Pipe2010 : Surge Modeling Tips and Procedures

Dr. Don J. Wood & Dr. Srinivasa Lingireddy

2010

This Document contains a collection of answers to often asked questions, tips and advice for folks doing surge modeling. As always please contact Dr. Wood, Dr. Lingireddy or one of our other support staff if you have questions.

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.

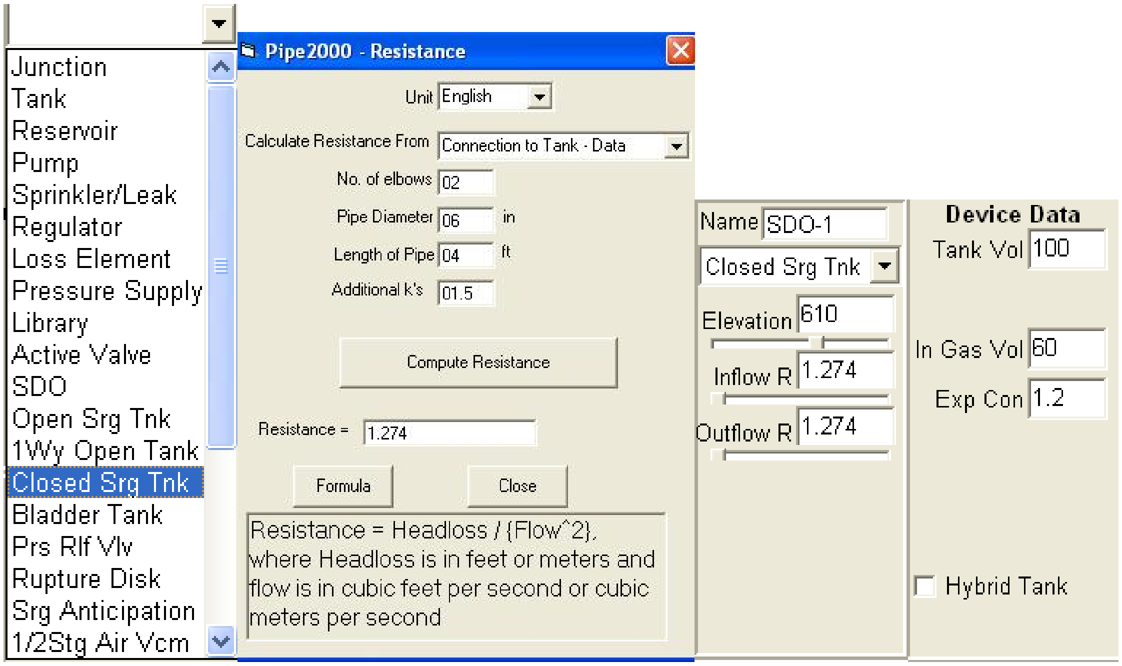
Our experience shows that the best results are obtained when we use a regular steady state pump curve (table) to get the correct operating point for each pump and then introduce the corresponding pump file with the rated conditions set to the operating points obtained using the pump curves. This will 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 Surge2000

It is relatively straightforward to size closed surge tanks (Compressor and Bladder Tanks) using Surge2000. 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 Surge2000 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.



Unit English

Calculate Resistance From Connection to Tank - Data

No. of elbows 02

Pipe Diameter 06 in

Length of Pipe 04 ft

Additional k's 01.5

Resistance = 1.274

Resistance = Headloss / {Flow^2}, where Headloss is in feet or meters and flow is in cubic feet per second or cubic meters per second

