) move mouse 500 feet to right and RC
4) move mouse 1200 feet right and up and RC (note the distance can be observed at the bottom of the screen)
5) select node at 'junction' and move 900 feet right and up and RC (this node is provided to add a surge tank, if required).
6) move mouse an additional 100 feet and RC

Now all the pipes and nodes are laid out. Note all nodes are either junction or intermediate nodes and Pipe2010: Surge has assigned pipe and node names.

b. change node types

Select any nodes which are different than shown and change to the correct node type. To do this LC to select the node and click on drop down node list (Node Information Window - below Name) and select desired type from list.

1) Select node at Reservoir A (LC) and change node type to Reservoir
2) Select node at Reservoir B and change node type to Reservoir
3) Select node at valve and change node type to active valve.
4) Select node at pump location and change node type to pump.

The system should now look as shown below.

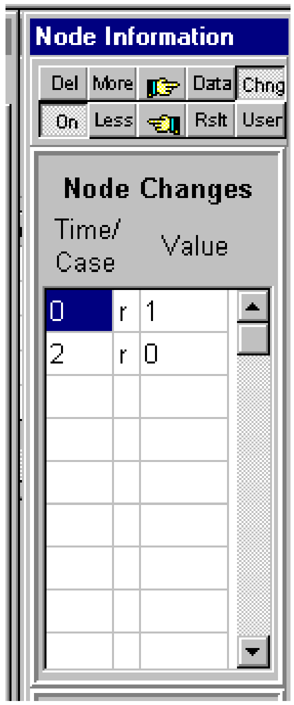
Step 3 - Provide pipe and node data

a) Select each pipe and node and provide data

1) Select each pipe and click Pipe Type (Pipe Information Window) and select choice from drop down list. Select Steel - 16 for pipe from Reservoir A and the appropriate selections for the rest of the pipes. Note that default roughness and wave speed values are provided.
2) Select each Reservoir and provide values shown for Grade (HGL) and Elevation
3) Select each junction and intermediate node and provide Elevation
4) Select the active valve and provide type (globe), elevation, and wide open (WO) resistance.
5) Select the pump and check Table for the pump type and enter the head flow data in the table for ID =1.

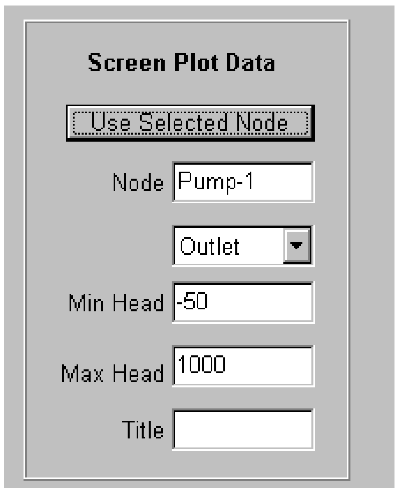
b) Provide Variable Input Data (Changes)

1) Select the Active Valve and open just the Change box (click CHNG in the Node Information window) and provide the data shown below to specify a 2-second closure. Note the r denotes the ratio for the stem position and is entered by clicking on this location and selecting ration from the pop-up list.



d) Select Node for Screen Plot Display

1) We want to observe the pump outlet pressure during the analysis. Select the pump (layout mode). Click on System Data | Other and Use Selected Node on the menu which appears (shown below). Select Outlet and provide the limits for the plot as shown.



Step 4 - Save data file

Provide a name and save your data file. Click on File (Main Menu) and Save As and provide a file name in the popup menu. Such as QSSurge (for Quick Start Surge).

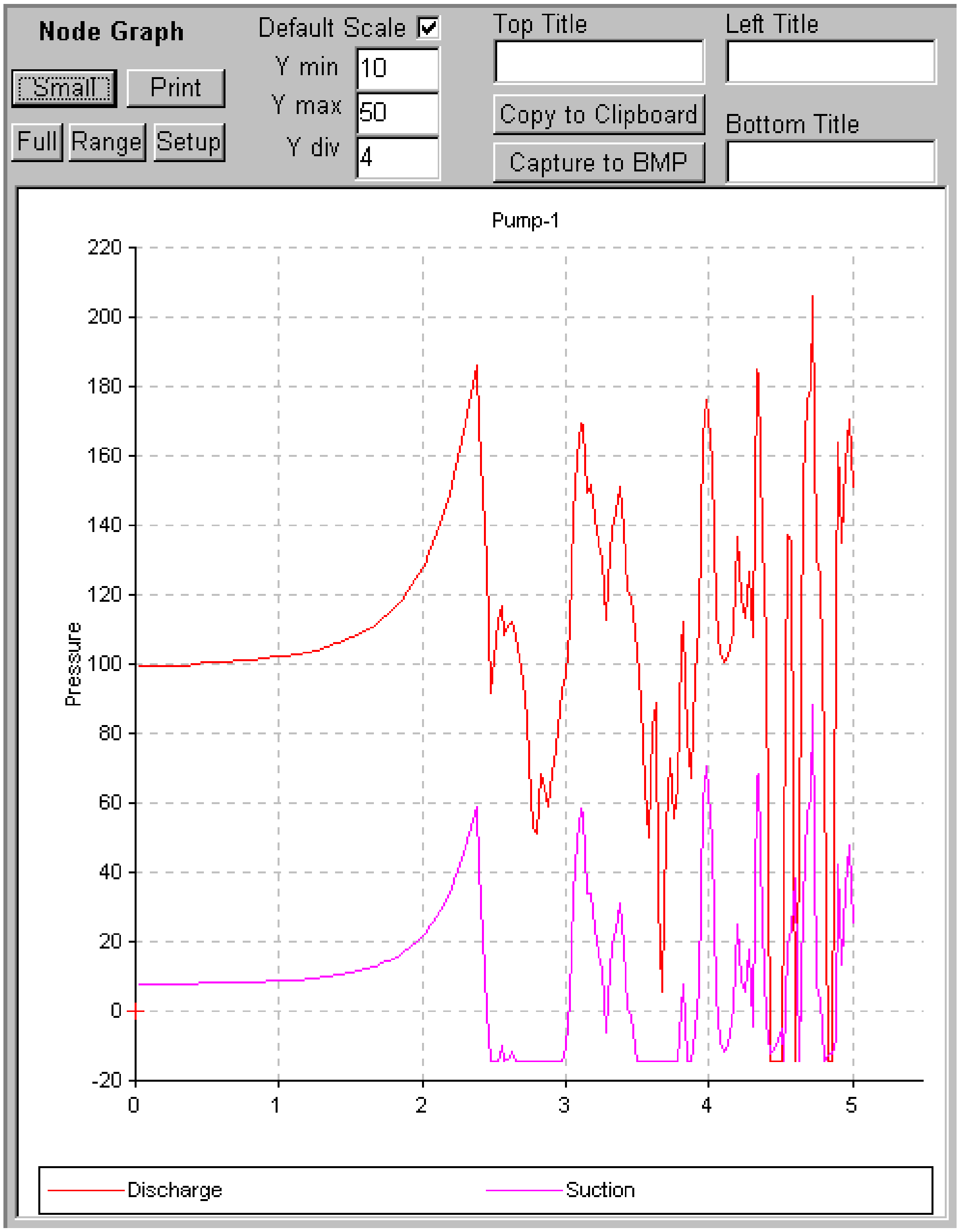
Step 5 - Analyze System and Review Results

a. check data and run analysis

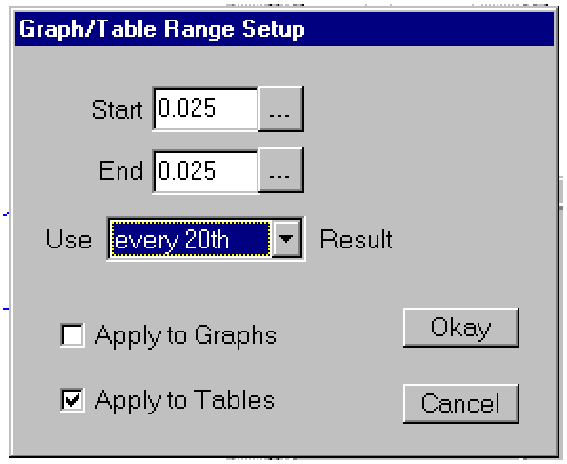
1) Click Analyze (Main menu) and select Error Check. If errors are flagged correct these. If the message "No Errors" appears proceed.
2) Click Analyze (Main Menu) and select Surge and click Analyze on the popup menu to accept the defaults (Analyze with Surge, Use Current Year, and Load Every Result). A plot of the pump discharge pressure will appear. Click OK when finished to load results for review.

b. review results
The results can be reviewed on the schematic using Results Labels, by looking at the tabulated output or by generating customized tables and plots.

1) Click on Report (Main tabs). Any warnings or error messages will appear. Click Load/Swap to load the tabulated results for the transient analysis. You can select a section or 'All' and scroll through the tabulated summary of data and results. Click on Load/Swap to go between the results for the initial steady state results (KYPIPE) and transient results (Surge). Note the table of Max/Min heads at the end of the Surge report provides a very useful overview.
2) Click on Labels (Main menu) and select Node Results **|** Node Result A to show the results depicted in the Results Selection bar on the bottom right of the screen. You can click on the N selector (Pressure for example) to change the node results parameter to head or HGL.
3) Plots and Tables - You can produce plots for heads at all nodes and flows at nodes other than junctions. Select the Pump (layout mode) and under Node Information turn on just the RSLT button (Results). There will be a small display of a pressure plot and table. Click on Full and you will see the transient pump suction and discharge pressure as shown below.



Click on Small to display map. For the Results Table showing every result will result in many values (at each 0.025 seconds). To limit this click on Range and select Every 20th Result and Apply to Tables as shown below.



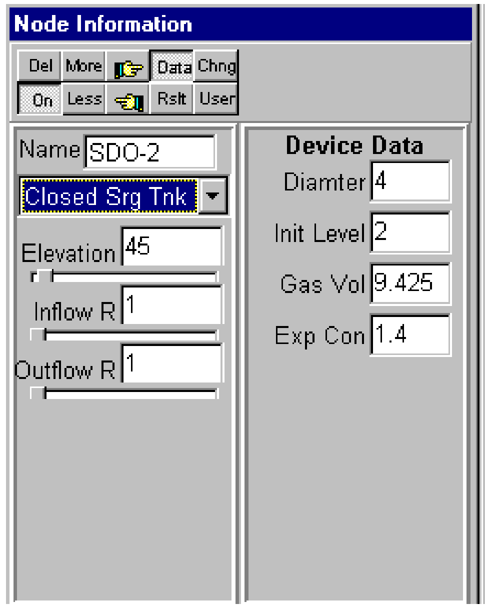
Click on Okay and then Full on the Results Table to show this table

Pressure		
Time	Discharge	Suction
0.025	99.7	7.6
0.525	100.4	8.1
1.025	102.3	8.8
1.525	108.0	11.4
2.025	130.2	23.0
2.525	112.9	-14.4
3.025	110.6	3.3
3.525	89.0	-14.4
4.025	158.1	48.7
4.525	40.3	-14.4
5	150.7	24.8
Avg	108.9	10.8
Min	-14.4	-14.4
Max	205.9	88.2

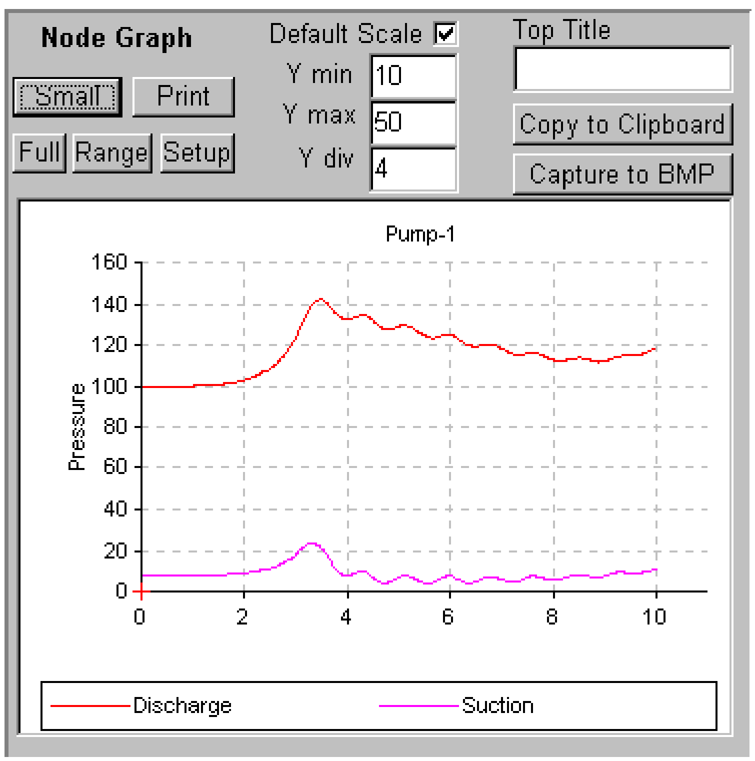
In Group Mode you can select groups of nodes to prepare customized plots and table.

