Developing a 1D-3D model to investigate the effect of entrapped air on pressure surge during the rapid filling of a pipe

R Maddahian¹, F Shaygan¹ and D M Bucur²

¹ Faculty of Mechanical Engineering, Tarbiat Modares University, Tehran, I.R. Iran
² Department of Hydraulics, Hydraulic Machinery and Environmental Engineering, University POLITEHNICA of Bucharest, 313, Splaiul Independentei, Bucharest, Romania

dmbucur@yahoo.com, diana.bucur@upb.ro

Abstract. An uncontrolled rapid filling or discharging of a pipeline can create compressed air columns in an urban water supply or sewage system. The expulsion of entrapped air out of pipelines is considered as one of the most destructive phenomena. Generally, the air valves are mounted and used to release the air from the pipe. Any failure of the air venting system may lead to a pressure surge inside the pipeline and even damage to the water systems infrastructure. Most of the investigations of multiphase flow during a rapid filling are limited to small distribution systems due to high computational costs. The aim of the present research is to develop a 1D-3D coupled model with the capability to handle multiphase flows. The effects of air pocket length, orifice size and upstream pressure on the flow field and maximum pressure surge inside the pipe are investigated. The classical water hammer equations are solved for the 1D domain using the Godunov scheme while the rest of the system is simulated using the Volume of Fluid (VOF) method. The coupling between 1D and 3D domains are handled using the developed boundary conditions based on flow characteristics. The results of the numerical simulations are compared with available experimental data. The obtained results show that the model can predict the features of the two-phase flow inside the pipeline during a rapid filling. The required computational time decreases by a factor of 4 compared to 3D simulations. This gives the possibility to investigate large scale water distribution or drainage systems.

1. Introduction

The pipeline systems such as urban water supply and sewage systems, water treatment plants and fuel transmission lines are designed to transport fluids. In the fluid transportation systems, air can enter the pipe by different ways, including the vortex flow caused by negative pressure in the suction side of a pump, the pipe filling and discharging, etc. The entrapped air should be evacuated through the air valves mounted on the pipe. Also, air may be entrapped inside the pipe due to the failure of the air venting system [1]. The entrapped air may create a transient pressure surge inside the pipeline system that could cause damages on the infrastructure of fluid transportation systems [2]. The entrapped air in the pipe creates a two-phase flow of air and liquid. The two-phase flow is affected by various factors such as air pocket size, upstream pressure, pipe diameter and air venting mechanism [3-5]. A similar phenomenon also occurs within the water distribution and urban wastewater systems. The transition
from open channel to pressurized flow due to the heavy rain can entrap different sizes of air pockets. The entrapped air pockets are compressed and move with the bulk flow. The compressed air pockets can cause Geysering phenomena when reaching to the street air vents [6, 7]. The air pockets inside the pipe can also have positive effects. The presence of the air pocket inside the flow compared to the single-phase flow of water increases the water elasticity and consequently reduces the wave velocity and pressure surges when the gas bubbles are distributed uniformly through the liquid [8]. Therefore, it is essential to analyze the parameters of the air pocket in order to predict more accurately its effects over the two-phase flow [9]. Many researchers investigated the rapid filling and discharging of the pipe and effective parameters of the pressure surge [10-13]. Bucur et al. [12] experimentally investigated the effective parameters such as air pocket size, the upstream pressure and the size of the outlet orifice on the maximum pressure surge during the rapid filling of a pipe. They showed that two different patterns are visible for the pressure wave. For the small orifice size, the air pocket compression is the predominant phenomenon while for the large orifice size, the water hammer phenomenon occurs. Recently, Tijsseling et al. [14] examined the rapid filling of a pipe with the entrapped air considering the oscillation of the upstream reservoir pressure. They obtained the liquid velocity as a function of column length and showed that the proposed model can be used for multiple gas pockets.

