Profile-Induced Column Separation and Rejoining during Rapid Pipeline Filling

A Malekpour, A.M.ASCE1; and B. W. Karney, M.ASCE2

Abstract: Water column separation during rapid pipeline filling is numerically explored using a one-dimensional (1D) model that employs the method of characteristics to solve the governing equations and the well-known discrete gas cavity model (DGCM) to represent column separation. Extensive numerical experiments helped to identify the conditions under which column separation may occur during the rapid filling and to gain a physical sense of when the local rejoining pressures can be most severe. The major findings are that local V-shaped pipeline profiles following knee points are prone to the occurrence of water column separation and that the magnitude of the resultant overpressures markedly depends on the geometrical and hydraulic characteristics of the profile. Significantly, the propagation and reflection of the first pressure spike following column rejoining at a knee point can cause the onset of column separation in other parts of the pipe system. It is also found that short pipes must usually be steep to give rise to column separation during rapid filling, whereas longer pipes require much milder slopes; however, potential overpressures are significantly higher in short, steep pipes. Overall, the paper seeks to provide a physical interpretation of the numerical results to provide design and operational insight into this potentially important phenomenon. DOI: 10.1061/(ASCE)HY.1943-7900.0000918. © 2014 American Society of Civil Engineers.

Author keywords: Rapid filling; Column separation; Transient analysis; Water hammer; Method of characteristics.

Introduction

Negative pressures may occur in pipe systems under a variety of conditions. Down surge pressures triggered by pump power failure induce negative pressures if the falling hydraulic-grade line (HGL) intersects the pipeline profile. Sudden closure of inline valves can also produce severe negative pressures on the downstream side, yet even the overpressures on the valve’s upstream side possess the potential to produce negative pressures if they are modified by reflections at reservoirs, dead ends, or other devices (Wylie and Streeter 1993). Negative pressures can be also generated during rapid filling of empty pipelines with undulating profiles when the front of the water column passes a knee point on the pipeline profile (Malekpour and Karney 2011; Liou and Hunt 1996). Regardless of the cause, negative pressures in pipelines are often associated with damaging events including structural buckling (Chaudhry 1987), intrusion or ingress of contaminants (Funk et al. 1999), compromised integrity of pipelines joints, and vaporous cavitation which could result in column separation with consequent large water hammer pressures (Bergant and Simpson 1999).

When the local pressure in a pipe system drops to the vapor pressure of the liquid, cavitating flow is established and the water column separates into two relatively independent segments. The flow imbalance within the zone of cavitation gives rise to a vapor cavity which will grow as long as the outflow from the cavitating zone exceeds the inflow. At a later stage of the transient event, following the collapse of the cavity, the momentum exchange between the adjacent water columns can produce huge pressure spikes which propagate throughout the whole connected hydraulic system. The induced pressures can be quite devastating and have been known to be the direct cause of pipe rupture (List et al. 1999).

Since the water column separation was first identified by Joukowsky early in the twentieth century, several systematic experimental studies have been conducted (Bergant et al. 2006; Simpson and Wylie 1991; Martin 1983; Bergant and Simpson 1999; Adamkowski and Lewandowski 2012). These experimental studies have all confirmed that the occurrence of high-pressure spikes is indeed often associated with column separation and have also assisted in validating a variety of workable numerical models (Bergant et al. 2006).

All the aforementioned studies have focused on systems with fully pressurized pipelines. However, column separation can also be caused by other operational conditions. For example, Liou and Hunt (1996) showed that the negative pressure can occur during rapid filling, particularly in pipelines with undulating profiles. However, Liou and Hunt’s rigid column model did not capture the hydraulic events associated with column separation. Indeed, this realm of filling-induced column separation has received little attention, perhaps because it is usually rare and should be easily avoided through patient filling (Watters 1984). However, operators are not always patient, and failures, external events, and poor judgement can sometimes result in lines being rapidly filled. Thus rapid filling is a potential reality, even if an undesired one.

An obvious step toward a better understanding of rapid filling with column separation would be to conduct a physical model study. However, to effectively design a test rig requires the designer to possess a good working understanding of the phenomenon of interest, knowledge of the required strength of the containment associated with the induced pressures, and a detailed knowledge of the parameters to be measured, none of which is currently available.

To avoid a costly trial-and-error physical model study, this paper thus aims to explore rapid filling with column separation through a numerical approach. In the proposed method a numerical model is introduced which can simultaneously track both rapid filling and...
column separation. Four research questions are given particular attention:

- Under what profile configurations is column separation formed in a filling pipe system?
- How does the pipe profile affect the magnitude of the induced overpressure?
- Can pressure spikes associated with column rejoining at one point in the system cause column separation at other points?
- How does the length of the pipe system being filled affect the occurrence of column separation?

Theoretical Background

**Governing Physics and Equations**

Transient flow in closed conduits is governed by two partial differential equations known as the momentum and continuity equations (Wylie and Streeter 1993; Chaudhry 1987), respectively

\[
\frac{\partial V}{\partial t} + \frac{g}{2} \frac{\partial H}{\partial x} = F(V, V) = 0
\]

where \(V\) = velocity; \(H\) = piezometric head; \(D\) = pipe diameter; \(a\) = elastic wave velocity; \(g\) = gravitational acceleration; \(f\) = friction factor in Darcy-Weisbach equation; and \(x, t\) = the independent variable of distance and time, respectively.

These equations are assumed valid as long as system pressures exceed the local vapor pressure. However, if local pressures drop to vapor pressure, cavitation will occur, and column separation necessitates a modified approach.

