U.P.B. Sci. Bull., Series D, Vol. 78, Iss. 2, 2016 ISSN 1454-2358

MODELING TRANSIENT MULTIPHASE FLOW IN PIPELINE

Livia Ioana PITORAC[1], Diana Maria BUCUR[2], Georgiana DUNCA[3],

Michel J. CERVANTES[4]

Numerical simulations of an air-water transient flow in a pipeline, subjected to an instantaneous valve opening, are performed. Two cases are considered: a dead end and a venting system at the downstream part. An Euler-Euler model and a VOF model are used for analyzing the compressible multiphase flow. Results show that a small amount of entrapped air remains at the upper part of the pipe, influencing the maximum pressure. As the reverse flow occurs, air is absorbed back in the pipe through the orifice.

Keywords: multiphase flow, VOF, Euler-Euler, gas-venting system, transient

1. Introduction

Systems used by the industries may encounter problems when undesired fluids, as air, enter and interact with the main fluid. Entrapped air may have both beneficial and detrimental effects, by inducing a cushioning effect that changes the hydraulic parameters of the system [1, 2, 3, 4]. Furthermore, the presence of air usually leads to lower efficiency, or even damages.

Martin [1] showed that the peak in a multiphase flow with air at a close end is higher than in a single phase flow situation. In the case of entrapped air between a valve and a dead end, Ocasio [5] showed that there is a correlation between the length of the air column and the pressure peak; the shorter the air column is, the higher the pressure peak. In the case of a gas venting system, for an instantaneous valve opening, the velocity jets are higher and oscillations are removed if there is air in the system, [6]. Detrimental effects of an air-water multiphase flow in a gas-venting system are caused by water hammer effect, which, in the case of a small orifice or partially opened valves, leads to high pressure surge. Experiments on different ratios of orifice to pipe diameter showed that the pressure surge increases with decreasing the diameter of the orifice [3, 5, 7].

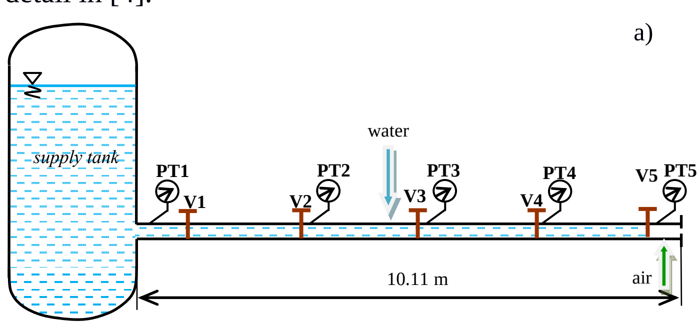
[1] MSc Student, Dept. of Hydraulic and Environmental Engineering, Norwegian University of Science and Technology, Norway, e-mail: liviaip@stud.ntnu.no / livia.pitorac@gmail.com
[2] Lect. PhD, Dept. of Hydraulics, Hydraulic Machinery and Environment Engineering, University POLITEHNICA of Bucharest, Romania, e-mail:dmbucur@yahoo.com (corresponding author)
[3] Lect. PhD, Dept. of Hydraulics, Hydraulic Machinery and Environment Engineering, University POLITEHNICA of Bucharest, Romania, e-mail:georgianadunca@yahoo.co.uk
[4] Prof., Division of Fluid and Experimental Mechanics, Luleå University of Technology, Sweden, Department of Energy and Process Engineering, Norwegian University of Science and Technology Norway, e-mail: michel.cervantes@ltu.se

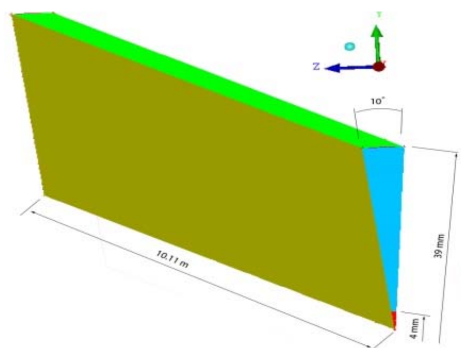
Research using one-dimensional modeling of such phenomenon needs assumptions requiring some investigation [2]. The accuracy of the results obtained from one-dimensional models depends on the friction coefficients and hypothesis such as an air-water interface perpendicular to the pipeline central axis at any moment and the flow is similar in any cross section. The first peak of the pressure-time is pretty well determined, but improvement regarding the timing and the overall pressure variation is lacking. A two dimensional model should lead to more accurate result. So far, few two-dimensional models have been developed.