Step 6 - Add a Closed Surge Tank

Add a closed surge tank at the node downstream from the pump. This tank is a 5 foot high cylinder initially with two feet of water. Assume a gas constant = 1.4. Click on the node and select Closed Srg Tnk from the drop down list. Then provide the required data as shown below. The resistance of the connection to the storage tank can be obtained by clicking on Tools | Resistance Calculation and Connection to Tank and specifying a 6" diameter connection. A resistance of 1.0 is calculated and used in the data. The gas Volume is based on a two foot diameter - three foot high cylindrical space.



Click on Analyze and run the Surge Analysis. After completion, click on the Pump and the Rslt button to get a plot of the pump discharge and suction pressures. The results show that the surge tank greatly reduced the surge pressures.



Appendix 1 : Waterhammer Analysis - Essential and Easy (and Efficient) *

By

Dr. Don J. Wood.
University of Kentucky

ABSTRACT

For most piping systems the maximum and minimum operating pressures occur during transient operations. Therefore it is ***essential*** to good design and operation to perform a transient analysis for normal startup and shutdown and for unplanned events such as a pump trip associated with a power outage. This author also claims that waterhammer (transient) analysis is ***easy***. Hydraulic engineers who have studied the traditional approach to transient analysis might dispute this claim but, in fact, carrying out an analysis using the concept of pressure wave action provides an accurate, intuitive and simple method for transient pipe system analysis of simple or complex pipe systems. Not only is this approach simple it is extremely ***efficient*** producing accurate solutions with far fewer calculations making this approach suitable for analyzing large pipe distribution systems..

INTRODUCTION

Waterhammer analysis has traditionally not received the attention it deserves in our engineering curriculums and the consideration it requires for safe and effective design and operation of piping systems. The principal reason for this situation is that transient analysis has been presented to engineer in a manner which is complex and difficult to apply to pipe system hydraulic design when, in fact, this topic can be presented in an intuitive and easily applied manner. In this paper an approach to transient flow analysis based on the action of pressure waves is presented. It is shown that this approach produces accurate solutions using far fewer calculations. In addition the approach provides the engineer an intuitive understanding of pipeline hydraulic transients which will result in improved designs and operations.

1 IMPORTANCE OF HYDRAULIC TRANSIENT ANALYSIS (ESSENTIAL)

Transient analysis of the performance of piping systems is often more important than the analysis of the steady state operating conditions that engineers normally use as the basis for system design. Transient pressures are most important when the rate of flow is changed rapidly, such as resulting from rapid valve closures or pump stoppages. Such disturbances, whether caused by design or accident, may create traveling pressure waves of large magnitudes. These transient pressures are superimposed on the steady state conditions present in the line at the time the transient occurs. The severity of transient pressures must be determined so that the water mains can be properly designed to withstand these additional loads. In fact, pipes are often characterized by their “pressure ratings” that define their mechanical strength and have a significant influence on their cost (Boulos et al, 2004).

Transient regimes in water distribution systems are inevitable and will normally be most severe at pump stations and control valves, high elevation areas, locations with low static pressures, and remote locations that are distanced from overhead storage (Friedman, 2003). All systems will, at some time, be started up, switched off, undergo unexpected flow changes, etc., and will likely experience the effects of human errors, equipment breakdowns, earthquakes, or other risky disturbances. Although transient conditions can result in many situations, the engineer is most concerned with those that might endanger the safety of a plant and its personnel, that have the potential to cause equipment or device damage, that results in operational difficulties or pose a risk to the public health.

Transient events have significant water quality implications. These events can generate high intensities of fluid shear and may cause re-suspension of settled particles as well as biofilm detachment. So-called red water events have often been associated with transient disturbances. Moreover, a low-pressure transient event, say arising from a power failure or pipe break, has the potential to cause the intrusion of contaminated groundwater into a pipe at a leaky joint or break. Depending on the size of the leaks, the volume of intrusion can range from a few gallons to hundreds of gallons (Funk et al, 1999; LeChevallier, 1999 and 2003, Karim et al, 2003). Negative pressures induce backsiphonage of non-potable water from domestic, industrial and institutional piping into the distribution system. Dissolved air (gas) can also be released steel and iron sections with subsequent rust formation and pipe damage. Even some common transient protection strategies, such as relief valves or air chambers, if not properly designed and maintained, may permit pathogens or other contaminants to find a “back door” route into the potable water distribution system.

Engineers must carefully consider all potential dangers for their pipe designs and estimate and eliminate the weak spots. They should then embark upon a detailed transient analysis to make informed decisions on how to best strengthen their systems and ensure safe, reliable operations (Karney and McInnis, 1990; McInnis and Karney, 1995).

CAUSES OF HYDRAULIC TRANSIENTS

Hydraulic transient events are disturbances in the water caused during a change in state, typically from one steady or equilibrium condition to another. The principle components of the disturbances are pressure and flow changes at a point that cause propagation of pressure waves throughout the distribution system. The pressure waves travel with the velocity of sound (acoustic or sonic speed), which depends on the elasticity of the water and that of the pipe walls. As these waves propagate, they create transient pressure and flow conditions. Over time, damping actions and friction reduces the waves until the system stabilizes at a new steady-state. Normally, only extremely slow flow regulation can result in smooth transitions from one steady-state to another without large fluctuations in pressure or flow.

In general, any disturbance in the water generated during a change in mean flow conditions will initiate a sequence of transient pressures (waves) in the water distribution system. Disturbances will normally originate from changes or actions that affect hydraulic devices or boundary conditions. Typical events that require transient considerations include:

- pump startup or shutdown;
- valve opening or closing (variation in cross-sectional flow area);
- changes in boundary pressures (e.g., losing overhead storage tank, adjustments in the water level at reservoirs, pressure changes in tanks, etc.);
- rapid changes in demand conditions (e.g., hydrant flushing);
- changes in transmission conditions (e.g., main break or line freezing); and
- pipe filling or draining – air release from pipes

- check valve or regulator valve action

Potentially, these disturbances can create serious consequences for water utilities if not properly recognized and addressed by proper analysis and appropriate design and operational considerations. Hydraulic systems must be designed to accommodate both normal and abnormal operations and be safeguarded to handle adverse external events such as power failure, pipeline fracture, etc. (Boulos et al, 2004)

ANALYZING TRANSIENTS IN PIPE SYSTENS

Rapidly varying pressure and flow conditions (waterhammer) in pipe systems are characterized by variations, which are both position (x) and time (t) dependent. These conditions are described by the continuity equation

$$\frac{\partial H}{\partial t} = - \frac{c^2}{gA_L}\frac{\partial Q}{\partial x} \qquad (1)$$

and the momentum equation

$$\frac{\partial H}{\partial x} = - \frac{1}{gA_L}\frac{\partial Q}{\partial t} + f(Q) \qquad (2)$$

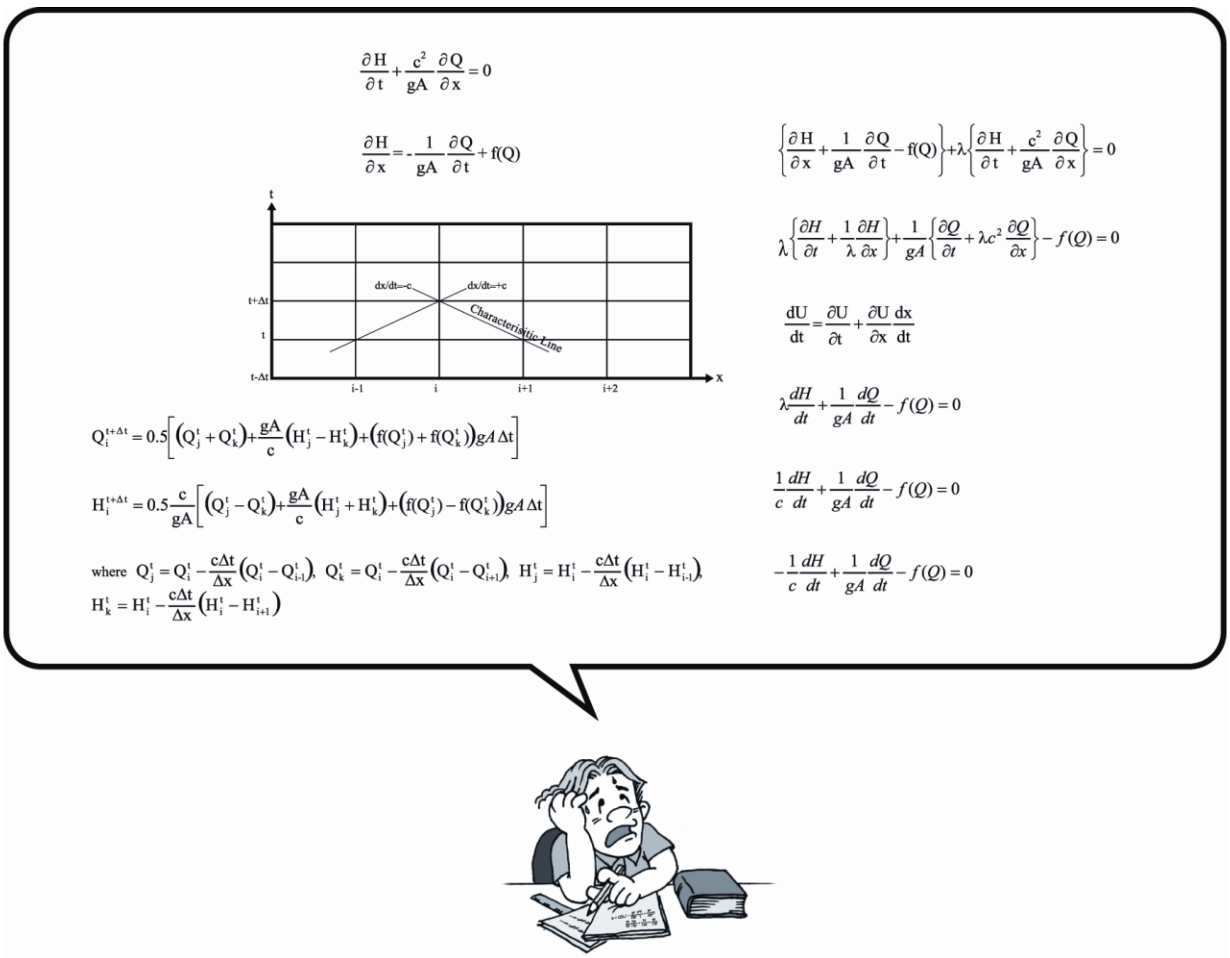
Here H is the pressure head (pressure/density), Q is the volumetric flowrate, c is the sonic wave speed in the pipe, A_L is the cross sectional area, g is the gravitational acceleration, P is the mass density and f (Q) represents a pipe resistance term which is a function of flowrate. Equations 1 and 2 have been simplified by considering only changes along the pipe axis (one dimensional flow) and discarding terms that can be shown to be of minor significance. A transient flow solution is obtained by solving equations 1 and 2 along with the appropriate initial and boundary conditions. However, except for very simple applications that neglect or greatly simplify the boundary conditions and the pipe resistance term, it is not possible to obtain a direct solution.