Cavitating flow naturally gives rise to one or more vapor cavities at specific points of the system whereby the water column effectively separates into independent segments. These events frequently occur when the HGL falls below the pipe profile at knee points, or following wave reflections. It can also occur when two rarefaction waves meet at specific points. These cavitating points function almost as hinges on the HGL, allowing this grade line on either side of the cavity to adopt different slopes. This hinge-like behavior has the potential to allow large local imbalances in flow and the evolution of substantial cavities with high-void fractions.

Distributed cavitating flow zone which is characterized with low-void fraction two-phase flow may be also established in a substantial distance of pipe system which controls the vapor pressure in the whole length of the zone (Wylie and Streeter 1993; Bergant and Simpson 1999). Since the pressure is constant, waves cannot propagate in this zone. Assuming the two-phase flow in this zone is a mixture of liquid water and vapor with a void fraction near zero, it is governed by the following continuity and momentum equations (Bergant and Simpson 1999):

\[
\frac{\partial V_m}{\partial t} + V_m \frac{\partial V_m}{\partial x} + g \sin \theta + \frac{f V_m |V_m|}{2D} = 0
\]

\[
\frac{\partial \alpha_v}{\partial t} + V_m \frac{\partial \alpha_v}{\partial x} - \frac{\partial V_m}{\partial x} = 0
\]

where \(\alpha_v\) = void fraction of the vapor; \(V_m\) = velocity of the mixture; and \(\theta\) = pipe angle with respect to the horizontal.

Complexity quickly arises in practical cases because transient flow can occur in pipe systems constructed in varying terrain, thus giving rise to zones within the pipe having only liquid water, other regions having distributed cavitating flow, and specific locations which experience the formation, growth, and collapse of local vapor cavities. Clearly, handling such a complex flow requires a careful choice of numerical model.

**Choice of Numerical Model**

As has been argued, simulating column separation during filling requires a full dynamic model, one capable of modeling the filling process itself, the formation, growth and collapse of vapor and gas cavities, and any water hammer spikes that result from cavity collapse. Certainly, a wide variety of models have been proposed to simulate the filling of sewers systems, and these could be adapted to the present purpose by considering that the two filling processes are physically similar except for the dominant pace of transients, which tends to be lower in sewers. However, if the filling models used in sewer systems are to be used in the context of filling water conveyance pipelines, they must be adjusted to higher wave-speed values to better capture water hammer events.

Two methods, the so-called shock-capturing and the shock-fitting methods, have the potential of solving the full dynamic one-dimensional (1D) equations (momentum and continuity equations). In the shock-capturing method, transient flow in open channel and pressurized flow is considered through a single set of equations. Indeed, a variety of shock-capturing models have historically been presented including the original (but now rarely recommended) Preissmann slot method (Cunge et al. 1980; Garcia Navarro et al. 1994; Capart et al. 1997; Ji 1998; Trajkovic et al. 1999), the two-component pressure approach (Vasconcelos et al. 2006), and a new version of the slot method (Kerger et al. 2011).

However, all of these approaches suffer from spurious oscillations and numerical instabilities as the acoustic wave velocities grow to values typical of pressure pipe systems. Because the magnitude and trace of the induced pressure rises following water column rejoining crucially depends on the magnitude of pipe acoustic wave velocity, existing shock capturing models can significantly distort the simulation. The alternative is to use shock-fitting approaches which are also widely employed in sewer systems. Song et al. (1983), Fuamba (2003), Cardle and Song (1998), Guo and Song (1990), and Leon et al. (2010) among others have successfully used these simulation methods. In these methods, the pressurization front (the interface) is tracked during each time step, and both the free flow and the pressurized sections are treated by their corresponding theories. Although they do not suffer from the problems associated with the slot method, their performance has not been tested in the context of the water pipe system with undulating profiles. Moreover, it is not known whether the methods are numerically stable when possibly huge water hammer pressure spikes interact with the interface separating pressurized flow from open channel flow. Malekpour and Karney (2011) proposed a shock-fitting model that enables computation of both rapid filling in systems with an undulating profile and the water hammer pressure spikes associated with cavity collapse. Thus, this model is adopted here.

A variety of numerical models have been proposed to capture essential features of the column separation in pipe systems (Bergant et al. 2006). The most popular models have included the discrete vapor cavity model (DVCM), discrete gas cavity model (DGCM), and two-phase flow model. Simpson and Bergant (1994) evaluated the accuracy of five different versions of DVCM and DGVM and concluded that the DVCM produces unrealistic pressure spikes if the number of computational nodes are too high. They found that the DVCM with very small initial void fractions (=10^{-7} m^3) at computational nodes always provided reasonable results.

Adamkowski and Lewandowski (2012) presented a new version...
of DVCM which is in good agreement with the experimental data and more importantly, does not suffer from predicting unrealistic pressure spikes.

A direct method to include the column separation is to solve two-phase flow equations (3,4) in parts of the system running with cavitating flow and to use water hammer equations (1,2) where liquid water is flowing (Wylie and Streeter 1993; Bergant and Simpson 1999). This requires one to numerically track the interface between the distributed cavitating flow and liquid water flow. Considering that the highly dynamic nature of rapid filling may cause column separations in different parts of the system simultaneously, tracking these interfaces can be complex or perhaps impossible (Wylie and Streeter 1993 p. 201). Bergant and Simpson (1999) compared the results obtained from DVCM, DGCM, and a two-phase flow model named the generalized interface vapor cavity model (GIVCM) with experimental data and concluded in the cases they studied that the GIVCM provided slightly better results than the two other models tested.