One-dimensional models including elastic water model [15] and rigid column [16] have been developed and used by several researchers to simulate the pipe flow with an air pocket. Lee [15] examined one-dimensional models and found that the 1D models were not efficient in simulating the pressure variation, especially when the pressure wave reaches the end of the pipe. Recently, Oscar et al. [17] employed a 1D rigid column model and the polytropic gas model to simulate discharging a pipe with the air pocket. The results showed that the applied mathematical model is capable of predicting the sub-atmospheric pressure. Although the 1D models can predict the pressure surges during the rapid filling or discharging, they are inappropriate to analyze the flow field and investigate the effective parameters on pressure surges. Therefore, researchers have used two and three-dimensional models to investigate the pressure during entrapped air expulsion [2, 5, 13]. Zhou et al. [5] employed the Volume Of Fluid (VOF) model to simulate the rapid filling of a pipe with an orifice at the end. The results showed that the VOF model could simulate pressure wave and air pocket motion. Li and Zhu [2] used a 3D model to simulate the two-phase flow of air and water during rapid pipe filling. They showed that the interface between air and water inclines due to gravity. The interface inclination during the filling process contradicts the assumption of the vertical interface in 1D models. Recently, Maddahian et al. [13] employed the 3D VOF model to investigate the process of filling the pipe and effective parameters on pressure surges. The results showed that the effect of Fluid-Structure Interaction (FSI) should be considered to capture the amplitude and the time interval between pressure surges. Although the 3D simulations give more details about the flow field, due to the computational costs the application of the 3D model is limited to the small-scale problems. To reduce the computational costs of large-scale problems, the 1D-3D modeling approach was developed [18, 19]. Wang et al. [18] coupled the 1D Method Of Characteristics (MOC) and the 3D finite volume model to simulate the compressible flow within the hydraulic systems. They investigated the water hammer phenomenon for a closed-end pipe and compared the results of the 1D-3D coupled model and 1D simulations. A good agreement was reported between models. Recently, Zhang et al. [19] employed the 1D-3D model to simulate the flow inside a centrifugal compressor. The suction pipe was modeled using a 1D model while the rest of the compressor was simulated using the 3D model. Still, the developed 1D-3D models are yet limited to the single-phase flows.

The aim of the present study is to develop a 1D-3D model with the capability of simulating the two-phase flow field and pressure surges during the rapid filling of an open-end pipe. The 1D model is developed based on the water hammer equations while the VOF model is used to simulate the two-phase flow. The pipe is divided into 1D and 3D sections. Simulations are performed simultaneously using the 1D and 3D codes. The simulation results are compared with available experimental data of Bucur et al. [12]. The governing equations are presented in section 2. The coupling approach between
1D and 3D regions is explained in section 3. Details of boundary conditions and geometry are presented in Section 4. Section 5 is devoted to the results from the simulation and discussions.

2. Numerical Method

2.1. 1D governing equations

To simulate the single-phase flow, the following 1D water hammer equations are used:

\[
\frac{\partial H}{\partial t} + V \frac{\partial H}{\partial x} + \frac{a^2}{g} \frac{\partial V}{\partial x} = 0
\]

(1)

\[
\frac{\partial V}{\partial t} + V \frac{\partial V}{\partial x} + g \frac{\partial H}{\partial x} + J = 0
\]

(2)

where \( x \) is the axial distance along the pipe, \( t \) - time; \( H \) - pressure (piezometric head), \( V \) - cross-sectional mean axial velocity, \( g \) - gravitational acceleration, \( a \) - wave speed and \( J \) - friction force on the pipe wall per-unit mass.

2.2. 3D governing equation

The continuity and momentum equations for a compressible two-phase flow are as follows:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0
\]

(3)

\[
\frac{\partial (\rho U)}{\partial t} + \nabla \cdot (\rho U U) - \nabla \cdot (\mu U U) = \sigma \kappa \nabla \alpha - g \cdot x \nabla \rho - \nabla p_d
\]

(4)

where \( \alpha \) is the volume fraction, \( \sigma \) - the surface tension, \( p_d \) - pressure and \( \kappa \) - the curvature of the interface and can be calculated as follows:

\[
\kappa = \nabla \cdot (\nabla \alpha / |\nabla \alpha|)
\]

(5)

The water volume fraction equation is employed to capture the interface between air and water and defined as follows

\[
\frac{\partial \alpha}{\partial t} + U \cdot \nabla \alpha + \nabla \cdot U \alpha (1-\alpha) = 0
\]

(6)

The last term in Eq. (6) is an anti-diffusion term used to sharpen the interface.