Graphical and algebraic methods for solving the basic transient flow (waterhammer) equations have been developed (Wylie and Streeter, 1967). These procedures are generally based on a numerical procedure using the method of characteristics (MOC). The MOC is conceptually somewhat complex and requires numerous steps or calculations to solve a typical transient pipe flow problem. As the complexity of the pipe system increases, the number of required calculations increases and for practical applications a computer program is required. Various computer programs have been developed based on the MOC and procedures for handling pipe junctions, pumps, surge tanks and cavitation have been included is most of these programs. The method of characteristics has been described in detain in numerous publications (Wylie and Streeter,1967, Chaudhry 1987, Watters, 1988 and Martin, 2000).

This paper describes an alternate numerical scheme for carrying out transient flow analysis in piping systems. This procedure, initially developed as the "Wave Plan Method (Wood et al., 1964)", yields solutions which are virtually identical to those obtained from exact solutions or those based on the method of characteristics. This approach, however, normally requires orders of magnitude fewer calculations and has the additional advantage of using a conceptually simple physical model as the basis for its development. Because of this, the engineer will gain a better understanding of the mechanics of transient pipe flow.

This method is based on the physically accurate concept that the transient pipe flow results from the generation and propagation of pressure waves that occur as a result of a disturbance in the pipe system (valve closure, pump trip, etc.). A pressure wave, which represents a rapid pressure and associated flow change, travels at sonic velocity for the liquid-pipe medium, and the wave is partially transmitted and reflected at all discontinuities in the pipe system (pipe junctions, pumps, open or closed ends, surge tanks, etc). A pressure wave can also be modified by pipe wall resistance. This description is one that closely represents the actual mechanism of transient pipe flow. In this paper this method is referred to as the wave characteristic method (WCM).

The primary purpose of this paper is illustrated in the pictures which appear below. As depicted in the upper picture transient analysis can be presented is such a manner that only selected engineering gurus will master the techniques and be able to follow this maze of manipulations and carry out these important calculations. The ordinary engineer will often become lost in the maze of equations and procedures. Or, as depicted in the lower picture, the simple, intuitive calculations based on pressure wave action and the basic waterhammer equation will be learned and utilized by all hydraulic engineers resulting in improved pipelines designs and operations which give full consideration to transient operations.



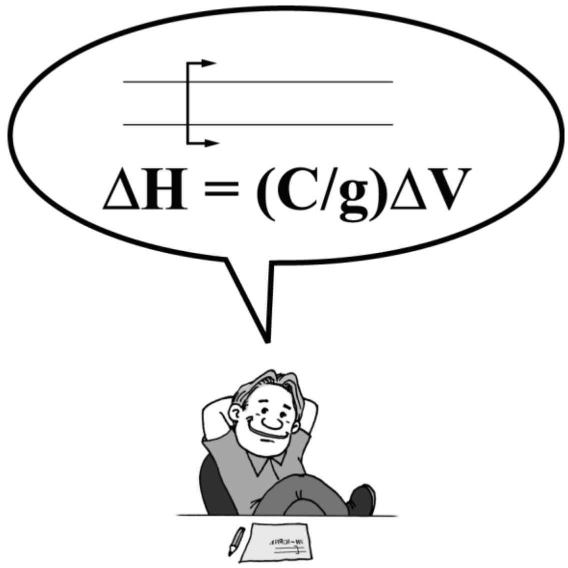


Figure 1 Calculations and Concepts – MOC and WCM Methods

2 TRANSIENT ANALYSIS OF PIPE SYSTEMS USING THE WAVE METHOD (EASY)

Pressure waves are generated at any point in a flow system where a disturbance that results in a change in flowrate is introduced. This can include a valve that is opening or closing, a pump that is started up or shut down, a change in a reservoir pressure, or a change in an inflow or outflow for the system. Pressure and flow conditions at a component are also affected by pressure waves impinging on the component. This approach to transient analysis is illustrated above. (1)A valve closes and a pressure wave is generated. (2)The wave travels toward the 3 pipe junction at sonic speed in the pipe. (3)The wave is transmitted into the two connecting pipes and reflected back into the original pipe producing 3 new pressure waves. (4)Each pressure wave travels at sonic speed toward the opposite end of the pipes and impinge on the elements located there. (5)The pressure waves modify conditions at the reservoir, valve, and pump and new pressure waves are generated and travel back toward the junction.

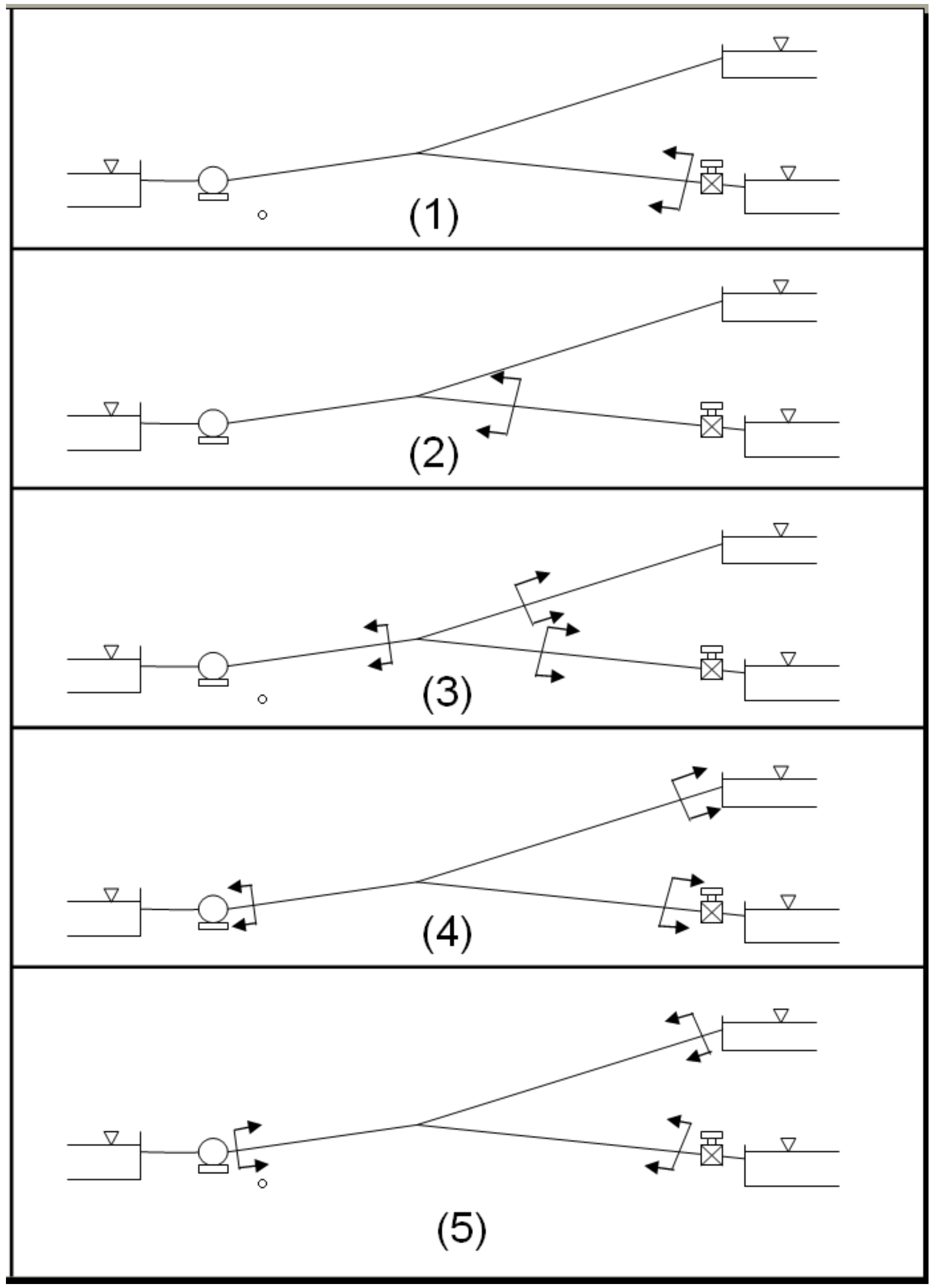


Figure 2 Illustration of the Wave Characteristic Method

This approach to transient analysis requires the calculation of the effects of pressure waves impinging on (1) components (such as valves and pumps), (2) junctions, (3) surge control elements and (4) a calculation the effect of line friction on the magnitude of pressure waves. These pressure wave action calculations required for general applications to pipe systems are illustrated below.