Taking all this previous work into account, the DGCM is adopted here. Following the accepted recommendations, a small gas pocket (e.g., $10^{-7}$ m$^3$) is assumed within each internal computational node, and the evolution of this gas pocket is treated as an internal boundary condition. Expansion and contraction of the gas pockets mimics the evolution of the flow during the column separation. Experiments have shown that DGCM provides reasonable simulation results even in pipe systems experiencing both local large vapor cavity growth and other regions with distributed cavitating flow (Bergant and Simpson 1999; Simpson and Bergant 1994).

**Numerical Implementation**

The Malekpour and Karney William’s model applies the method of characteristics to a dynamic computation grid with the following assumptions: (1) the pipeline remains full with a well-defined vertical front during the filling, (2) the pipe system is sufficiently vented so that the air pressure remains essentially atmospheric and imposes little retarding force against the filling water column, (3) except in the immediate vicinity of the water column front (where a very short length of fluid is assumed incompressible), the water-pipe systems are (slightly) compressible everywhere, and (4) the frictional flow-resistance relationships for steady flow are a good approximation for the transient flow. Obviously, some of these assumptions are quite easy to relax, but a direct implementation of these assumptions is made here for this preliminary exploration.

As shown in Fig. 1, numerical computations are performed on a dynamic computational grid initially having just one computational cell. As the water column elongates, additional computational cells are sequentially admitted into the computational domain. Yet, the water column front remains within each computational cell for several time steps because the time step is much less than the time required for the water column to fill the computational cell. Such a small time step is in fact enforced because of numerical stability requirements calculated based on numerical considerations associated with the acoustic wave velocity.

To facilitate discretization of the equations, consider Fig. 2 which details the computational cells in the conditions that the computational grid has one, two, and three cells, respectively. The large ovals depicted in this figure represent gas pockets assumed at computational nodes as part of the DGCM method. In the first stage of simulation, when there is just one active computational cell [Fig. 2(a)], the following finite difference equations represent the discrete form of continuity and momentum equations (derived in the appendix), the energy balance between the reservoir and a point just downstream of the control valve, a volume balance for the gas pocket, and an expression for gas pocket pressure evolution, respectively.

\[
\psi(Q_{P0d})_{t+\Delta t} + (1 - \psi)(Q_{P0d}) = \frac{\Delta x}{\Delta t} A = 0
\]

\[
\frac{1}{A \Delta t} \left[(L_f + \Delta x)(Q_{P0d})_{t+\Delta t} - L_f(Q_{P0d})_{t} - \frac{1}{A} \left[\psi(Q_{P0d})_{t+\Delta t} \right.\right.

\left.\left.\right.\right] + (1 - \psi)(Q_{P0d})_{t}^2 - gS_0(\psi \Delta x + L_f) - g\psi(H_{P0d} - Z_{P0d})_{t+\Delta t} - g(1 - \psi)(H_{P0d} - Z_{P0d})_{t} + g\psi(L_f + \Delta x)S_{t+\Delta t} + g(1 - \psi)L_fS_t = 0
\]

\[
\frac{1 + K_V + KS}{gA^2} \left(\frac{Q_{P0u}}{g} \right)_{t+\Delta t}^2 + \frac{L_s}{gA} \left[\frac{(Q_{P0u})_{t+\Delta t} - (Q_{P0u})_{t}}{\Delta t} - Z_{P0u} - H_{res} = 0 \right]
\]

\[
(\psi_{g})_{t+\Delta t} = (\psi_{g})_{t} + \left[\psi(Q_{P0d} - Q_{P0u})_{t+\Delta t} + (1 - \psi)(Q_{P0d} - Q_{P0u})_{t}\right] \Delta t
\]

\[
(\psi_{g})_{t+\Delta t} = \frac{P_{gs}\alpha_{g} \psi_{g}}{\rho g(H_{P0d} - Z_{P0d} - H_V)}
\]

In the above equations, $S$ = energy grade line slope and can be calculated as $S = \frac{\Delta Z_{P0u}}{\Delta x}$; $Q_{P0d}$ and $Q_{P0u}$ = discharge at the upstream and downstream side of the cavity at point P0, respectively; $H_{P0d}$ = piezometric head at the downstream side of the cavity at point P0; $Z_{P0u}$ = $Z_{P0d}$ = pipe elevation upstream and downstream side of cavity at point P0, respectively; $A$ = cross-sectional area of the pipe; $\rho$ = water density; $L_f$ = water column length in the front cell in the
previous time line; \( \delta x \) = water column change during the current
time step; \( V_g \) = cavity volume; \( H_V \) = gauge vapor pressure; \( \varphi_0 \) =
initial void fraction; \( V_r \) = computational cell volume; \( P_0' \) = refer-
ence pressure; \( K_V \) = valve head loss coefficient; \( L_s \) = initially stag-
nant water column length; \( K_s \) = \( fL_s/D \) = head loss coefficient
accounting for the friction loss in the initially stagnant water col-
mn; \( H_{re} \) = reservoir water height; \( \Delta t \) = computational time step;
\( \psi \) = time weighing factor; \( S_0 \) = pipeline slope; \( g \) = gravitational
acceleration; and \( t, t + \Delta t \) = time subindexes for the previous
and current time lines.

To prevent dealing with short pipe segments (which can greatly
increase the simulation time), the initially stagnant water column
trapped between the valve and the reservoir is treated in the bound-
dary equation rather than as a separate pipe. As shown in Eq. (7),
the third term calculates the head required to accelerate water in the
water column preceding the control valve. This term is crucial in
the early phase of the filling because it slows down the water col-
umn by absorbing a significant portion of the driving head supplied
by the reservoir.