In the present work, two-dimensional numerical models are developed for a multiphase flow with a vertical separation interface in a horizontal pipe. The two simulated cases consist of a pipe with either a dead end, or a venting system at the downstream end. In each case, different boundary conditions are assumed, results being analyzed and compared to experimental results for validation. The objective of this study is to model the maximum pressure in a pipe during the expulsion by water of entrapped air. Special attention is given to a hypothetical cushioning effect of the air. Two different approaches are used to model the flow: Euler-Euler approach and Volume of Fluid (VOF) approach, in order to evaluate the performance of the models.

2. Test case

Fig. 1.a. present a schematic of the test case consisting of a 10.11 m long pipe, with an internal diameter of 39 mm, made of Plexiglas, with an upper tank at the inlet, used to create the supply pressure. The pipe outlet can be completely closed or provided with an orifice. The pipe is made of 6 sections, separated by butterfly valves located at 0.96, 3.11, 5.27, 7.42, and 9.56 m from inlet (V1, V2, V3, V4, V5), to each corresponding a pressure transducer (PT) located downstream the valve, close to it. The pressure transducers are located downstream the supply tank as follows: PT1 at 0.16 m, PT2 - 3.20 m, PT3 - 5.35 m, PT4 - 7.50 m and PT5 - 9.65 m. The test set-up and measuring procedures are described in detail in [4].





b)

Fig. 1. Experimental set-up and model.
a) schematic of the test case, b) computational domain

3. Numerical model

Fig. 1.b. presents the computational domain; a 2D version of the simplified experimental test rig. The following assumptions were done:

- inlet of the domain has the same diameter as the pipe
- inlet pressure is constant during the entire simulation
- valves V1, V2, V3 and V4 are not included in the model;
- separation valve V5 is reduced to a wall, which instantly disappears (0 s);
- outlet is either a closed end, or an orifice.

The geometry and the mesh were created using ANSYS ICEM CFD. The Computational Fluid Dynamics (CFD) software ANSYS-CFX was used. From a mathematical point of view, the Navier-Stokes equations (momentum, heat and mass transfer) are solved. Other physical processes, as turbulence or compressibility, that encounter in the simulated phenomenon can be solved in conjugation with the Navier-Stokes equations. In ANSYS CFX different models are implemented, as turbulence, heat transfer and radiation, multiphase flow, chemical reactions and combustion, rotating machinery, etc.

Two multiphase flow models are included in the software, a homogeneous model, which correspond to the VOF model, and an inhomogeneous model, which corresponds to the Euler-Euler approach.

In the Eulerian concept, the flow is described by setting a fixed point in space (p), and analyzing the evolution of the flow properties in time in that specific point. Therefore, the fluid's properties can be described as: ρ_p (, t), v_p (, t), p_p (, t), etc. where ρ is density, v is velocity and p is pressure.

The model consists of two continuum phases mathematically modeled as interpenetrating continua, appropriate for separated flows. A set of conservation equations is solved for each phase, using interphase exchange coefficients and shared pressure for coupling the phases. More information can be found in [8].

The VOF model is used when the interface interests more than the overall flow. The model analyzes the flow by tracking the interface between the two phases, being suitable for immiscible fluids. For this approach a single set of momentum equations is solved, with a volume fraction equation for each fluid [9].

Compressibility effects are significant in the present setup and are modeled in the simulations. The variation of the density is mostly dependent on the pressure, *p*, and the temperature, *T*. The field of compressible flows is part of the fluid dynamics field. Thus, the compressibility effect is determined as:

$$\rho_W = \rho_0 * \left(1 + \frac{p - p_0}{k_f}\right) \quad (1)$$

where the subscript 0 stands for the property of the material in an incompressible case, at atmospheric pressure and 25˚C, and k_f is the bulk modulus.

4. Results

This section presents the results of two numerical simulations: one performed with a dead end and one with a 4 mm outlet orifice. The simulations with the orifice are performed with the VOF and Euler-Euler model. A comparison to experimental results is also done.