Name SDO-1

Closed Srg Tnk

Elevation 610

Inflow R 1.274

Outflow R 1.274

Tank Vol 100

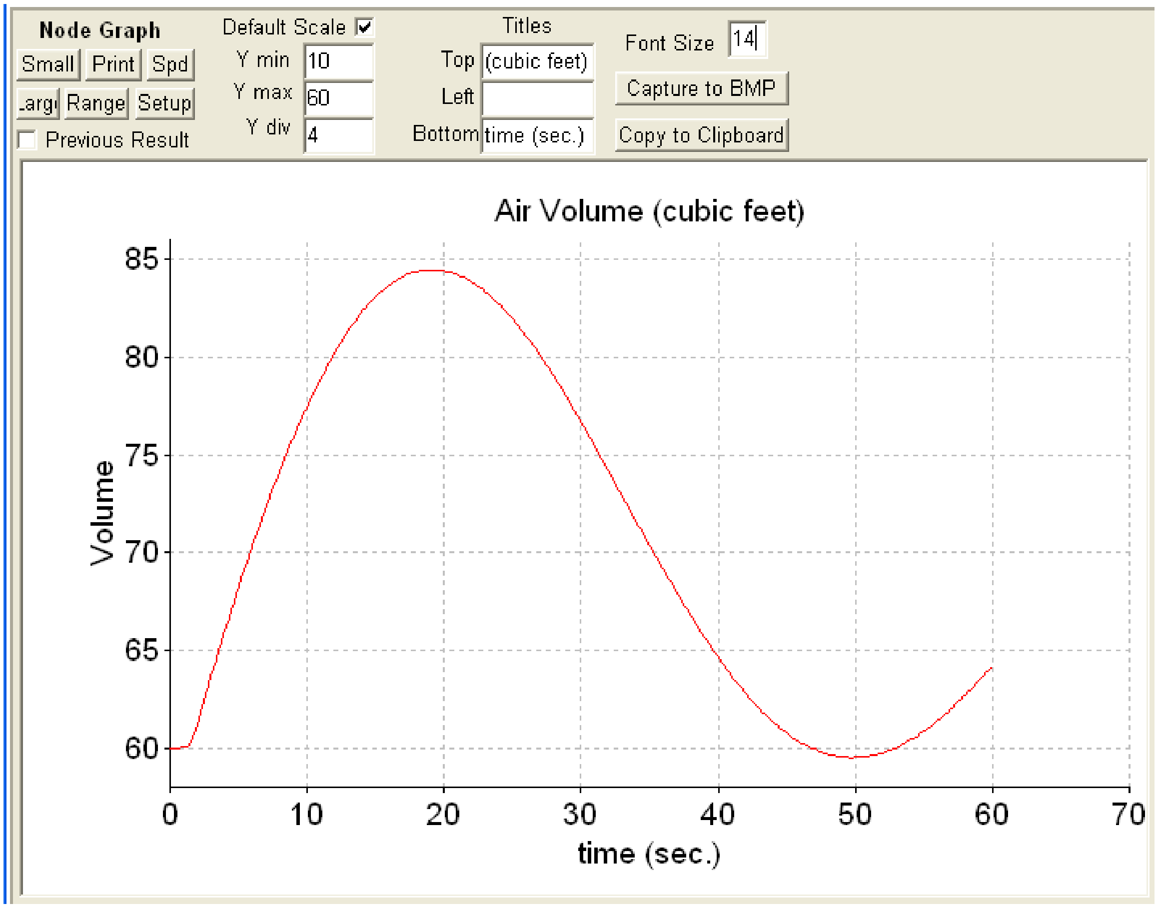
In Gas Vol 60

Exp Con 1.2

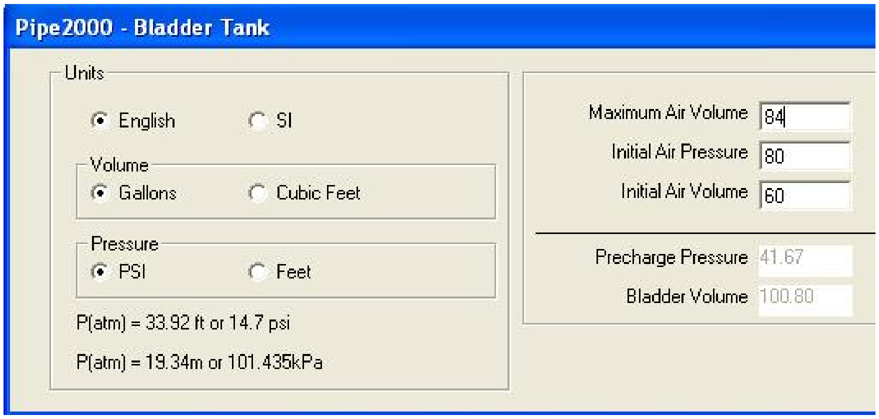
☐ Hybrid Tank

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 Surge2000 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%).