The above equations are in a form that permits direct solution
and can be simultaneously solved to obtain the five unknowns:
(1) \( Q_{P0u} \), (2) \( Q_{P0d} \), (3) \( H_{P0d} \), (4) \( (\psi V')_{t+\Delta t} \), and (5) \( \delta x \) at the
time new time level. However, the relations are nonlinear in nature
and must be solved iteratively in this research by the Newton-
Raphson method. Because the unknowns in this stage are
effectively calculated implicitly, the time step is not limited by con-
sideration of numerical instability. However, to obtain greater res-
olution during the early stage of the filling and also for the sake of
consistency with other stages of calculations, the time step is cal-
culated through using the well-known Courant-Friedrich-Lewy
(CFL) stability criterion (Chaudhry 1987). For a system with a sin-
gle pipe, the time step can be easily calculated by \( \Delta t = \Delta x/a \)
where the Courant number is set to one. In complex pipe systems
having multiple pipes, each pipe may require its own time steps as indicated by the following relation:

\[
\Delta t_f = \frac{L_f}{a_j N_j}
\]

where \( L \) = length of the pipe line, \( a \) = wave velocity, \( N \) = integer
number showing the number of division of the pipe, \( \Delta x \) = reach
length, and \( J \) = pipe index.

In practice, it is a great benefit to use a common time step. One
simple solution is to simply impose the least calculated time step
for all pipes. Yet an arbitrary time step implies that the character-
istic lines will be unlikely to meet at the computational grid points
at the pipe junctions, and linear interpolation would thus be re-
quired to extract information of the previous time line. However,
the interpolation error in the method of characteristics dramatically
distorts the solution (Goldberg and Wylie 1983; Ghidaoui and
Karney 1994), so for the sake of accuracy, all test cases in this re-
search are selected in such a way that the Courant number in all
pipes remains at one (that is, so each pipe has an integer number of
reaches based on the smallest time step).

The computations proceed until the water column fills the whole
length of the first computational cell. In case the water column length
infinitesimally exceeds the cell length, the linear interpola-
tion is used to adjust the results. The following generic formula is
employed to modify the unknowns at the computational node in the
upstream side of the front cell:

\[
FV'_{\text{modified}} = FV' + \frac{(\Delta x - L'_j)(FV' + \Delta t - FV')}{\delta x + \Delta t}
\]

where \( FV \) = generic flow variable.

This is the transition point for the next stage in which an addi-
tional cell is appended to the computation. Fig. 2(b) demonstrates
computational cells in this stage. The unknowns at the first com-
putational or boundary node, \( P0 \), can be calculated through simulta-
aneous solution of the Eqs. (7)–(9) coupled with the negative charac-
teristic equation, \( C^- \).

Fig. 2. Typical computational cells in three subsequent stages
The resultant nonlinear equations can be solved with the Newton-Raphson method to calculate the four unknowns: (1) \( Q_{Piu} \), (2) \( Q_{P0d} \), (3) \( H_{P0d} \), and (4) \( \sqrt[4]{g} \). The unknowns in computational node \( P1 \) \( \{Q_{Piu}, \; Q_{P0d}, \; H_{P0d}, \; \sqrt[4]{g}\} \), and \( \Delta x \) can then be obtained through simultaneous solution of nonlinear Eqs. (5), (6), (8), and (9) coupled with the positive characteristic equation, \( C^+ \). All indexes started with \( P0 \) in Eqs. (5), (6), (8), and (9) should now be started with \( P1 \)

\[
C^+: H_{P1d} = H_{P1u} = C_1 - C_2 Q_{P1u}
\]  
(13)

where \( C_1 \) and \( C_2 \) can be calculated as follows (Wylie and Streeter 1993, p. 42):

\[
C_1 = H_{P0d} + \frac{a}{gA} Q_{P0d}; \quad C_2 = \frac{a}{gA} + \frac{f \Delta x}{2gDA^2} |Q_{P0d}|
\]  
(14)

The above procedure is followed until the water column length fills the second computational cell. Computations then proceed with a computational grid having three cells [Fig. 2(c)]. Except the intermediate grid node, \( P1 \), unknowns in two other points, \( P0 \) and \( P2 \), can be calculated by exactly the same approach employed in the previous stage. The unknowns in the intermediate nodes \( \{Q_{Piu}, \; Q_{P0d}, \; H_{P0d}, \; \sqrt[4]{g}\} \) are obtained through simultaneous solution of the Eqs. (8), (9), (11), and (13). By algebraic manipulation, these equations are reduced to the following quadratic equation:

\[
(H_{P1d} - Z_{P1d} - H_V)^2 + 2B_1(H_{P1d} - Z_{P1d} - H_V) - B_4 = 0
\]  
(15)

The analytical solution of the above equations results in the following solutions:

\[
H_{P1d} - Z_{P1d} - H_V = -B_1 \left( 1 + \sqrt{1 + \frac{4}{B_1}} \right) \quad \text{if} \quad B_1 \leq 0
\]  
(16)

\[
H_{P1d} - Z_{P1d} - H_V = -B_1 \left( 1 - \sqrt{1 + \frac{4}{B_1}} \right) \quad \text{if} \quad B_1 > 0
\]  
(17)

where

\[
B_1 = B_2 C_4 C_2 V + 0.5(Z_{P1d} + H_V)
\]

\[
- B_2 (C_2 C_3 + C_4 C_1), \quad B_3 = \frac{B_4}{B_1}
\]

\[
B_2 = \frac{0.5}{C_2 + C_4}, \quad B_3 = \frac{\sqrt[4]{g} \sqrt[4]{g} + 1 - \sqrt[4]{g}}{\sqrt[4]{g}} (Q_{P1d} - Q_{P1u})
\]

\[
B_4 = \frac{P_{new} V_{new} B_2 C_4 C_2}{0.5 \rho g \psi \Delta t}
\]  
(18)

In cases having a large gas volume and low pressure or small gas volume but high pressure, Eqs. (16) or (17) may provide inaccurate results because of miscalculation of the radical. This happens when \( B_1 \) is small (less than 0.001 for example), and Eqs. (16) and (17) are used otherwise.