All the simulations presented are transient and multiphase in order to model the instantaneous valve opening and the air and water flow. Both fluids are modeled as compressible and the SST turbulence model is used to resolve the flow behavior up to $y^+<1$ in the boundary layer. The pipe is considered to have an open inlet. The separation valve between the two fluids is located in different places for the two cases. As initial condition, the upstream part of the pipe is filled with water at 3 bar, and the downstream part contains air at atmospheric pressure. The closed valve and the two fluids with corresponding pressures are modeled using a step function. The valve is modeled as a wall, which instantly disappears at t = 0 s. All walls are modeled as smooth no slip walls. The simulation period is 1 and 5 s for the orifice simulation and the dead end, respectively. The time step is 0.0001 s for all performed simulations. The absolute pressure is monitored in 5 points during the simulations, corresponding to the five pressure transducers from the experimental study [4].

Multiphase flow for a pipe with dead end

The pipe is considered to have an open inlet and a closed downstream end. The water column length is 7.43 m, and the air water column 2.68 m.

Figure 2 presents the pressure variation after opening the valve. The valve is opened instantaneous at t = 0 s. The pressure upstream the valve is 3 bar, while the pressure downstream the valve is atmospheric. The monitor points located in the upstream part show a 1 bar pressure drop tendency before the pressure starts to increase. Similar oscillations to the ones in the open end pipe case can be observed at the beginning of the transient regime. The five monitor points, starting with the downstream one, showed pressures of 8.3, 7.1, 6, 4.8 and 3.1 bar, respectively, for the first peak. The pressure oscillation period for the five registered peaks is approximately 0.55 s. When the water hits the downstream wall, a water hammer effect is produced, generating a reverse flow which results into a pressure drop. The first pressure drop is damped by 1 bar at the closest downstream monitor point (PT5). The entire phenomenon repeats with lower amplitudes, the pressure getting more and more amortized, while the frequency is constant.

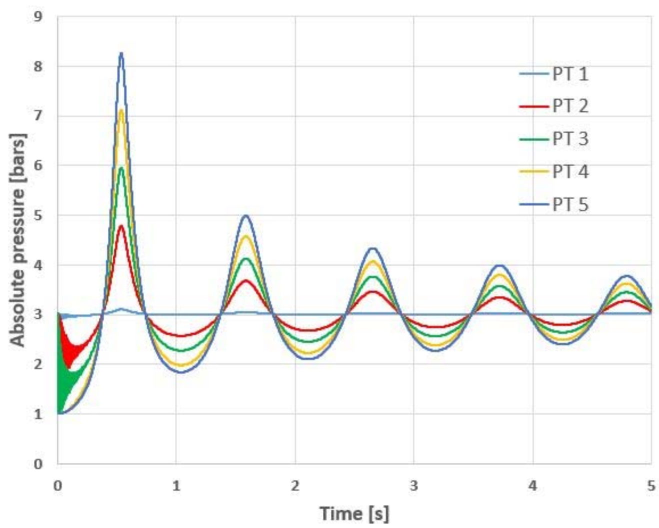


Fig. 2. Pressure variation in time for closed outlet

Multiphase flow for a pipe with a 4 mm orifice at the outlet

In this case the presented water column length is 9.56 m, and the air water column 0.55 m. As in the previous case the pipe is considered to have an open inlet with a constant supply pressure of 3 bar, but a outlet orifice of 4 mm located along the pipe’s axis at the downstream part.

Fig. 3 presents the pressure variation with time determined in 5 points along the pipe, corresponding to the location of the pressure measurement sections. Three pressure peaks can be distinguished at 0.25, 0.50 and 0.64 s. The highest amplitudes are in the first peak and have a value of 42, 40, 37, 29 and 5 bar from downstream to upstream, respectively.

At t = 0 s, when the wall disappears, an instantaneous pressure drop can be observed for the pressure points located upstream the interface. The beginning of the first pressure peak encounters when the water gets to the end of the pipe. Thus, at t = 0.23 s, an approximately 3 bar pressure is registered in all the monitored points. The value of the first pressure peak is 14 times higher than the inlet pressure at the section closest to the orifice, being reached due to water inertia. Oscillations similar to water hammer behavior appear in the flow caused by the sudden narrowing at the outlet section.