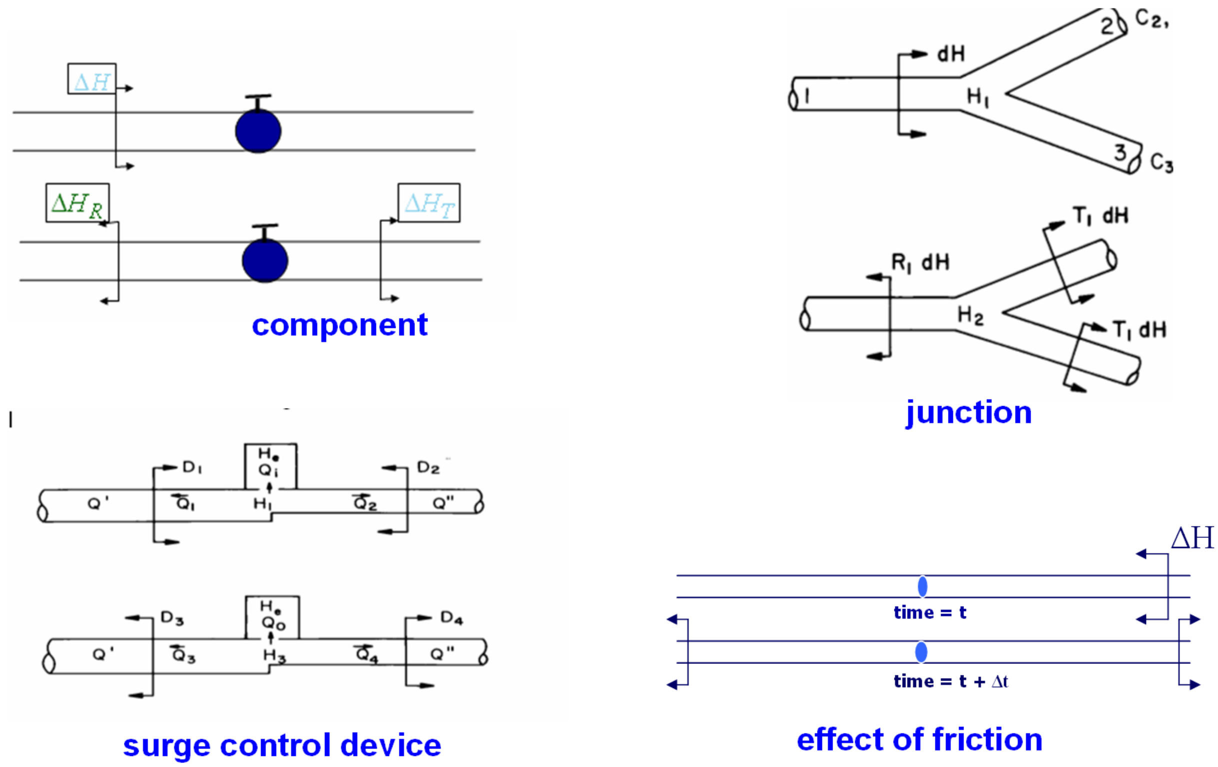


Figure 3 The Analysis of Pressure Wave Action for the WCM

Computer routines developed for wave action at (1) components, (2) junctions, (3) surge control devices and (4) the effect of pipe friction have been utilized to create a general purpose computer model for pipe system transient analysis based on the wave method. The program uses the fact that pressure waves are transmitted between elements at known speeds and are modified by pipeline friction to determine the characteristics of the impinging waves at any time during the simulation. This technique may be applied to complex pipe systems (the Wave Characteristic Method (WCM)) and has been widely used in commercially available software for over 20 years (Wood and Funk, 1996). A textbook presenting this technique in detail will soon be available (Wood, Lingireddy and Boulos, 2004)

3 COMPARING THE WCM AND THE MOC (EFFICIENT)

MOC and WCM Numerical Techniques - The strategy employed by the MOC is to convert the governing partial differential equations to ordinary differential equations and then to a difference form for solution by a numerical method. The equations express the head and flow for small time steps (Δt) at numerous locations along the pipe sections. Calculations during the transient analysis must begin with a known initial steady state and boundary conditions. That is head and flow at time $t = 0$ will be known along with head and/or flows at the boundaries at all time periods. To handle the wave characteristics of the transient flow head and flow values at time $t+\Delta t$ at interior locations are calculated making use of known values of head and flow at the previous time step at adjacent locations using the ordinary differential equations expressed in difference form. Exact solutions of the basic wave equations have been compared to numerical solutions based on the MOC and WCM and have been shown to be identical (Boulos, 1990). However these comparisons were limited to extremely simple systems.

The WCM is based on the concept that the transient pipe flow results from the generation and propagation of pressure waves that occur as a result of a disturbance in the pipe system (valve closure, pump trip, etc.). The wave characteristics are handled using pressure waves, which represents rapid pressure and associated flow changes that travel at sonic velocity for the liquid-pipe medium. A pressure wave is partially transmitted and reflected at all discontinuities in the pipe system (pipe junctions, pumps, open or closed ends, surge tanks, etc). The pressure wave will also be modified by pipe wall resistance. This description is one that closely represents the actual mechanism of transient pipe flow (Thorley, 1991; Boulos et al, 2004).

Both the MOC and WCM obtain solutions at intervals of Δt at all junctions and components. However, the MOC also requires solutions at all interior points for each time step. This requirement basically handles the effects of pipe wall friction and the wave propagation characteristics of the solutions. The WCM handles these effects by using the pressure wave characteristics. The waves propagate through pipes at sonic speed and are modified for the effects of friction by a single calculation for each pipe section.

Required Number of Calculations. Both the MOC and the WCM require many calculations to solve the transient flow problem. These calculations involve updating the pressure and flow at required locations at increments of the time step, Δt. In order to compare the number of calculations required, we define one calculation as the operation required to update the pressure and flow at a single location.

The MOC requires a calculation at all nodes and all interior points at each time step. The WCM requires a calculation at each node and one calculation for each pipe at each time step. The pipe calculations are required to modify the pressure waves in that pipe to account for the effect of pipe wall and fittings friction.

The time step used in the analysis will be determined by the tolerance set for the accuracy of the model pipe lengths or wave speed. A time step must be chosen such that pressure waves traverse each pipe segment in time which is a multiple of the time step. For the comparisons shown the pipe length tolerance was set to 20 ft (6 m). This means that the largest possible time increment was chosen so that the maximum error in the length of pipes in the model would not exceed 20 ft (6 m).

The table below summarizes the calculations requirements for two example systems (Wood et al., 2004). Details for these examples follow. In addition a comparison is made for three additional larger (existing) water distribution systems which have been modeled and analyzed using both approaches but not described herein (Examples 3 - 5).

	No. nodes	No. pipes	Δt (sec)	No inter. Points	Calculations /Δt MOC	WCM	MOC/WCM
Example 1	7	9	0.1	41	48	16	3.0
Example 2	36	40	.0139	680	716	76	9.4
Example 3	589	788	.0056	15,117	15,708	1,377	11.4
Example 4	1,170	1,676	.0067	81,508	82,678	2,846	29.0
Example 5	1,849	2,649	.0056	159,640	161,486	4,495	35.9

Table 1 Comparison of Required Calculations (MOC and WCM)

It should be noted that the number of calculations for the WCM per time step does not change with accuracy. For the MOC the number of calculations per time step is roughly proportional to the accuracy. For the above examples, the calculations/Δt required for the MOC would roughly double if an accuracy of 10 ft (3 m) is required and will be halved if an accuracy of 40 feet (12 m) is called for.

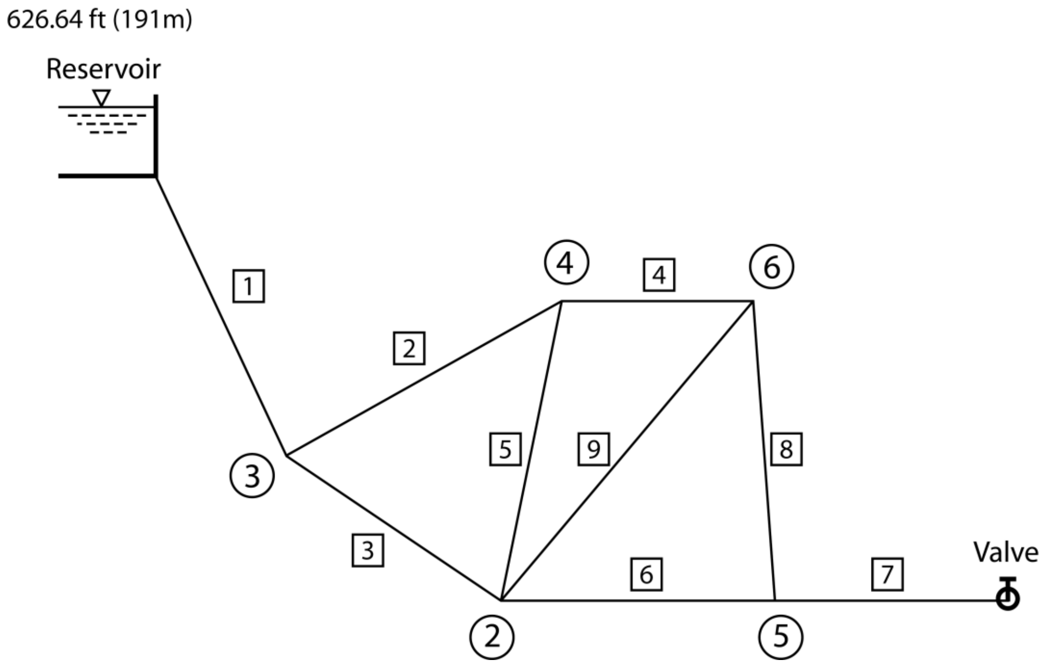
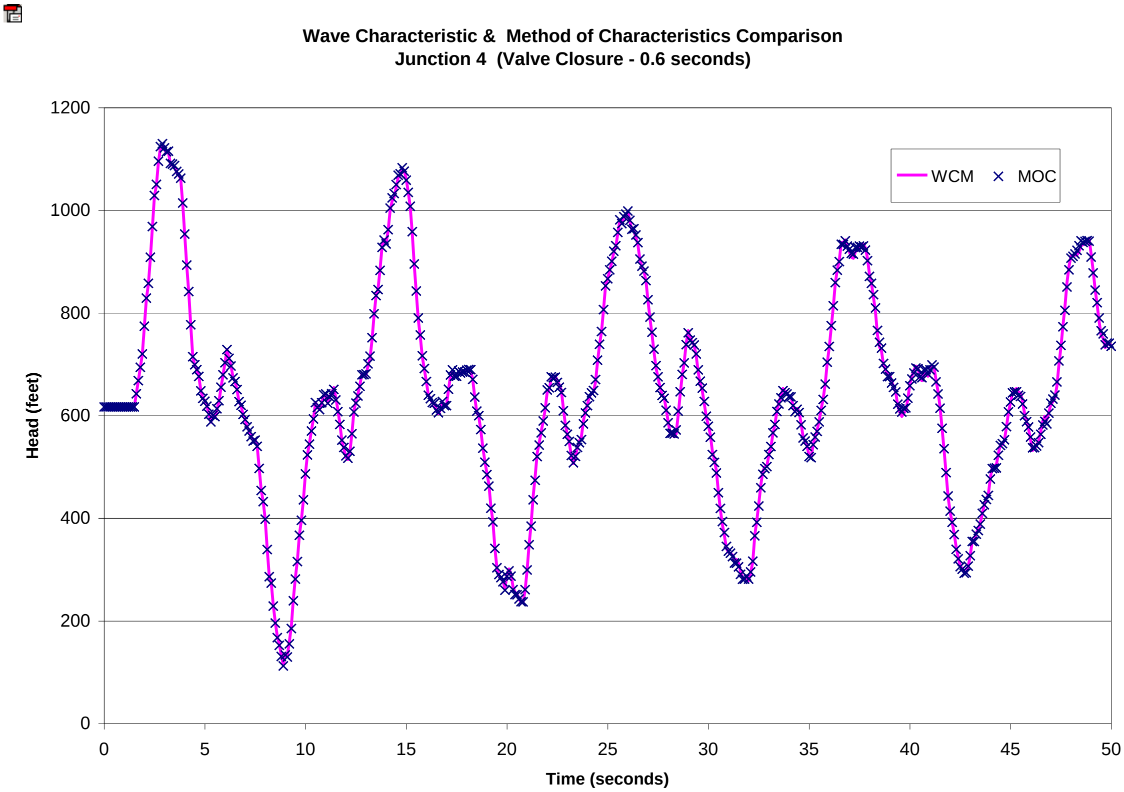


Figure 4 Example 1 Schematic

Pipe Number	Length ft (m)	Diameter in (mm)	Roughness	Minor Loss
1	2,000 (610)	36 (914)	92	0
2	3,000 (914)	30 (762)	107	0
3	2,000 (610)	24 (610)	98	0
4	1,500 (457)	18 (457)	105	0
5	1,800 (549)	18 (457)	100	0
6	2,200 (671)	30 (762)	93	0
7	2,000 (610)	36 (914)	105	0
8	1,500 (457)	24 (610)	105	0
9	1,600 (488)	18 (457)	140	0

TABLE 2 Pipe characteristics for example 1.