Thus, Eqs. (19) and (20) are used if \( |B_1| \) is small. The numerical calculation proceeds using this described approach until the third numerical cell is also filled. However, the extension of the method should now be clear; thus, a similar numerical procedure is used successively until the water column reaches the end of the pipe. Junction boundary condition where two pipes meet with different slopes are treated as though they were intermediate nodes, with the only difference being that the characteristic equations now pick up appropriately indexed information specific to each pipe.

When the water column front reaches the end of the pipeline, the rest of the computations continue with a fixed number of computational cells. The downstream boundary condition now changes to the simple state of a free release to the atmosphere. Assuming the HGL remains constant at the elevation of the pipe’s axis at the downstream side of the pipeline (\( Z_{end} \)) and neglecting minor losses, the discharge released to the atmosphere (\( Q_{end} \)) can be simply calculated for the new time line by using the positive characteristic equation [Eq. (13)]

\[
Q_{end} = \frac{C_1 - Z_{end}}{C_2}
\]  
(21)

Numerical Verification

To verify the model results, a filling scenario is considered in a simple (almost trivial) pipe system. The system consists of a frictionless-horizontal pipe conveying water from a constant water level at its upstream side to the atmosphere at its downstream side. The pipe internal diameter, length, and acoustic wave velocity are 1 m, 10,000 m, and 500 m/s, respectively. For reasons that will soon become clear, the reservoir water level is set at the very specific value of 100.8155 m which is 0.8155 m above the pipe installation elevation.

The filling scenario is initiated by suddenly opening the upstream valve. Because the pipe is assumed frictionless, the reservoir head will be converted to velocity head and will produce a constant velocity of 4 m/s along the pipe when the water column front reaches the end of the pipe, and steady state flow is established along the pipe system. Numerical experiment shows that in a time period of 3,000 s, all transients are removed and steady-state flow is established throughout the pipe system. At this time, a sudden closure of the downstream valve induces a stepwise overpressure that propagates to the upstream side at the acoustic wave speed of the pipe. Considering that the overpressure can be analytically calculated by the Joukowskian equation \( \Delta H = \frac{2}{9} \Delta V = 203.8 \) m, the induced piezometric head along the pipe at time 3,012 s is thus analytically calculated as follows:

\[
H(x) = \begin{cases} 
303.8 \text{ m,} & x \geq 6,000 \\
100 \text{ m,} & 0 \leq x \leq 4,000
\end{cases}
\]

Considering a Courant number of 0.8, the proposed scenario is performed by the model for different number of computational nodes and the two-norm of errors over the computational domain are drawn versus the number of computational nodes in Fig. 3. As shown, increasing the number of computational cells by an order
of magnitude reduces the error by a nearly proportional amount. This confirms that the model result consistently approaches the analytical solution as the computational grid is refined and thus, the model achieves the same order of accuracy typical of the method of characteristics.

Fig. 4 compares the numerical solutions with the analytical solution. For different computational nodes, the results are presented for the Courant number of 0.8; when 100 nodes are used, the result is also presented for the Courant number 1. As shown, as expected, the model is somewhat too diffusive for a Courant number 0.8 and high accuracy can only be achieved with a high number of computational nodes. However, with a Courant number 1, the model is not diffusive. Significantly, the same accuracy that can be achieved using 10,000 nodes with a Courant number 0.8 can be reached with a mere 100 nodes when the Courant number is exactly 1.

To more completely validate the model, the experimental data presented by Liou and Hunt (1996) is now considered. The test rig consists of an upstream constant-head water supply reservoir, and a PVC pipe with inside diameter and length of 22.9 mm and 6.66 m (2.29 cm and 666 cm), respectively. The first 3.35 m (335 cm) of the pipe has a downward slope of 2.66° and the remaining portion of the pipe is extended on a 2.25° favorable slope. A quarter-turn ball valve with an inside diameter of 24.4 mm (2.44 cm) is installed on the pipe at 438.2 mm (43.82 cm) from the inlet to isolate the pipes from the reservoir. The average values for the entrance loss coefficient ($K$) and the friction factor in the Darcy-Weisbach equation ($f$) are determined as 0.8 and 0.0242, respectively based on a steady-state test. Twenty filling tests were also conducted and the advance times were recorded in eight timing sections located at 576, 729, 1186, 1793, 3016, 4235, 5455, and 6617 mm (57.6, 72.9, 118.6, 179.3, 301.6, 423.5, 545.5, and 661.7 cm) from the inlet.

Fig. 5 depicts the water column velocity versus the water column length obtained from the Liou and Hunt’s experimental study and the rigid water column model and from the proposed model. As shown, the proposed model closely replicates the experimental, whereas the rigid column model proposed by Liou and Hunt overestimates the velocities. It was found by Malekpour and Karney (2011) that the rigid column could potentially provide almost the same results as an elastic model if the velocity head is accounted for at the reservoir boundary condition. Interestingly, as shown in Fig. 5, ignoring the velocity head in the proposed model generates the same result as that obtained by Liou and Hunt.