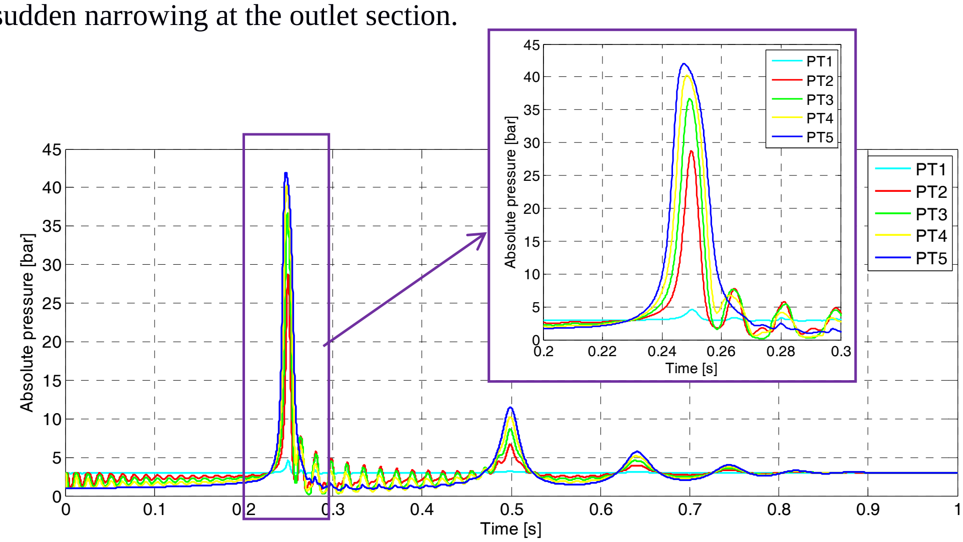


Fig. 3. Pressure variation in time for 3 bar inlet pressure with a 4 mm orifice

Figs. 4, 5 and 6 present the interface between the two fluids and the velocity vectors. Fig. 4 corresponds to the pressure increasing until the first pressure peak. The accelerating water column causes the air to be expelled, but due to the large velocity of the water-air interface, the critical velocity for air expulsion is reached at the outlet at t = 0.23 s. Not all the air leaves the system; some air remains trapped when the water reaches the outlet and covers it completely. Due to the presence of the diaphragm causing a narrowing at the outlet section, the water velocity decreases, until it becomes 0 m/s at t = 0.25 s, corresponding to the first pressure peak.

After the system reaches the maximum pressure, a reversed flow occurs. The complete phenomenon is shown in Fig. 5. At t = 0.26 s the water velocity is close to 0 m/s, continuing with a reversed flow similar to a water hammer effect at t = 0.27 s. The reversed flow is followed by an air venting phenomenon, which occurs at t = 0.28 s. A temporary equilibrium is reached at t = 0.30 s, when the minimum pressure is achieved. At this moment the air continues to vent, while the water column changes the flow direction for the second time, the entire phenomenon being repeated with lower amplitudes (Fig. 6).

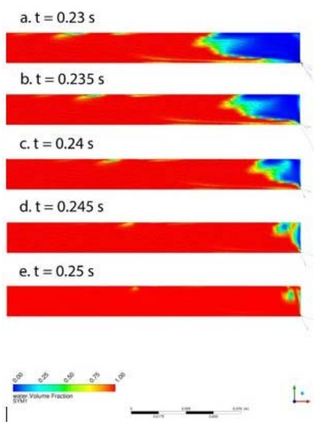


Fig. 4. Volume fraction and velocity vectors for the flow until the first pressure peak

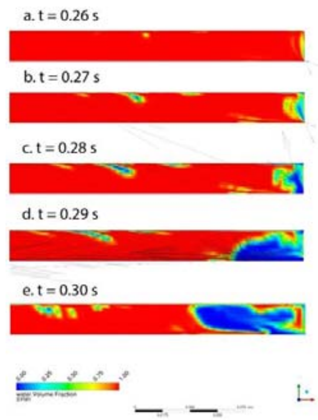


Fig. 5. Volume fraction and velocity vectors for the first reversed flow

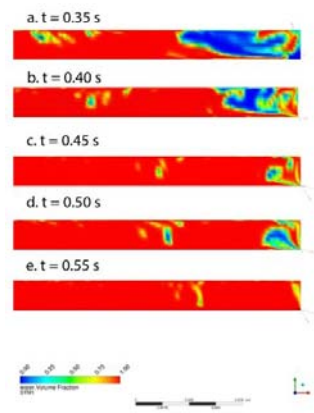


Fig. 6. Volume fraction and flow velocity vectors during the second pressure peak