2.3. Discretization

The Finite Volume Method (FVM) is employed to discretize the governing equation in 1D and 3D domains. Equations (1) and (2) are discretized in the 1D domain using the Godunov FVM scheme. The matrix form of the equations is shown below:

\[
\frac{\partial \mathbf{u}}{\partial t} + \frac{\partial \mathbf{f}(\mathbf{u})}{\partial x} = \mathbf{S}
\]

(7)

where \( \mathbf{u} = \begin{pmatrix} H \\ V \end{pmatrix} \), \( \mathbf{A} = \begin{pmatrix} V & \frac{a^2}{g} \\ J & 0 \end{pmatrix} \), \( \mathbf{S} = \begin{pmatrix} 0 \\ J \end{pmatrix} \) and \( \mathbf{f}(\mathbf{u}) = \mathbf{A} \mathbf{u} \). Integrating Eq. (7) for a control volume from \( i - \frac{1}{2} \) to \( i + \frac{1}{2} \) results in the following equation:

\[
\frac{d \mathbf{u}}{dt} = \frac{f_{i+\frac{1}{2}} - f_{i-\frac{1}{2}}}{\Delta x} + \frac{1}{\Delta x} \int_{i-\frac{1}{2}}^{i+\frac{1}{2}} S \, dx
\]

(8)

where \( \mathbf{u} \) is the mean value of \( \mathbf{u} \) in the control volume. The fluxes at the control volume faces can be approximated using the Godunov scheme [20]. The exact solution of the Riemann problem is employed to approximate the fluxes [21]:

\[
f_{i+\frac{1}{2}} = \mathbf{A}_{i+\frac{1}{2}} \mathbf{u}_{i+\frac{1}{2}}
\]

(9)
where
\[
\begin{align*}
u_{i+1/2}^L &= \left( \frac{H_{i+1/2}^n}{V_{i+1/2}^n} \right) = \frac{1}{2} \left( \frac{H_L^n + H_R^n}{V_L^n + V_R^n} \right) + \frac{a}{g} \left( V_L^n - V_R^n \right) \\
u_{i+1/2}^R &= \left( \frac{H_{i+1/2}^n}{V_{i+1/2}^n} \right) = \frac{1}{2} \left( \frac{H_L^n + H_R^n}{V_L^n + V_R^n} \right) + \frac{a}{g} \left( H_L^n - H_R^n \right)
\end{align*}
\]

To evaluate \( u \) on the left and right of the interface, the first order Godunov scheme is used, given by [20]:
\[
\begin{align*}
u_L^0 &= u_{i-1}^0 \\
u_R^0 &= u_{i+1}^0
\end{align*}
\]

Equations (3) and (4) are discretized in the 3D domain using the second-order upwind and central schemes for convection and diffusion terms, respectively. The surface tension force is calculated using the Continuum Surface Force (CSF) method [22]. The MUlti-dimensional Limiter for Explicit Solution (MULES) is also employed to assure the boundedness and stability of the volume fraction equation.

3. 1D-3D coupling algorithm

In this section, a two-way coupling of 1D and 3D domains are explained. The velocity and pressure should be exchanged between domains during the simulations. The development procedure is explained for a boundary that the 1D domain is located on the left side of the 3D domain. The schematic of a boundary is shown in Fig. 1.

The Riemann invariants associated with the positive and negative characteristic lines are defined as follows:
\[
\begin{align*}
C^+ &\rightarrow \frac{a}{g} \left( V_{i+1/2}^n - V_L^n \right) + \left( H_{i+1/2}^n - H_L^n \right) = 0 \\
C^- &\rightarrow \frac{a}{g} \left( V_{i+1/2}^n - V_R^n \right) - \left( H_{i+1/2}^n - H_R^n \right) = 0
\end{align*}
\]

The positive invariant can be employed to calculate the boundary condition between domains in Fig. 1. The averaged face velocity of computational cells in the 3D domain is calculated and considered as \( V_{N_{i+1/2}} \) to estimate the pressure \( (H_{N_{i+1/2}}) \) at the boundary using Eq. (12). The boundary flux \( f_{N_{i+1/2}} \) can be calculated using the estimated pressure and velocity at the boundary.