Example 1. The first example network was studied earlier by Streeter and Wylie (1967) and is shown in Figure 3. The network comprises 9 pipes, 5 junctions, one reservoir, 3 closed loops, and one valve located at the downstream end of the system. The valve is shut to create the transient. Table 2 summarizes the pertinent pipe system characteristics. The reservoir level is shown in the figure. Figures 4 and 5 compare the transient results obtained using the MOC and WCM solution schemes at the valve and junction 4, respectively. A 20 ft (67 m) length tolerance was used in the analysis which resulted in a required time step of 0.1 seconds. In the figures both solutions are plotted and the two methods produced results that are virtually indistinguishable.



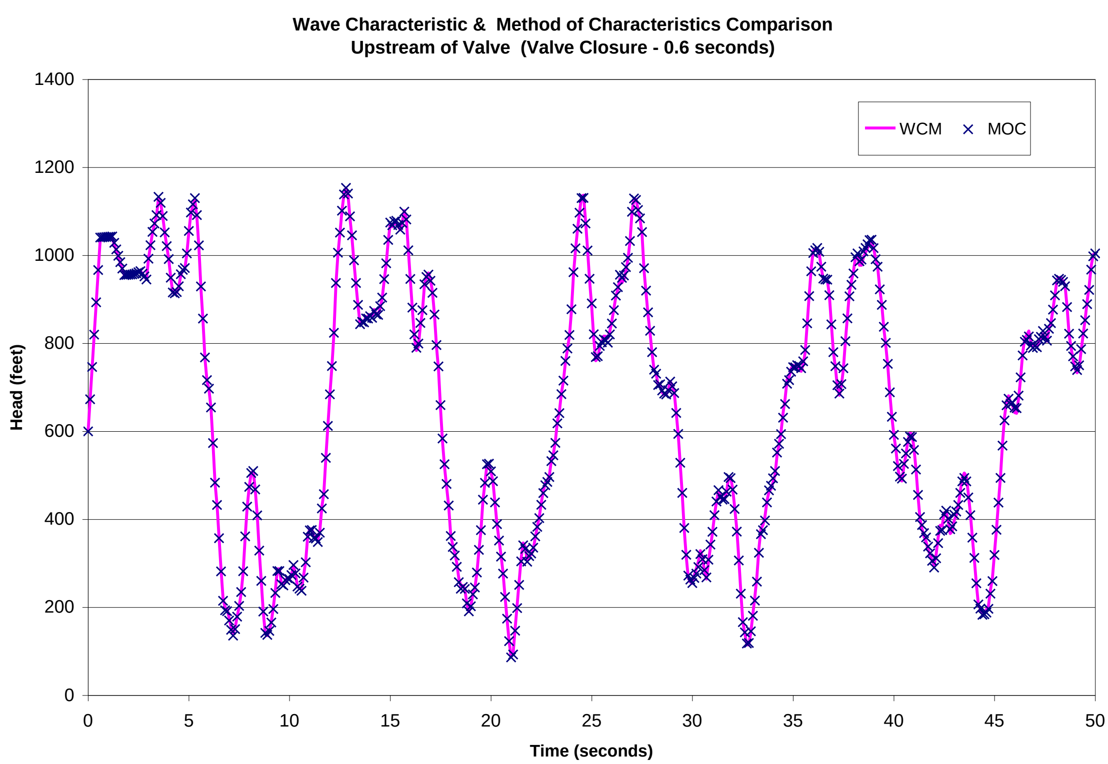


Figure 5 Comparison of Results for Example 1

Example 2. Using a slightly larger more complex system, the methods were applied to the network shown in Figure 6. This represents an actual water system and consists of 40 pipes, 35 junctions, one supply pump and one tank. This example appears in the EPANET (Rossman, 1993) documentation. Table 3 summarizes the pertinent pipe system characteristics. The Pump Station is modeled by designating the inflow at that location. Figures 7 compares the transient results obtained using the MOC and the WCM solution schemes at nodes 1 and 19 respectively, following a pump shutdown simulated by reducing the inflow to zero over a period of 6 seconds. A 20 ft length (6 m) tolerance was employed in the analysis resulting in a required time step of 0.0139 seconds. As can be seen from the figures, both methods yielded virtually identical results.

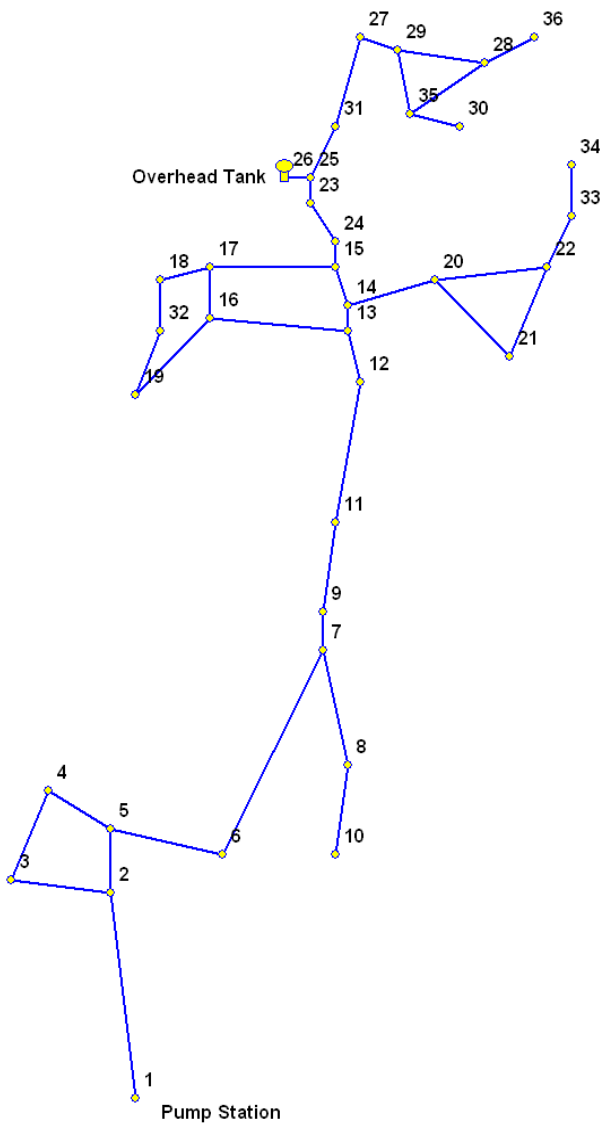
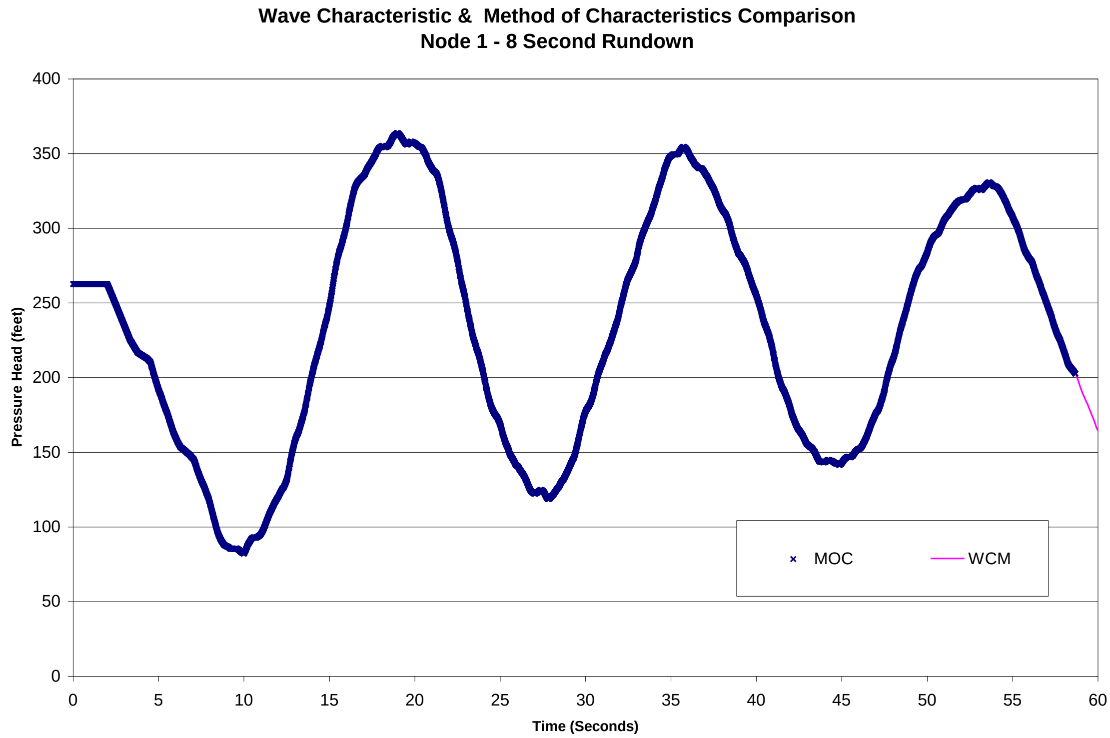


Figure 6 Example 2 Schematic

Pipe ID	Length ft (m)	Diameter in (mm)	Roughness	Node ID	Elevation ft (m)	Demand gpm (L/s)
1	2400 (732)	12 (305)	100	1	50 (15)	-694.4 (-44)
2	800 (244)	12 (305)	100	2	100 (30)	8 (0.5)
3	1300 (396)	8 (203)	100	3	60 (18)	14 (0.9)
4	1200 (366)	8 (203)	100	4	60 (18)	8 (0.5)
5	1000 (305)	12 (305)	100	5	100 (30)	8 (0.5)
6	1200 (366)	12 (305)	100	6	125 (38)	5 (0.3)
7	2700 (823)	12 (305)	100	7	160 (49)	4 (0.3)
8	1200 (366)	12 (305)	140	8	110 (34)	9 (0.6)
9	400 (122)	12 (305)	100	9	180 (55)	14 (0.9)
10	1000 (305)	8 (203)	140	10	130 (40)	5 (0.3)
11	700 (213)	12 (305)	100	11	185 (56)	34.78(2.2)
12	1900 (579)	12 (305)	100	12	210 (64)	16 (1)
13	600 (183)	12 (305)	100	13	210 (64)	2 (0.1)
14	400 (122)	12 (305)	100	14	200 (61)	2 (0.1)
15	300 (91)	12 (305)	100	15	190 (58)	2 (0.1)
16	1500 (457)	8 (203)	100	16	150 (46)	20 (1.3)
17	1500 (457)	8 (203)	100	17	180 (55)	20 (1.3)
18	600 (183)	8 (203)	100	18	100 (30)	20 (1.3)
19	700 (213)	12 (305)	100	19	150 (46)	5 (0.3)
20	350 (107)	12 (305)	100	20	170 (52)	19 (1.2)
21	1400 (427)	8 (203)	100	21	150 (46)	16 (1.0)
22	1100 (335)	12 (305)	100	22	200 (61)	10 (0.6)
23	1300 (396)	8 (203)	100	23	230 (70)	8 (0.5)
24	1300 (396)	8 (203)	100	24	190 (58)	11 (0.7)
25	1300 (396)	8 (203)	100	25	230 (70)	6 (0.4)
26	600 (183)	12 (305)	100	27	130 (40)	8 (0.5)
27	250 (76)	12 (305)	100	28	110 (34)	0 (0)
28	300 (91)	12 (305)	100	29	110 (34)	7 (0.4)
29	200 (61)	12 (305)	100	30	130 (40)	3 (0.2)
30	600 (183)	12 (305)	100	31	190 (58)	17 (1.1)
31	400 (122)	8 (203)	100	32	110 (34)	17 (1.1)
32	400 (122)	8 (203)	100	33	180 (55)	1.5 (0.1)
34	700 (213)	8 (203)	100	34	190 (58)	1.5 (0.1)
35	1000 (305)	8 (203)	100	35	110 (34)	0 (0)
36	400 (122)	8 (203)	100	36	110 (34)	1 (0.1)
37	500(152)	8 (203)	100	26	235 (72)	Tank
38	500 (152)	8 (203)	100			
39	1000 (305)	8 (203)	100			
40	700 (213)	8 (203)	100			
41	300 (91)	8 (203)	100			

TABLE 3 Network characteristics for example 2.