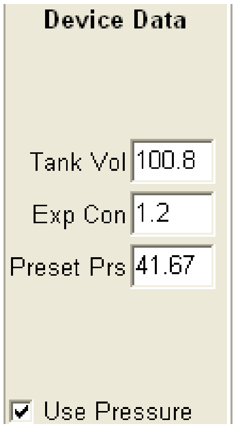


Default Scale ☑
Y min 10
Y max 60
Y div 4
☐ Previous Result
Top (cubic feet)
Left
Bottom time (sec.)
Font Size 14



Units
● English ○ SI
Volume
● Gallons ○ Cubic Feet
Pressure
● PSI ○ Feet
P(atm) = 33.92 ft or 14.7 psi
P(atm) = 19.34m or 101.435kPa

Maximum Air Volume 84
Initial Air Pressure 80
Initial Air Volume 60
Precharge Pressure 41.67
Bladder Volume 100.80



Tank Vol 100.8
Exp Con 1.2
Preset Prs 41.67
☑ Use Pressure

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, i.e. 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 (i.e. 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)

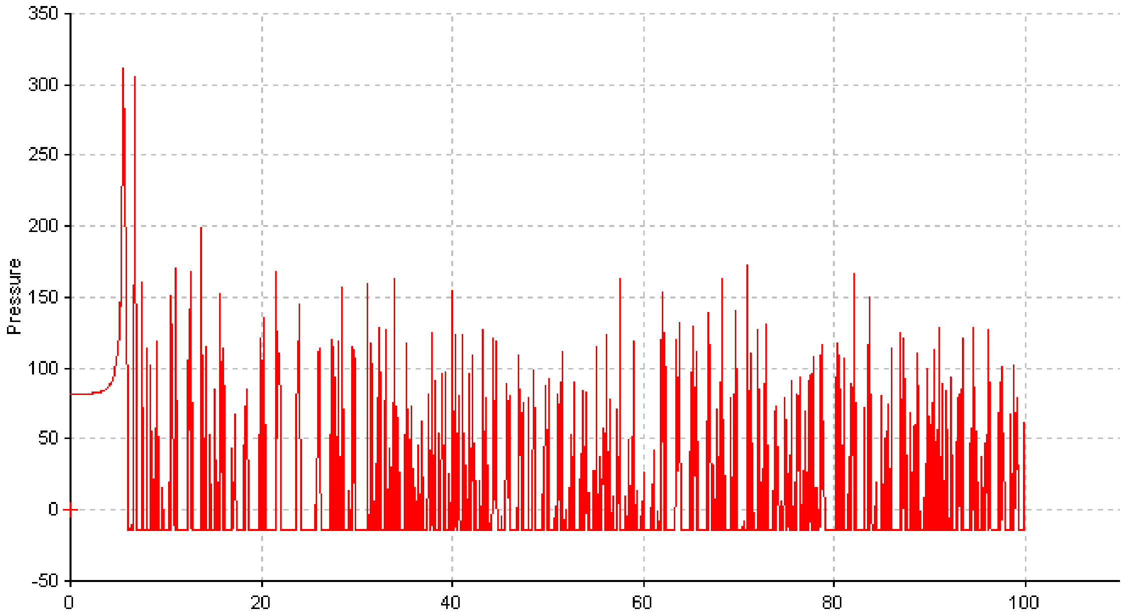
2) If the SDO device does not activate (like a pressure relief valve) you can replace it by a junction which will handle cavitation.

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 occurrence 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 volatile 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 further 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 2000 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 Surge2000. 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).

.