The discretized equations in the 1D domain can be solved with the prescribed fluxes at boundaries. The solution procedure updates the internal value of velocity \( (V) \) and pressure \( (H) \). Using the updated internal values of pressure and velocity, the pressure at the boundary is calculated using Eq. (12) and considered as a pressure boundary condition for the 3D domain.

The solution of governing equations in the 3D domain updates the boundary face velocity which is transferred to the 1D domain as a boundary condition. Exchanging velocity and pressure between
domains continues until the convergence of the solution in both domains. The coupling algorithm is described in Fig. 2.

![Fig. 2 The coupling algorithm of 1D-3D](image)

It should be mentioned that for the 1D domain located on the right side of the 3D domain, the algorithm is not changed and the pressure can be calculated using Eq. (13). The coupling algorithm and the solution of the governing equation are implemented in the OpenFOAM framework.

4. Geometry and boundary conditions
The schematic of the test case considered in the numerical simulations is shown in Fig. 3. The upstream tank supplies water to an open pipe. The tank head \( P_R \) is constant. The pipe diameter \( D \) and length \( L_t \) are equal to 39 mm and 10.11 m, respectively. An orifice is mounted at the downstream and acts similar to the air venting valves. The orifice size \( d \) can be changed to control the amount of air venting (3, 5, 7 and 9 mm). Five butterfly valves are employed to control the initial water column length. The pressure measurements are performed using five pressure transducers mounted along the pipe. The measuring range of pressure transducers is 0-40 bar with an accuracy of 0.25% of the full scale. The upstream tank pressure varies from 2 to 4 bar. Further details can be found in Bucur et al. [12].

The pipe length is divided into 1D and 3D domains. The effect of interface location on the computational results is also checked. The interface between 1D and 3D domains are considered far enough to assure that the two-phase flow cannot reach the interface. Considering the interface in the single-phase domain is enough to ensure the independency of results from the interface location.
The uniform grid is used in the 1D domain while the non-uniform block-structured grid is employed in the 3D domain. The grid is fine enough to capture the regions with high gradients. The flow regime in the 3D domain and the pressure variations in the 1D domain are compared for different grids to check the grid independency of results. The final grid in the 1D domain has 4000 cells in the axial direction while for the 3D domain, the grid has 1050 and 35 cells in the axial and radial directions, respectively. It should be mentioned that the axisymmetric assumption is considered in the 3D domain to reduce computational costs. Previous investigations [5] showed that the assumption of axisymmetric flow can be employed for the straight pipes.

The upstream boundary condition is located in the 1D domain and considers a constant pressure value. The flux at constant pressure boundary can be calculated using the negative Riemann invariant. The details of the boundary condition can be found in [20].

The fixed pressure equal to the atmospheric pressure is considered for the downstream pressure located in the 3D domain. The no-slip boundary condition is implemented for the pipe wall. The effect of friction in the 1D domain is also handled using the Brunone et al. [23] model. It should be mentioned that the effects of upstream valves in the 1D domain are taken into account using the valves k-factor. The k-factors of butterfly valves are derived according to the valves’ catalog [24].

5. Results and discussion

5.1. Model validation

The results of the numerical 1D-3D model are compared with experimental data [12] in Fig. 4. It should be mentioned that the TP5 in numerical simulations is located in the 3D domain while the rest of the pressure measurements are placed in the 1D domain. Opening the valve creates a rarefaction wave which propagates in the water column and decreases the water column pressure. The axial pressure gradient in the water column and the pressure difference between the upstream tank and downstream atmospheric pressure accelerates the fluid element. The direction of the rarefaction wave is reversed when reaching the upstream tank.

The minor pressure fluctuations due to the propagation of the rarefaction wave in the water column before the pressure surge are captured by the numerical model. The amplitude of measured minor pressure fluctuations are less than the numerical ones. The transient friction factor of fluids affects the captured minor pressure fluctuations. Also, the effect of the upstream tank which is a boundary condition in the 1D domain on the water column movement and acceleration in the 3D domain is properly modeled. A pressure surge is generated in the water due to the impact of the water column on the downstream wall. The pressure surge obtained from the 1D-3D model agrees well with experimental measurements. Not only the amplitude of pressure surge but also the time of occurrence is predicted accurately by the developed model.
Figure 5 shows the water column movement in the 3D domain. Water and air are coloured in red and blue, respectively. It should be mentioned that the 1D domain only contains water and the interface between air and water located in the 3D domain. By moving the water column toward the downstream orifice, the air pocket is compressed. As shown in Fig. 5, the interface between air and water is not perpendicular to the pipe axis which is in correspondence with experimental measurements of Bucur et al. [12]. The part of entrapped air is discharged through the orifice while the rest is compressed and entrapped near the top wall of the pipe. The air entrapment can be seen at \( t = 0.22 \) s in Fig. 5. The results of the 1D-3D model are in good agreement with experimental data and previous 3D simulation [13].