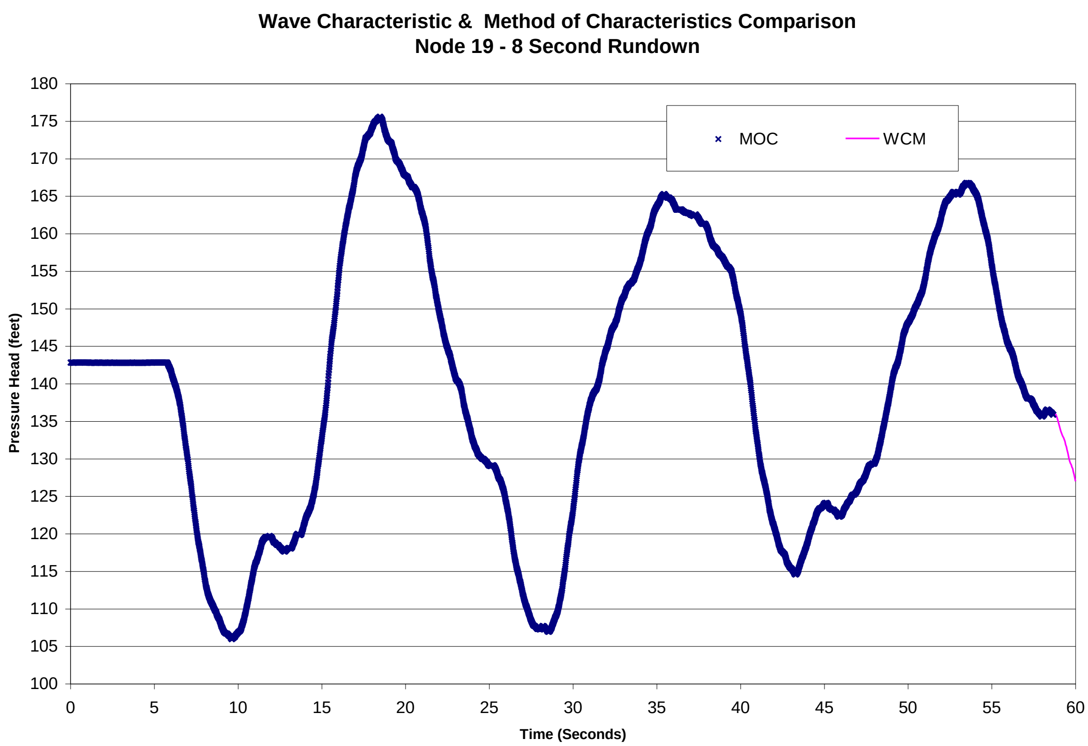


Figure 7 Comparison of Results for Example 2

CONCLUSIONS

Transient (waterhammer) analysis is essential to good design and operation of piping systems. This important analysis can be done using the mathematically based Method of Characteristics (MOC) or the Wave Characteristic Method (WCM) based on the action of pressure waves. The MOC and WCM methods are both capable of accurately solving for transient pressures and flows in water distribution networks including the effects of pipe friction. The MOC requires calculations at interior points to handle the wave propagation and the effects of pipe friction. The WCM handles these effects using pressure waves. Therefore, for the same modeling accuracy the WCM will normally require fewer calculations and faster execution times. In addition, the number of calculations per time step does not increase for the WCM when more accuracy is required. Because of the difference in calculation requirements and the comparable accuracy of the two techniques, the use of the WCM will be more suitable for analyzing large pipe networks.

REFERENCES.

Boulos, P.F., Wood, D.J. & Funk, J.E., 1990. A Comparison of Numerical and Exact Solutions for Pressure Surge Analysis. Proc. of the 6th International BHRA Conf. on Pressure Surges, A.R.D. Thorley editor, Cambridge, England

Boulos, P.F., Lansey, K.E. & Karney, B.W., 2004. *Comprehensive Water Distribution Systems Analysis Handbook for Engineers and Planners*. MWH Soft, Inc. Publ., Pasadena, CA.

Boulos, P.F.; Karney, B. W., Wood, D.J.; & Lingireddy, S., 2004, Hydraulic Transient Guidelines For Design, Operation And Protection Of Water Distribution Systems (Under Review

Boulos, P.F.; Wood, D.J.; & Lingireddy, S., 2003. Users Guide for H_2OSURGE – The Comprehensive Transient/Surge Analysis Solution for Water Distribution Design, Operation, Management and Protection. MWH Soft, Inc., 300 North Lake Avenue, Suite 1200, Pasadena, CA.

Chaudhry, H. M., 1987. *Applied Hydraulic Transients*. Van Nostrand Reinhold Co., New York, NY.

Friedman, M.C., 2003. Verification and Control of Low Pressure Transients in Distribution Systems. Proc. 18th Annual ASDWA Conf., Boston, MA.

Funk, J.E. et al, 1999. Pathogen Intrusion Into Water Distribution Systems Due to Transients. Proc. ASME/JSME Joint Fluids Engrg. Conf., San Fancisco, CA.

Karim, M.R., Abbaszadegan, M. & LeChevallier, M.W., 2003. Potential for Pathogen Intrusion During Pressure Transients. *Jour. AWWA*, 95:5:134.

Karney, B.W. & McInnis, D., 1990. Transient Analysis of Water Distribution Systems. *Jour. AWWA*, 82:7:62.

Kirmeyer, G.J. et al, 2001. *Pathogen Intrusion Into the Distribution System*. AWWA Res. Fdn. and AWWA, Denver, CO.

Larock, B.E.; Jeppson, R.W.; & Watters, G.Z., 1999. *Hydraulics of Pipeline Systems*. CRC Press LLC, Boca Raton, FL.

LeChevallier, M.W., 1999. *The Case for Maintaining a Disinfectant Residual*. Jour. AWWA, 91:1:86.

LeChevallier, M.W. et al, 2003. *The Potential for Health Risks From Intrusion of Contaminants Into the Distribution System From Pressure Transients.* Jour. Water & Health, 1:1.

Martin, C.S., 2000. Hydraulic Transient Design for Pipeline Systems. *In Water Distribution Systems Handbook*, Larry W. Mays ed., The McGraw Hill Companies, New York, NY.

McInnis, D. & Karney, B.W., 1995. *Transient in Distribution Networks: Field Tests and Demand Models.* Jour. of Hydraulic Engrg. ASCE, 121:3:218.

Rossman, L.A., 1993. *User's Manual for EPANET.* US Environmental Protection Agency, Drinking Water Research Division, Risk Reduction Engineering Laboratory, Cincinnati, OH.

Streeter, V.L. & Wylie, E.B., 1967. *Hydraulic Transients*. McGraw Hill Book Co., New York, NY.

Thorley, A.R.D., 1991. *Fluid Transients in Pipeline Systems*. D. & L. George Ltd Publ., Herts, England.

Watters, G.Z., 1984. *Analysis and Control of Unsteady Flow in Pipelines*. Second Edition, Ann Arbor Science, Boston, MA.

Wood, D.J., Dorsch, R.G. & Lightner, C, 1966. *Wave Plan Analysis of Unsteady Flow in Closed Conduits.* Jour. of Hydraulics Division ASCE, 92:2:83.

Wood, D.J.; Lingireddy, S.**,** Boulos, P.F.; Karney, B. W., & McPherson, D. L.,2004 *Numerical Methods For Modeling Transient Flow In Water Distribution Systems* (under review)

Wood, D. J.; & Funk, J.E., 1996. *Surge Reference Manual - Computer Analysis of Transient Flow in Pipe Networks*. University of Kentucky, Civil Engineering Software Center, Lexington, KY.

Wood, D.J.; Lingireddy, S.**,** and Boulos, P.F., 2004 *Pressure Wave Analysis Of Transient Flow In Pipe Networks* MWHSoft Inc. Pasadena, CA. (in press)

* This paper was presented by Dr. Wood as the Simon Freese Lecture at the 2004 ASCE annual meeting and appeared in the ASCE Journal of Environmental Engineering, Aug. 2005

Appendix 2   Pipe2010 : Surge Modeling Tips and Procedures

Length Tolerance

Pipe lengths (or wave speeds) in the model must be adjusted so each pipe will be a length – wave speed combination such that the pressure wave will traverse the pipe in a time which is an exact multiple of the computational time increment. The Pipe Segment Length Tolerance is the maximum difference between adjusted pipe lengths in the model and actual system. For example if we use 20 the largest difference between the model adjusted lengths and actual length is 20 feet (say 385 feet for a 405 foot pipe). While the shortest pipe in the model often does set the time step this is not always the case. We determine the largest time step we can use and meet the length tolerance for all pipes in the model.

Using Pump Files

There is an inherent problem using a pump file and this is that we only can use one point to match a pump to a specific pump file. Since we will almost never have the 4 quadrant pump data for a particular pump this is the best we can do. We have discovered for many pumps if the single point we choose is not near enough (or at) the operating point we get a significantly different steady state solution (because the steady state portion of the pump file will deviate from the actual steady state head/flow curve for the pump). In rare cases the solution may not converge because the pump file is not providing a satisfactory representation of the normal pump curve.

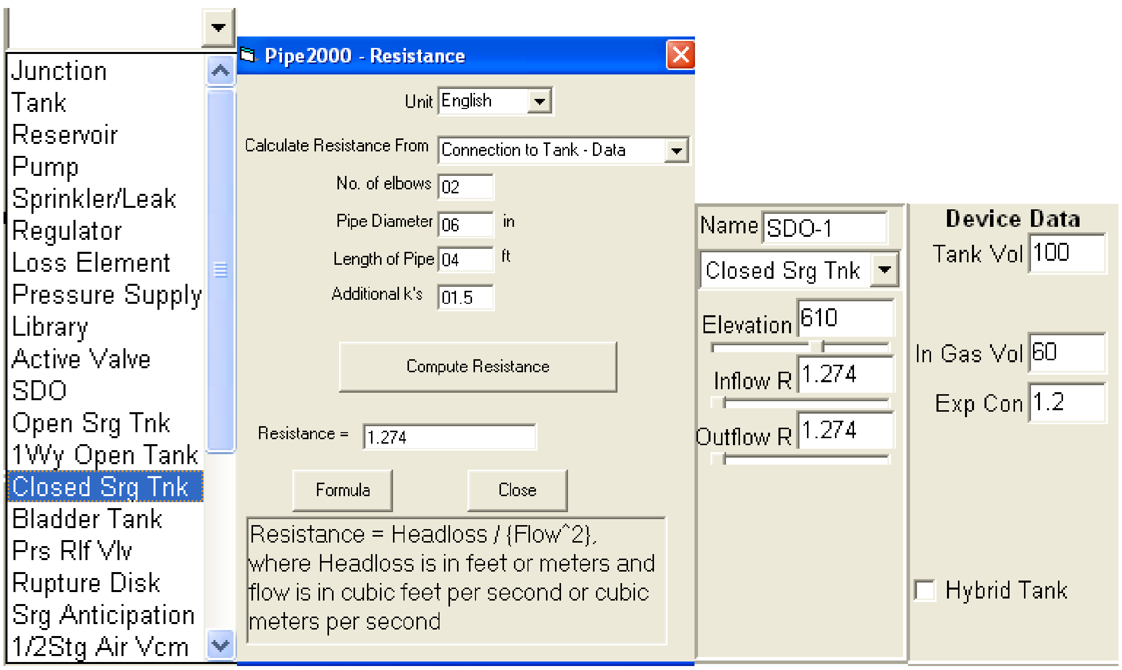
Our experience shows that the best results are obtained when we use a regular steady state pump curve (table) to get the correct operating point for each pump and then introduce the corresponding pump file with the rated conditions set to the operating points obtained using the pump curves. This will assure that your pump files will give satisfactory initial conditions for your surge analysis. Note that the Pump File/Inertia Tool will allow you to select the appropriate pump file and pump/motor inertia for your application.