**Numerical Exploration**

The proposed model is next employed to explore how column separation might arise during rapid filling with specific shapes of pipeline profile. A typical profile possessing such a potential is shown in Fig. 6. The gravity pipe system connects the upstream reservoir with the water surface elevation of 105 m to atmosphere at an elevation of 100 m through three pipeline segments. All pipes have the same diameter, friction factor, and wave velocity of 500 mm, 0.024, and 1,000 m/s, respectively, with different lengths of 20, 30, and 70 m, respectively. The elevations at the pipe junctions are also shown in Fig. 6. The same initial gas pocket volume, upstream control valve head loss coefficient, and time weighting factor of $10^{-7}$ m$^3$, 0.8, and 1, respectively, are used for all pipe systems considered in this section.

Rapid filling is initiated by suddenly opening the control valve at the upstream side of the first pipe. After 25.3 s, the filling front reaches the end of the pipe system and the numerical computation then continues with a constant number (24) of computational cells. The time history of piezometric head and cavity volume at point A of the system is summarized in Fig. 7. A dramatic pressure head spike of around 110 m is predicted to shock the system 8.5 s after the start of filling.

The physics of this high-pressure spike can be easily explained in terms of the four stages summarized in Fig. 8. After the filling front passes the knee point (point A in Fig. 6), the pressures continually decrease as the water column descends into pipe 2. When the pressures at the knee point drops to the vapor pressure, cavitation occurs and enforces a constant pressure at that point (stage 1). This fixes the HGLs slope in pipe 1 and makes the...
reservoir inject constant flow at this point. This threshold is marked on Fig. 7 as point 1. The HGL slope in pipe 2, however, is not constant and becomes steeper and steeper as the water column further descends pipe 2. The break in the HGL at the knee point clearly shows that the inflow to the cavitating zone is less than the outflow from that area. This results in cavity expansion (stage 2) and prolonged water column separation. When the water column enters pipe 3, the HGL in the downstream side of the cavitating zone becomes progressively flatter until it decreases in pipe 1 when the cavity reaches its maximum size. This point which is marked in Fig. 7 as point 2 is the transition point to stage 3, the cavity contraction stage. From this point, the cavity shrinks because the inflow to the cavity zone is greater than the outflow. When the cavity shrinks to zero, the two water columns in the opposite sides of the cavity zone, having different velocities, suddenly collide and produce the huge pressure head spike of approximately 110 m.

(stage 4). This pressure also propagates outward from its point of creation. Fig. 9 shows the maximum and minimum piezometric head generated throughout the system during the filling. As shown, the pressure spike influences the majority of the system, and almost the entire length of the pipe experiences both full vacuum and high positive pressures.

The magnitude of overpressure depends directly on the difference between the velocities of the colliding columns and the acoustic wave velocities of the pipes. This velocity difference increases with the maximum air cavity volume because the moving column in the cavity’s downstream side has enough time to strongly decelerate before cavity volume reaches zero. Changing the elevations of some profile points obviously adjusts the air cavity volume and also the magnitude of the resulting pressure spike. To observe this numerically, the profile shape is slightly adjusted by decreasing point B’s elevation by a mere 3 m. As shown in Fig. 10, the cavity volume in this case increases from 0.26 to 0.64 m$^3$ and the pressure head spike increases from 110 m to nearly 160 m. Similarly, if the elevation of point B increases by 2 m (Fig. 11), the air cavity volume and the overpressure would decrease.

Each cavity collapse will naturally generate a pressure spike. However, in Fig. 10, there are some pressure spikes which are not followed by a cavity collapse at point A and thus originated from other places. The inspection of the maximum cavity size in each computational cell reveals that the column separation also occurred in many other points than point A, although they were major at just two points. The points are located in pipe 3 which are 75 and 90 m away from the upstream reservoir, respectively. The cavity volume and piezometric head time histories of the first point are depicted in Fig. 12. A similar response, but with lower maximum cavity volume, occurs at the secondary point.
Interestingly, discrete cavities usually appearing in knee points or dead ends are created in a straight pipe, pipe 3. Such an unusual cavity formation is clearly associated with the rapid filling and needs to be understood in that light. By considering the time history of the water column front’s velocity, extreme fluctuations during rapid filling can be observed. These fluctuations are induced through the reflection of the overpressure spikes at the water column front. As the pressure spikes are reflected at the front, they suddenly push the water column forward to the filling zone in which the empty pipe exerts no resistive force. Intense driving forces dramatically accelerate the front to a high velocity. However, the leading water column is too massive to be similarly accelerated, which results in negative pressure and consequent column separation behind the column front. As shown in Fig. 13, the mechanism of the cavity formation and collapse is similar to the previous case with the only difference being that the cavity size is now much larger. As shown in Fig. 16, the overall consequence of the column separation is onset of intense negative and positive pressures along the system.

Although numerical results show that the column separation could be caused by the pipeline profile, slopes in the system considered are too steep to be representative of the conventional water pipe systems. Thus, the question arises whether column separation is also relevant in pipe systems with milder pipe slopes. Such questions can begin to be addressed through analysis of rapid filling in a fairly long pipe system depicted in Fig. 14. Pipe diameters and wave velocities are exactly the same as before, but the pipe friction, lengths, and slopes are different. In this case, the slopes of pipes 2 and 3 are 4 and −8%, respectively, whereas they are 100 and −43% in the previous case. The numerical exploration is conducted with 55 computational cells and continued for quite a long time after the 1,856 s mark when the water column reaches the end of the pipe system. Fig. 15 shows the piezometric head and cavity volume time history at point A. As shown, the mechanism of the cavity formation and collapse is similar to the previous case with the only difference being that the cavity size is much larger. As shown in Fig. 16, the overall consequence of the column separation is onset of intense negative and positive pressures along the system.
spike between the upstream reservoir and the water column front is the obvious explanation of the resulting column separation at this point.