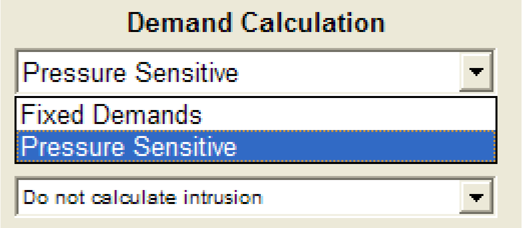
In general I do not believe these actions causes any major problems in surge modeling and can be employed. However, situations where check valve action leading to pressure spiking and failures have been observed. A conservative design will model the piping and devices within the pump station and address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

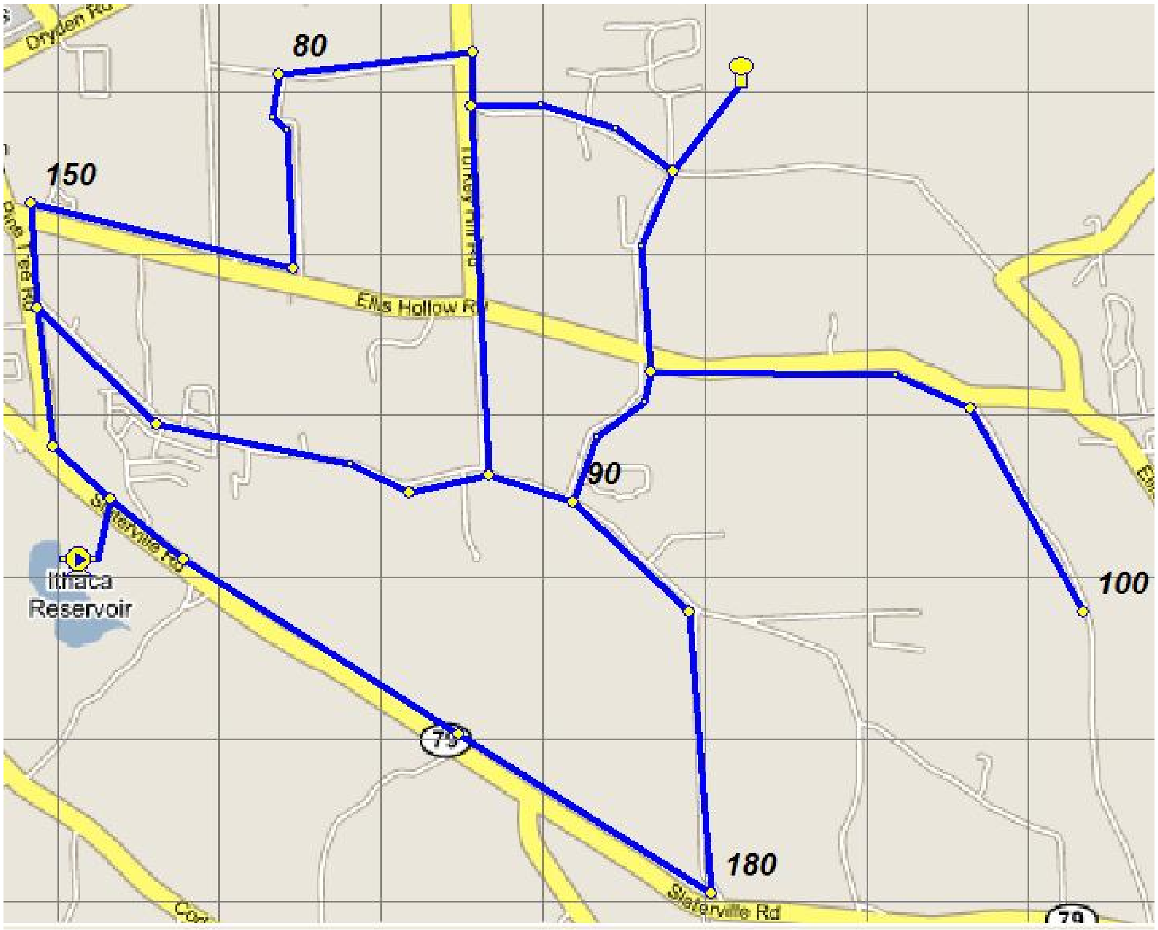
Trapped High Pressure Liquid

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.