5.2. Influence of the effective parameters
To check the applicability of the developed 1D-3D model, the effect of downstream orifice size on the pressure surge is investigated. The results of numerical simulation are compared with experimental measurements and shown in Fig. 6. The pressure surge due to the air entrapment is lower for the large orifice size in comparison to the small one. The resistance to the air vent is minimum for the orifice sizes \( d/D = 0.2307 \). As seen in Fig. 6, a critical orifice size \( (d/D = 0.128) \) exists which creates the
maximum pressure. The obtained results predict the critical orifice size which is in accordance with the previous investigation [12].

![Image](image1.png)

Fig. 5 Water column movement between 0 to 0.2 seconds after valve opening ($d/D = 0.128, P_R = 4, \alpha = 0.0544$)

![Image](image2.png)

Fig. 6 Effect of orifice size on pressure variations at TP5 ($P_R = 4, \alpha = 0.0544$)

The effect of upstream pressure on pressure surge is shown in Fig. 7. Reducing the upstream pressure decreases the pressure surge and the water column acceleration. The highest pressure surge is captured for the tank pressure $P_R = 4$. By reducing the upstream pressure, the water column acceleration and the pressure surge are decreased. The agreement between the numerical results of the 1D-3D model and experimental data is good. The developed model could accurately capture the water column acceleration and movement.

6. Conclusions
A 1D-3D algorithm was developed in the present research to investigate the entrapped air effects during the rapid filling of a pipe. The algorithm was based on calculating the Riemann invariant characteristics between 1D and 3D domains. The one-dimensional water hammer and the mixture approach of two-phase flow equations were solved in the 1D and 3D domains, respectively. To
calculate the Riemann invariants and connect the computational domains, the pressure and velocity were exchanged between domains.

The numerical results showed that the proposed algorithm could predict the pressure surge during the filling process. The influence of effective parameters including the orifice size and the upstream pressure was also investigated. The proposed model could be employed to investigate the rapid filling in large-scale water distribution systems. The only drawback of the model is that the domain interface should be located in the single-phase region. Considering the interface of domains in the two-phase flow region will be addressed in future works.

Fig. 7 Effect of upstream pressure on pressure variations at TP5 \((d/D = 0.128, \alpha = 0.0544)\)

**Acknowledgment**
The work of the Romanian part was supported by a grant of the Romanian Ministry of Education and Research, CCCDI - UEFISCDI, project number PN-III-P2-2.1-PED-2019-3247, within PNCDI III.

**REFERENCES**

[1] L. Ramezani, B. Karney, and A. Malekpour, "The challenge of air valves: A selective critical literature review," *Journal of Water Resources Planning and Management*, vol. 141, p. 04015017, 2015.

[2] L. Li and D. Z. Zhu, "Modulation of transient pressure by an air pocket in a horizontal pipe with an end orifice," *Water Science and Technology*, vol. 77, pp. 2528-2536, 2018.

[3] V. S. Fuertes-Miquel, O. E. Coronado-Hernández, P. L. Iglesias-Rey, and D. Mora-Meliá, "Transient phenomena during the emptying process of a single pipe with water–air interaction," *Journal of Hydraulic Research*, vol. 57, pp. 318-326, 2019.

[4] C. Lauchlan, M. Escarameia, R. May, R. Burrows, and C. Gahan, "Air in pipelines-A literature review," 2005.

[5] L. Zhou, D. Liu, B. Karney, and Q. Zhang, "Influence of entrapped air pockets on hydraulic transients in water pipelines," *Journal of Hydraulic Engineering*, vol. 137, pp. 1686-1692, 2011.