Discussion and Some Practical Implications

The numerical results indicate that a slight change in pipe profile can significantly change the magnitude of the overpressure. Although this makes the quantitative phenomenon’s generalization somewhat difficult, the obtained physical insight could be still assisted in making a qualitative generalization.

Except during the earliest phase of filling, the friction force dominates the filling process in most pipe systems. Thus, the HGL can be accurately estimated by simply drawing a straight line between the location of the filling front and the upstream reservoir (Malekpour and Karney 2011). This provides an excellent tool to roughly predict if a particular pipe system is subjected to the column separation during rapid filling (Liou and Hunt 1996). To check if water column separation could occur at a particular knee point of the system, a straight line, hereafter called a guide-line, can be drawn from a point whose elevation is lower than the knee point by vapor pressure head to the upstream water surface. If the slope of the line is less than the pipe leaving the knee point, column separation will occur. Otherwise, the pipe profile is not sufficiently undulating to generate column separation. This simple rule can be used to draw some important practical recommendations.

First, in the short pipe lines, the guide-slope is often much steeper than in longer pipes, so the pipe downstream of the knee point needs to be much steeper than the short pipe system to ensure that column separation occurs during rapid filling. Thus, shorter pipe systems require steeper HGLs to result in column separation, and for a given pipe system with a uniformly distributed undulation, column separation is more likely at the points further from the upstream reservoir.

Second, systems with shorter pipes often produce much higher velocities in the upstream pipes feeding knee points because they provide steeper HGLs during the cavity formation at knee points. This means that the magnitude of overpressure can be potentially much higher in shorter pipe systems. Assuming water column velocity downstream of the knee point is decelerated to zero once it impacts the upstream column, the maximum possible water hammer overpressure could be simply estimated as \(0.5 g V_g^2\), where \(V_g\) is...
the velocity of the water column established in the upstream water column, a value easily calculated using the guide-slope. This worst condition can be physically imagined if the V-shaped profile which is responsible for cavity growth and collapse have specific geometric and hydraulic characteristics: (1) the descending leg should be long and steep enough to give rise to a large enough cavity; (2) the larger cavity in turn provides the water column with enough time to decelerate in the ascending leg and thus prevent the water columns from an early or premature collision; and (3) finally, the slope of the ascending leg should be sufficiently adverse or resistive to quickly decelerate the water column.

For longer pipe systems the negative pressures will be sustained for a considerable duration during rapid filling. This forces a large air mass out of solution and thus generates considerable free gas in the water column. Free air in the system appreciably alleviates the overpressures because it both cushions the system and dramatically reduces the wave velocity in the system (Wylie and Streeter 1993, p. 10). Even in such cases, the course of events will be fairly well-captured by the model, although the calculated overpressures are expected to be conservative in that they are likely overestimated.

Finally, the presented model can be used to help configure a test rig to study the phenomenon in more detail. Numerical experiments show that even with assistance of the numerical model, exhaustive trial and error is required to ensure that the column separation would occur at the proposed knee point and that the induced water hammer pressure can give rise to the water column behind the water column front as well. This confirms that the numerical study should precede the experimental study in this particular case.

Summary and Conclusions

A numerical exploration approach sheds light on a column separation process that can occur during the rapid filling of gravity pipe systems. To this end, a 1D numerical model is developed to address both rapid filling and column separation simultaneously. The numerical model solves the water hammer equations and treats the column separation as an internal boundary condition by the well-known DGCM.

Numerical experiments show that the column separation is induced by specific conditions in the pipe profile, particularly a V shape in the profile. Generally speaking, the descending leg of the V-shape profile gives rise to cavity formation at the upstream knee point by inducing vapor pressure at that location. Yet, when the filling front enters the ascending leg, the deceleration assists in cavity contraction and collapse. Huge water hammer pressure can result at the instant of collapse and these induced pressures propagate throughout the system and can cause secondary damage and disruption. The magnitude of water hammer pressures is highly sensitive to the pipe profile and only slight adjustments to this profile can dramatically change the predicted water hammer pressures.

The first pressure spikes initiated at the knee points can in turn induce column separation in other parts of the system. In some cases, the water hammer pressure is large enough to give rise to the column separation just behind the water column front. The mechanism hinges on sudden jump of the water column front occurring while it is responding to the overpressures coming from the knee point. This results in intense negative pressure and the column separation behind the water column front. The resultant vapor cavity shrinks and collapses as the suddenly accelerated water column front is then decelerated by gravity. The column separation can be also induced in the upstream side of the knee point because of the reflection of the first pressure spike between the upstream reservoir and the water column front.

Short pipe systems should possess steep slopes to induce column separation, whereas the longer pipe could initiate column separation with much gentler slopes. By the same token, in long pipe lines in which the degree of undulation is uniformly distributed, the water column separation is more likely to occur at points of the system which are more distant from the upstream reservoir. However, the induced water hammer pressure could be potentially much higher in the short pipe system than the longer pipe system.