Euler-Euler vs. VOF and comparison to experiments

Fig. 7 presents the results measured and computed with both approaches at PT5 located in the air domain before opening. The first peak has a similar amplitude for both models, with just a 0.5 bar difference between them. For the second peak the pressure difference is 4 bar, with a pressure of 17.5 bar for the Euler-Euler model, comparing to 13.5 bar for the VOF model. If looking at the time until the second peak, a delay of 0.02 s is registered between the two models. When comparing the time approximation for the two models with the experimental data, a delay of 0.30 s is observed for the first peak. A very important influence over this time difference is due to the valve opening time of 0.02 s in the experimental, which is reduced to 0 seconds in the numerical simulations.

The pressure peak from the numerical results compared to the experimental ones, gives a rough approximation. One reason for this might be the local pressure losses introduced by the valves in the real set-up, which are not considered in the simulations.

The pressure amplitude and time from Fig 7 are presented in table 1, together with the experimental measurements. For each model a ratio is determined, and compared to the ratio for the experimental data. The pressure ratio is determined as the pressure value of the second peak to the pressure value of the first peak. The ratio between the absolute pressures is 0.39 for Euler-Euler approach, 0.27 for VOF model, and 0.35 for the experimental data. Table 2 presents the time intervals from the opening of the valve, t_0, until the occurrence of first pressure peak, t_{P1} and of the second peak, t_{P2}, together with their ratio.

Table 1

Comparison of maximum pressure at PT5

Method	Absolute pressure [bar]			Relative pressure [bar]		
	P_{a1}	P_{a2}	P_{a2}/P_{a1}	P_{r1}	P_{r2}	P_{r2}/P_{r1}
Euler-Euler	45	17.5	0.39	44	16.5	0.38
VOF	42	11.5	0.27	41	10.5	0.26
Experimental	17	6	0.35	16	5	0.31

Table 2

Time until maximum pressure occurance at PT5

Method	Time [s]			Time [s]		
	t_0-t_{P1}	t_0-t_{P2}	(t_0-t_{P1})/(t_0-t_{P2})	t_{p1}-t_{P2}	t_0-t_{P1}	(t_{p1}-t_{P2})/(t_0-t_{P1})
Euler-Euler	0.25	0.46	0.54	0.21	0.25	0.84
VOF	0.25	0.50	0.50	0.25	0.25	1.00
Experimental	0.28	0.51	0.55	0.23	0.28	0.82

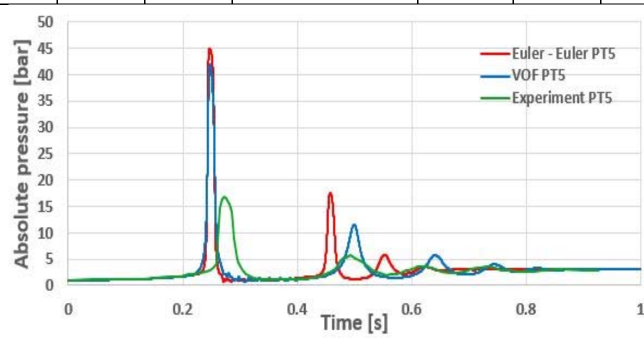


Fig. 7. Comparison of pressure obtained with VOF, Euler-Euler and experimental at PT5, for 3 bar inlet pressure and 4 mm orifice

A similar comparison is presented in Fig. 8, and tables 3, respectively 4, for the pressure in the PT4 section, situated 2.15 m upstream PT5. The pressure time variation has the same tendency as the one in the previous case.