[6] C. D. Chosie, T. M. Hatcher, and J. G. Vasconcelos, "Experimental and numerical investigation on the motion of discrete air pockets in pressurized water flows," *Journal of Hydraulic Engineering*, vol. 140, p. 04014038, 2014.
[7] L. Zhou, T. Pan, H. Wang, D. Liu, and P. Wang, "Rapid air expulsion through an orifice in a vertical water pipe," *Journal of Hydraulic Research*, vol. 57, pp. 307-317, 2019.

[8] E. B. Wylie, V. L. Streeter, and L. Suo, *Fluid transients in systems* vol. 1: Prentice Hall Englewood Cliffs, NJ, 1993.

[9] S. C. Martins, H. M. Ramos, and A. B. Almeida, "Conceptual analogy for modelling entrapped air action in hydraulic systems," *Journal of Hydraulic Research*, vol. 53, pp. 678-686, 2015.

[10] J. K. da Silva, J. G. Vasconcelos, R. D. Maestri, and A. L. de Oliveira Borges, "Experimental investigation of air-water interactions during rapid filling process in an urban stormwater system model," in *World Environmental and Water Resources Congress 2018: Hydraulics and Waterways, Water Distribution Systems Analysis, and Smart Water*, 2018, pp. 222-232.

[11] J. G. Vasconcelos and S. J. Wright, "Geysering generated by large air pockets released through water-filled ventilation shafts," *Journal of hydraulic engineering*, vol. 137, pp. 543-555, 2011.

[12] D. M. Bucur, G. Dunca, and M. Cervantes, "Maximum pressure evaluation during expulsion of entrapped air from pressurized pipelines," *Journal of Applied Fluid Mechanics*, vol. 10, pp. 11-20, 2017.

[13] R. Maddahian, M. J. Cervantes, and D. M. Bucur, "Numerical investigation of entrapped air pockets on pressure surges and flow structure in a pipe," *Journal of Hydraulic Research*, pp. 1-13, 2019.

[14] A. S. Tijsseling, Q. Hou, and Z. Bozkış, "Rapid Liquid Filling of a Pipe With Venting Encrusted Gas: Analytical and Numerical Solutions," *Journal of Pressure Vessel Technology*, vol. 141, p. 041301, 2019.

[15] N. H. Lee, "Effect of pressurization and expulsion of entrapped air in pipelines," Georgia Institute of Technology, 2005.

[16] E. Cabrera, J. Abreu, R. Pérez, and A. Vela, "Influence of liquid length variation in hydraulic transients," *Journal of Hydraulic Engineering*, vol. 118, pp. 1639-1650, 1992.

[17] O. E. Coronado-Hernández, V. S. Fuertes-Miquel, M. Besharat, and H. M. Ramos, "Subatmospheric pressure in a water draining pipeline with an air pocket," *Urban Water Journal*, vol. 15, pp. 346-352, 2018.

[18] C. Wang, H. Nilsson, J. Yang, and O. Petit, "1D–3D coupling for hydraulic system transient simulations," *Computer Physics Communications*, vol. 210, pp. 1-9, 2017.

[19] M. Zhang, X. Zheng, Q. Huang, and Z. Sun, "A Novel 1D–3D Coupled Method to Predict Surge Boundary of Centrifugal Compressors," 2018.

[20] M. Zhao and M. S. Ghidaoui, "Godunov-type solutions for water hammer flows," *Journal of Hydraulic Engineering*, vol. 130, pp. 341-348, 2004.

[21] S. K. Godunov, "A difference scheme for numerical computation of discontinuous solutions of hydrodynamic equations.," *Math. Sbornik*, vol. 47, pp. 271–300., 1959.

[22] Z. Ma, D. Causon, L. Qian, C. Mingham, and P. M. Ferrer, "Numerical investigation of air enclosed wave impacts in a depressurised tank," *Ocean Engineering*, vol. 123, pp. 15-27, 2016.

[23] B. Brunone, U. Golia, and M. Greco, "Effects of two-dimensionality on pipe transients modeling," *Journal of Hydraulic Engineering*, vol. 121, pp. 906-912, 1995.

[24] O. AUTOMATION, "Butterfly Valve-ITEMS 375-376-377 & 385-386-387."