Although the features of column separation during the rapid filling captured by the model are physically interpreted and are thus expected to occur in field, a physical model study is still required to validate the numerical results. The insights from this study, however, could be employed to configure a test rig and its associated instrumentation.

Appendix. Front Cell’s Discretized Equations

Considering Fig. 2(a), finite difference discretization of the momentum and continuity equations applied to the front cell can be easily derived by utilizing the following simplified form of control volume equations:

$$ F_g + F_p - F_f = \frac{(mV)_{t+\Delta t} - (mV)_t}{\Delta t} + (\rho QV)_{out} - (\rho QV)_{in} $$

$$ 0 = \frac{m_{t+\Delta t} - m_t}{\Delta t} + \dot{m}_{out} - \dot{m}_{in} $$

where $F_g$ = gravity force; $F_p =$ net pressure force; $F_f =$ friction force; $(mV)_t$ and $(mV)_{t+\Delta t}$ are the momentums of the liquid in the front cell for two subsequent time lines, respectively; $(\rho QV)_{in}$ and $(\rho QV)_{out}$ are the momentum fluxes entering and leaving the front cell, respectively; $m_t$ and $m_{t+\Delta t}$ are the masses of the liquid in the front cell for two subsequent time lines, respectively; and $\dot{m}_{out}$ and $\dot{m}_{in}$ are the mass fluxes leaving and entering the front cell, respectively.

Different components of Eqs. (21) and (22) can be discretized using a finite difference box scheme as follows:

$$ (\rho QV)_{out} = 0 $$

$$ (\rho QV)_{in} = \frac{\rho}{A} [\psi Q_{t+\Delta t} + (1 - \psi) Q_t^2] $$

$$ F_g = \psi(F_g)_{t+\Delta t} + (1 - \psi)(F_g)_t = \rho g A S_0 (\psi \ddot{x} + L_f) $$

$$ F_p = \psi(F_p)_{t+\Delta t} + (1 - \psi)(F_p)_t = \rho g A [\psi(H - Z)_{t+\Delta t} + (1 - \psi)(H - Z)_t] $$

$$ F_f = \rho g A [\psi(L_f + \ddot{x})(S)_{t+\Delta t} + (1 - \psi)L_f(S)_t] $$

$$ m_{out} = 0 $$

$$ m_{in} = \Delta t \rho [\psi Q_{t+\Delta t} + (1 - \psi) Q_t] $$

$$ \dot{m}_{t+\Delta t} = \frac{\rho A (L_f + \ddot{x})}{\Delta t} $$
\[ m_t = \frac{\rho AL_f}{\Delta t} \] (32)

Plugging Eqs. (24)–(32) into Eqs. (22) and (23), the discrete form of the continuity and momentum equation for the front cell, Eqs. (5) and (6) can be easily obtained.

**Notation**

The following symbols are used in this paper:

- \( A \) = pipe cross-sectional area;
- \( a \) = elastic wave velocity;
- \( B_2, B_3, B_4, B_5 \) = dummy variables;
- \( C_1, C_2, C_3, C_4 \) = positive and negative characteristic line equations’ parameters;
- \( D \) = pipe diameter;
- \( F_r \) = friction force;
- \( F_g \) = gravity force;
- \( F_P \) = pressure force;
- \( F_V \) = generic variable;
- \( f \) = Darcy-Weisbach friction factor;
- \( g \) = gravitational acceleration;
- \( H \) = piezometric head;
- \( H_{\text{pout}} \) = piezometric head downstream side of cavity at point \( P_0 \);
- \( H_{\text{pot}} \) = piezometric head upstream side of cavity at point \( P_0 \);
- \( H_{\text{res}} \) = reservoir water height;
- \( H_L \) = gauge water pressure;
- \( J \) = pipe index;
- \( K_S \) = head loss coefficient accounting for friction loss in stagnant water column;
- \( K_v \) = valve head loss coefficient;
- \( L \) = pipe length;
- \( L_f \) = water column length in front cell in previous time line;
- \( L_s \) = initially stagnant water column length;
- \( m \) = liquid mass;
- \( m_{\text{in}} \) = mass rate entering front cell;
- \( m_{\text{out}} \) = mass rate leaving front cell;
- \( N \) = number of division of pipes;
- \( P_0 \) = reference pressure;
- \( Q_{\text{end}} \) = discharge released at the end of the pipe;
- \( Q_{\text{pout}} \) = discharge upstream side of cavity at point \( P_0 \);
- \( Q_{\text{pout}} \) = discharge downstream side of cavity at point \( P_0 \);
- \( S \) = energy slope;
- \( S_0 \) = pipeline slope;
- \( t \) = time;
- \( V \) = liquid velocity;
- \( V_m \) = liquid-water mixture velocity;
- \( x \) = distance;
- \( Z_{\text{end}} \) = pipe elevation at downstream side of last pipe;
- \( Z_{\text{pout}} \) = pipe elevation downstream side of the cavity at point \( P_0 \);
- \( Z_{\text{pout}} \) = pipe elevation upstream side of the cavity at point \( P_0 \);
- \( Z_{\text{pout}} \) = pipe elevation upstream side of the cavity at point \( P_1 \);
- \( \alpha_x \) = vapor void fraction;
- \( \alpha_0 \) = initial void fraction;
- \( \Delta x \) = reach length;
- \( \Delta t \) = time step;
- \( \Delta x \) = water column advancement;
- \( \theta \) = pipe angle respect to the horizontal;
- \( \rho \) = water density;
- \( \psi \) = time weighting factor;
- \( V_{\text{c}} \) = cavity volume; and
- \( V_R \) = computational cell volume.

**References**