Alternately, you can use the steady state curve and specify a run down time (1-2 seconds) which works for a pump curve instead of using a pump file and pump trip. We find this usually works very well (gives very similar results to a pump trip) When you do this the initial steady state results will match. If you want to try this I suggest you run both ways and compare the results. It has been my experience that this works well. This is because the pumps normally have a check valve which prevents the pump from running abnormally (such as turbining) so it pretty much stays on the steady state curve during the transient. You really only need to use the pump file if some significant abnormal conditions are encountered.

Sizing Compressor and Bladder Surge Tanks Using Pipe2010 : Surge

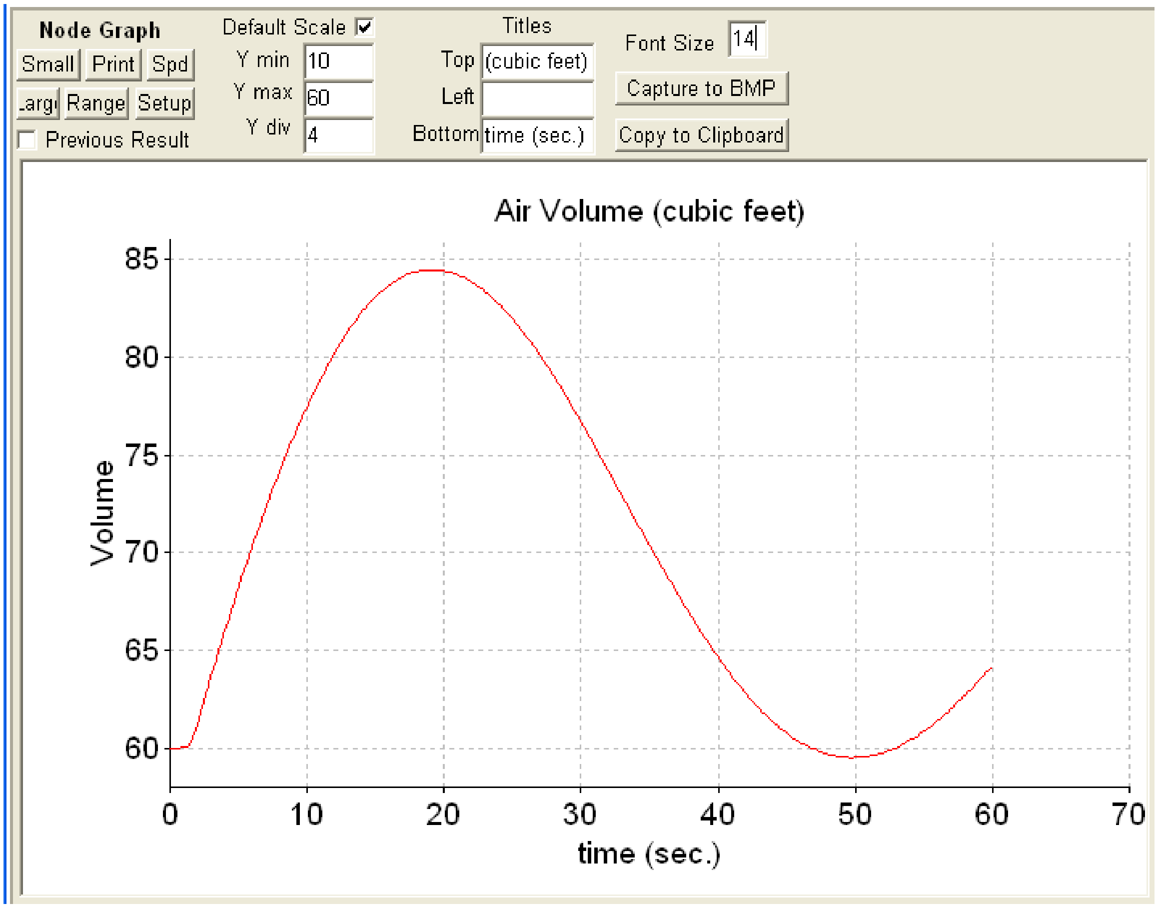
It is relatively straightforward to size closed surge tanks (Compressor and Bladder Tanks) using Surge. The recommended approach is to add a Closed Surge Tank to the desired location using the node selection drop down as shown below. The significant data includes the Inflow and Outflow Resistance which is readily determined using the Surge Resistance Tool as shown below. This shows the calculation for a 4 foot long 6 inch pipe diameter with 2 elbows and an entrance T (K = 1.5). The entrance connection is generally a smaller diameter than the pipeline. However, for sewage lines this diameter is generally larger and a grid may be present which will provide additional resistance.

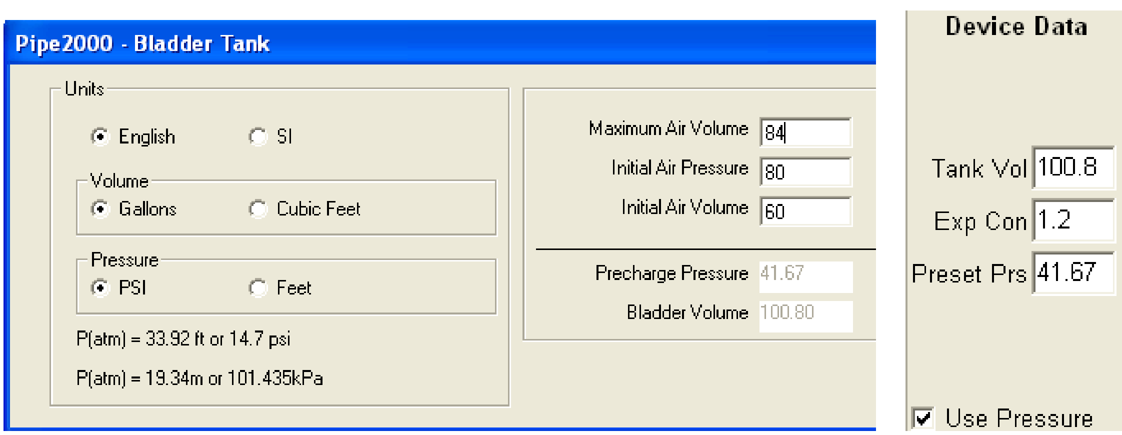
The only other required data is the initial gas volume. This is the parameter which is varied to determine an acceptable size for the surge tank. A rough idea of the required initial air volume can be estimated based on the initial pipeline flow and the overall length of the pipeline to other storage facilities. If, for example, the initial flow is 1000 gpm and it takes 30 seconds for wave reflections to return to the pump you may want to size a surge tank to provide 30 seconds of flow or 500 gallons. It is very easy to vary both the resistances and the initial gas volume until you get an acceptable transient response. For the example below the initial gas volume is 60 cubic feet or around 450 gallons.



Using a Bladder Tank – Once you get an acceptable result for the closed surge tank it is very simple to change from a Closed Surge Tank to a Bladder Tank and enter the corresponding appropriate data for a Bladder Tank. This includes the Tank Volume and Bladder Precharge pressure (head). The Surge Bladder Precharge Tool is designed to calculate the required Tank Volume and Bladder Precharge based on the results obtained running the closed surge tank analysis

The Maximum Air Volume should be determined by plotting the air volume for the closed surge tank as shown below. For this example the initial air volume is 60 cubic feet and the maximum is 84 cubic feet as shown in the plot. The initial pressure at the bladder tank is known. Using this data the Tank Volume and Precharge Pressure are determined using the Bladder Tank Tool as shown below. Now the data for the Bladder Tank is entered as shown. This bladder tank will provide the same response as the equivalent closed surge tank while providing a 20% volume margin (the bladder should expand to maximum of 80%)





Damping of Surges at zero flow

You may notice that the model predicts that transients damp out very slowly when systems are shut down. The reason that the transient doesn't damp out more rapidly is due to the way resistance is modeled. For both the Darcy Weisbach and the Hazen Williams approach the resistance in a pipe is assumed to be a constant term which is determined using the starting conditions. ie the resistance for a pipe section is the initial head drop divided by the initial flow squared. We actually use this approach to calculate pipe segment resistance for all situations. What this does is ignore the fact that pipe resistance increases very much as the flow approaches zero. Therefore the models don't damp the wave nearly as fast as they should when the final flow is zero as in a complete shutdown.

To illustrate this situation put in a valve with only a small initial loss (so it has little effect in the steady state) Then close the valve to 99% after the system shutdown. This creates a large resistance which quickly damps the wave. Without this the small initial pipe resistances damp the waves very slowly.

In general this causes no significant problems in surge modeling. The magnitude of the transients which are generated by an event are not really affected significantly by using a constant pipe resistance - just the rate of damping. Note that this situation only shows up (slow damping) for systems where the final flows are zero - if the final flow is non zero then the constant pipe resistance works quite well to provide damping effects

Handling Cavitation at SDO Devices

Because of various practical concerns and modeling complexities we do not calculate cavitation at SDO devices (ie. pressures can drop below cavitation). There are several reasons for not calculating cavitation at these locations and some things you can do to address this:

1) If the SDO device is supposed to provide water or air flow to the system (like a surge tank or air valve)) this situation tells you that the SDO resistance is too high and can't supply fluid fast enough. You need to use a larger connection (lower resistance)

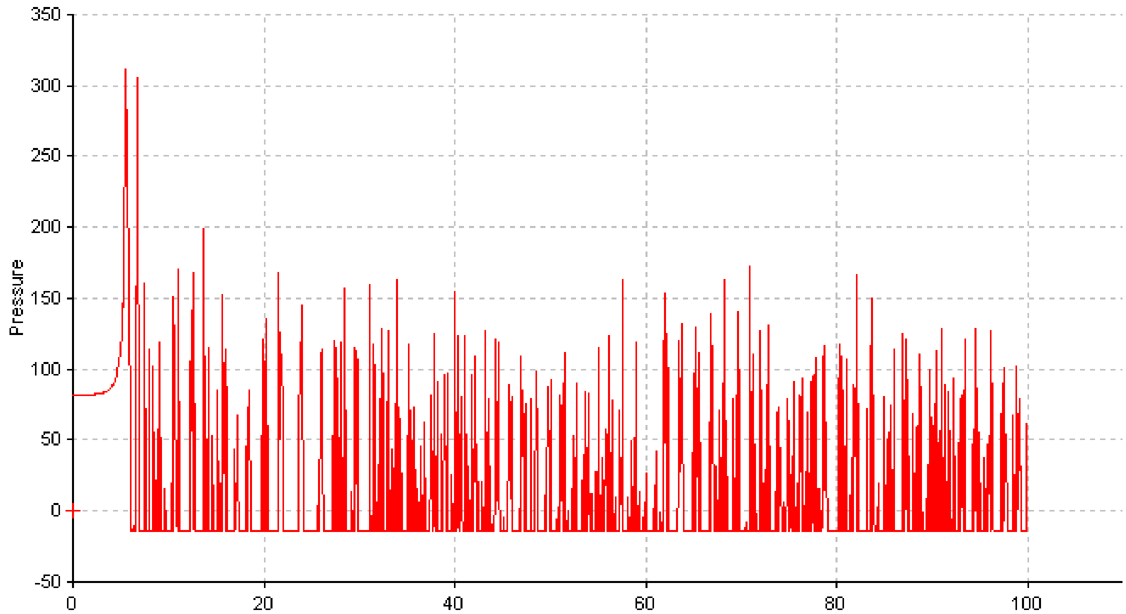
2) If the SDO device does not activate (like a pressure relief valve) you can replace it by a junction which will handle cavitation.