Fixed vs. Pressure Sensitive Demands

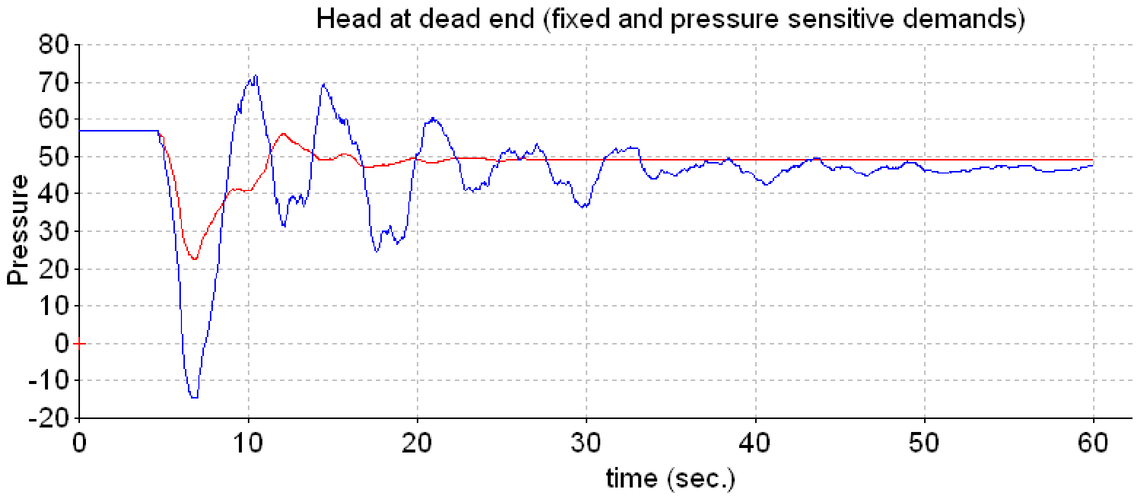
The small system below is Class Exercise 2 from the Examples Manual. A surge analysis is carried out for a pump trip using both Fixed and Pressure Sensitive Demands. The Demand Calculation selection is made on the System Data Screen using the following Menu:





Surge Results – The pressure vs. time plot for the dead end node (right side of system) is shown in Figure CE2-17(below and in Examples Manual). The blue trace is the result for Fixed Demands. When demands are fixed they remain constant even when the pressure is reduced. Pressure Sensitive Demands vary with pressure and will be reduced as the pressure goes down. These are modeled by analyzing the effect of an orifice at the junction. The orifice coefficient is determined from the initial pressure difference (inside – outside) and demand at the junction and is calculated as flow (gpm) / (pressure(psi)^0.5. For example, if a demand of 20 gpm is imposed at a junction where the initial pressure difference is 64 psi then the orifice coefficient will be (20/(64)^.5) = 2.5. This will have a very significant effect on the pressure transient. This option is implemented on the System Data/Simulation Specs screen as shown above. The same result (pressure at the dead end) is shown on Figure CE2-17 using Pressure Sensitive Demands. Note that if the pressure inside the pipe system drops below the outside pressure there will be intrusion into the piping system.

The On-Line Help has some additional discussion about Fixed and Pressure Sensitive Demands and Intrusion Calculations.



Dynamic vs Static Friction

Doing a Surge Analysis using Static Friction means that a single value of Resistance is computed for each pipe based on the initial flow (Qi) and initial head drop (DHi) using (Resistance = DHi/Qi^2). This value for Resistance is then used to compute the effect of friction throughout the entire analysis. Since neither the Hazen Williams or the Darcy Weisbach head loss equations are based on a resistance that remains constant with changing flowrates the use of the constant resistance may result in final steady state solution for the transient analysis may differ (usually slightly) from that calculated from a steady state analysis using resistances based on the final flowrates. By introducing Dynamic Friction the Resistance for pipes is recalculated each time step so that it is correct for the flowrate at this time. The final steady state solution for the transient analysis will then agree very closely with the steady state solution for those conditions.

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 disadvantage 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 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

Convergence Issues Due to Pump Curve Shape

Convergence problems can occur when using a pump curve which is not concave downward throughout its entire range. For example, the curve shown in red has a substantial region where the curve is concave upward. This particular curve would not allow a correct solution to be computed. The curve in green is concave downward throughout its entire range and will produce a good solution. Note that even though the two curves characteristics are quite different in the range up to 600 gpm they are very similar beyond 600 gpm. Thus solutions beyond 600 gpm will closely match the operating conditions for either curve.

If you encounter a problem while using a curve with concave upward regions (like the red curve) you should replace the curve with one which is concave downward throughout and closely matches the initial and final steady state operation.