3) You can locate a junction node near the device and that will simulate the cavitation near the device but still allow the device to operate.

We don't plan to change this approach to cavitation at SDO devices but the above technique should handle situations when the user is concerned because pressures fall below cavitation at SDO devices.

Surge Model Results – Excessive Pressure Spiking

Sometimes the results of a transient analysis show excessive spiking of the pressure as shown



The solution may continue as pressure spikes and no final steady state result will be reached. The spikes may even grow and reach very high values. This occurance is almost always due to:

1) Cavitation – spikes generated due to cavity collapse
2) Check Valve action – opening and closing of CV's. A review of the tabulated results report will indicate whether this action is occurring because check valve action is noted in this report.

Either one or a combination of these situations can produce this type of result.

If this type of solution occurs due to check valve action at a pump which has been shut down then the pump is operating in an abnormal fashion (flow reversals, etc.). Therefore, it is essential that a pump file be used in the analysis and the pump trip option used for the pump shutdown. In this manner the behavior of the pump can be calculated. Also the effects of inertia and check valve properties can be evaluated.

When these results are obtained it is important to view the results more in a qualitative than quantitative manner. The actual calculated magnitude of the spikes are very sensitive to the system data and small changes can significantly affect the magnitude of the pressure spikes. The important result is that the response is very voliatle and unstable. Because of the sensitivity of actual spike magnitudes to the timing of the events and data it is not reasonable to compare solutions based on the highest calculated pressure spikes. The solutions are just too sensitive. What can be concluded is that the transients can be unstable and excessive pressure spikes are possible.

If you want to futher evaluate the cause of an unstable result you can:

1) Set the default Cavitation Head to a very low value (such as -1000 ft. (m)). When this is done and cavitation and the resulting unstable solution does not occur you will know that cavity collapse is the cause of the pressure spiking..
2) Either remove check valves or set them to non reopening type so they will not constantly open and close.

These actions should allow for the calculation of a stable response and will allow you to evaluate the cause of the instability for your system.

Using Reduced Wave Speeds in Pump Stations

If the actual distances between a pump and a surge tank are small we find that the effects of surges between a pump and surge tanks are often exaggerated by the models - usually because the model has a greater length between pumps and the surge tank than reality. This often results in pressure spiking which, in turn, affects check valve action leading to increased unstable pressure spiking. Because check valves are modeled without added inertia they will respond quickly to pressure fluctuations. Some adjustment in the input data may be justified to reduce these effects. Of course, the worst possible conditions are predicted if no attempt is made to adjust the data.

.By putting in a shorter length and lower wave speed the model reacts more like the devices are closer together and doesn't allow unrealistically high transients to occur between the pump and surge tank. In addition the pump action will release air and “soften” the water further justifying the use of a reduced wave speed. The alternative is to run with a much smaller length accuracy which will greatly increase computational time. If there is a surge tank downstream from the pump station I feel quite comfortable doing this and often recommend this approach to users. I usually suggest lowering the wave speed and the pipe length by a factor of 4. This will result in the same travel time between the pump and the surge tank and correctly transmits the flow changes with correspondingly smaller pressure changes.

Check Valve Modeling and Responses

A check valve (CV) is modeled in Surge as follows: The CV will start to open (or close) whenever the pressure gradient reverses. During a period of opening or closing the CV setting changes over each time interval at a rate of $\Delta t/T_{CV}$ where Δt is the computational time interval and T_{CV} is the closure time (delay) for the check valve. If the gradient reverses during the period that the CV is open the opening (or closing) will be reversed and proceed at the same rate.

This CV model will often produce an unstable response due to wave actions on both sides of the CV which lead to rapid valve opening and closing. This action produces pressure waves which are reflected back to the valve and provide additional impetus to the instability. Many times the pressure spiking causes vapor cavities to form and collapse which further add to the instability.

This action is all based on an accurate surge analysis for the check valve model used in Surge. When you get this response you should realize that CV action can produce unstable responses and large pressure surges. However, for a number of reasons the model may over predict the instability. Some factors are:

1 Air released which dampens the action.
2 The model assumes air instantaneous response for the check valve (it will start to open or change directions at the instant the pressure gradient switches.)
3 If the suction line is modeled rapid pressure changes occur in the suction line increasing the CV action.

4 Time delays for closing may allow significant velocity to develop just prior to closure – causing pressure surges.

There are several ways to reduce or eliminate the instable CV responses obtained by your model..

1 Use a non reopening CV. This device will close only one time and will remain closed.
2 Eliminate the suction line by modeling the pump connected directly to the supply reservoir.
3 Reduce the CV closing time (time delay).

.
In general I do not believe these actions causes any major problems in surge modeling and can be employed. However, situations where check valve action leading to pressure spiking and failures have been observed. A conservative design will model the piping and devices within the pump station and address any predicted instabilities.

Trapped High Pressure Liquid

On occasion we have observed a shut down of a pipe system where a higher than expected pressure (above pump shutoff head) remains in the system. This can occur when a downstream valve is closed before the upstream pump is shutdown. Due to the valve closure the pressure in the pipe increases as the pump continues to pump. When the pressure at the pump exceeds shutoff pressure the CV at the pump subsequently closes and the liquid trapped between the CV and the downstream valve is compressed (at an elevated pressure) The CV can't reopen because of the elevated pressure so the final pressure of the trapped liquid is higher than it would be if the CV is not there and the pump provides the pressurization at shutoff head. This condition can exist even after the pump is shut off.

Comparing The MOC and the WCM

The following statement appears on www. Haestad.com.

========================

HAMMER uses the Method of Characteristics - the benchmark standard and unquestionably the most rigorous and robust algorithm for hydraulic transient flow analysis.

Algorithms like the Wave Plan Method (a.k.a. the Wave Characteristic Method) compromise the accuracy of solutions by only computing results at junctions. The Method of Characteristics computes results along the pipeline, accurately capturing critical changes that could otherwise be missed.

=========================

The above statements from Haestad's www site is misleading and just plain wrong. The following items address the issue of the Method of Characteristics (MOC) vs the Wave Characteristic Method (WCM) methods of transient analysis. The above statement appears to be an attempt to put a positive spin on an enormous disadvantange of the MOC – the computational inefficiency of the MOC technique.

1) An acceptable technique for solving the basic pipe system momentum and continuity transient flow equations produces a correct solution. Since the solution techniques are not exact mathematical solutions a correct solution is one which satisfies all the basic equations and boundary conditions with an acceptable degree of accuracy. Although there are multiple techniques for obtaining a solution there is only one correct solution for a given problem. The concept that one viable technique (MOC) is more rigorous and robust than another (WCM) is nonsense since they both produce essentially the same result. The fact that the MOC and the WCM produce the same result is documented in several technical journal articles (listed at end of this section)
2) The efficiency of the solution technique used is an entirely different concept. Certainly different computational procedures can be used to obtain the correct solution and the WCM happens to be orders of magnitude more computationally efficient than the MOC. This is particularly important because transient flow analysis in a sizable piping system requires an extremely large number of computations and an efficient algorithm is necessary to handle larger piping systems in a timely manner
3) The implication that the WCM compromises accuracy because it computes results only at junctions is also flawed. The WCM computes results at all devices in the system and at junctions and any desired additional location. Good pipe system modeling (steady state and transient) always dictates that modeling nodes are placed at critical high and low points which are normally the only points of real concern along a pipeline. No engineer would suggest that we add a node every 20-40 feet in every pipe in the steady state pipe system model because we might miss a critical event. This would add great difficulty and overhead to the modeling and analysis and rarely (if ever) provide any additional useful information. Yet this is exactly what the above statement implies
4) It needs to be stressed that the only transient event (critical change referred to in the above statement) occurring within a pipeline which affects the results is the formation and analysis of a vapor cavity. Vapor cavities normally occur at a device such as a pump or valve. When they occur within a pipeline they normally form at local high points. As noted above good modeling will place a node and define the elevation at local high points within the pipeline. An accurate prediction of this event within a pipeline requires that the elevation of the location is known precisely. A difference of just a few feet will compromise this calculation. MOC models normally interpolate elevations at interior points. This approximation will affect the accuracy of the prediction of the formation of a vapor cavity – the critical change referred to in the above statement. Certainly nodes placed precisely at high points will adequately predict the occurance of cavitation.

5) The WCM technique for solving transient flow in piping systems requires that solutions be calculated at all nodes (pumps, valves, etc), junctions, and additional nodes (if any) inserted at critical locations. The MOC technique makes the same calculations plus many additional required ones at numerous internal locations. The MOC technique requires these internal calculations to handle the wave propagation and frictional effects. Pressure wave action is incorporated into the WCM method to handle the wave propagation and the effects of wall friction and requires just one additional calculation for each pipe section. The result of this is that the MOC usually requires order of magnitudes more calculations than does the WCM to obtain the same solution. Because calculations are required at small time increments (often .01 seconds or less) and simulations of 60 to 300 or more seconds may be necessary, millions of calculations are often needed. Using a technique which increases this requirement by orders of magnitude to get the same result doesn't make much sense. Even with modern fast computers the time requirements for

$$\frac{\partial H}{\partial x} = -\frac{1}{gA_L}$$

handling many water distribution systems could be very significant (1 minute (WCM) vs 45 minutes (MOC), for example). Interestingly, the Method of Characteristics was originally developed for solving open channel transient flow problems with relatively slow moving pressure waves (compared to the fast wave speeds of closed conduit flows) and that speaks volumes about the inefficiency of MOC method when applied to closed conduit flows.

Boulos, P. F.. Wood D.J and Funk, J.E. "A Comparison of Numerical and Exact Solutions for Pressure Surge Analysis," Chapter 12, Proceedings, 6th International Conference on Pressure Surges, Cambridge, England, Oct. 1989, pp. 149-159.

Wood, D.J.; Lingireddy, S., and Boulos, P.F., 2004 *Pressure Wave Analysis Of Transient Flow In Pipe Distribution Systems,* MWHSoft Inc. Pasadena, CA. (2005)

Wood, D.J.; Lingireddy, S., Boulos, P.F., Karney B. W. and McPherson, D. L. Numerical Methods for Modeling Transient Flow in Distribution Systems, Journal AWWA, July 2005