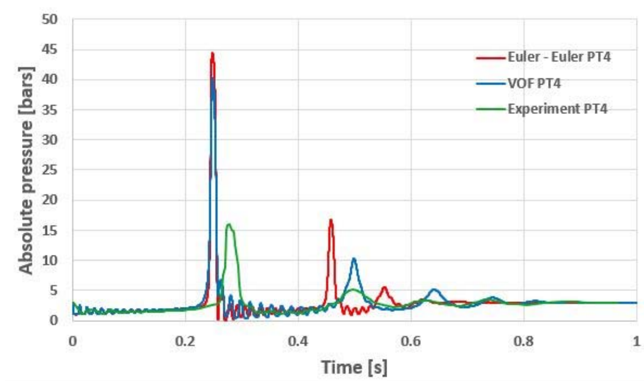


Fig. 8. Comparison of pressure obtained with VOF, Euler-Euler and experimental at PT4, for 3 bar inlet pressure and 4 mm orifice

Table 3

Maximum pressure comparison at PT4

Method	Absolute pressure [bar]			Relative pressure [bar]		
	P_{a1}	P_{a2}	P_{a2}/P_{a1}	P_{r2}	P_{r1}	P_{r2}/P_{r1}
Euler-Euler	44.3	16.7	0.38	43.3	15.7	0.36
VOF	40.2	10.3	0.26	39.2	9.3	0.24
Experimental	17	6	0.35	16	5	0.31

Table 4

Time until maximum pressure at PT4

Method	Time [s]			Time [s]		
	t_0-t_{P1}	t_0-t_{P2}	$(t_0$-$t_{P1})/(t_0$-$t_{P2})$	t_{p1}-t_{P2}	t_0-t_{P1}	$(t_{p1}$-$t_{P2})/(t_0$-$t_{P1})$
Euler-Euler	0.25	0.46	0.54	0.21	0.25	0.84
VOF	0.25	0.50	0.50	0.25	0.25	1.00
Experimental	0.28	0.51	0.55	0.23	0.28	0.82

5. Conclusions

The purpose of this paper is to investigate an air-water transient flow in a horizontal pipe subjected to an instantaneous opening of a valve, with either a dead end or an orifice.

Two numerical models were developed, an Euler-Euler model and a Volume of Fluid model, and simulation results were compared to experimental [4] for validation. Parametric and comparative studies were investigated in order to determine the effect of using a homogeneous multiphase model for the studied cases. Following are the major findings:

– Both numerical models have captured the correct trend of the phenomenon, with good time approximation.
– A small amount of entrapped air remains at the upper part of the pipe, when the orifice is completely covered with water
– If the main interest of the phenomenon is the prediction of the first peak, both numerical models give similar results. However, due to computational reasons, the VOF model is preferred, this solving just one set of equations, together with a volume fraction equation.
– The Euler-Euler model is recommended if the entire trend is of interest, being capable of a better prediction of the phenomenon that encounters after the first pressure peak is reached.
– Both models in the case of open end pipe showed that during the expansion of the air, pressure decreases under atmospheric pressure level at the outlet section, this resulting in absorption of air in the pipe through the orifice. This is also verified by the direction of the velocity vectors at the outlet section during the pressure drop.

The general conclusion is that a multiphase flow simulation involves a very large number of parameters like compressibility, turbulence, phase interaction, etc., due to the complex interaction between the phases, so that further research is needed. Also, it is of interest to include in the model geometry the upstream valves and to investigate their influence over the maximum pressure.

Acknowledgment

The work has been funded by the Sectoral Operational Programme Human Resources Development 2007-2013 of the Ministry of European Funds through the Financial Agreement POSDRU/159/1.5/S/134398 and by the Executive Agency for Higher Education, Research, Development and Innovation, PN-II-PT-PCCA-2013-4-0814, ECOTURB project.

R E F E R E N C E S

[1] *C. Martin*, "Entrapped air in pipelines," in Proceeding of the Second International Conference on Pressure Surge, Bedford, England, London, 1976.

[2] *N. Lee*, Effect of Pressurization and Expulsion of Entrapped Air in Pipelines, School of Civil and Environmental Engineering, Georgia Institut of Technology, Georgia, USA, 2005.

[3] *K. Hashimoto, M. Imaeda and A. Osayama*, "Transients of Fluid Lines Containing an Air Pocket or Liquid Column," Journal of Fluid Control, **vol. 18**, no. 4, pp. 38-54, 1988.

[4] *D.M. Bucur*, Contributii privind curgerea bifazica in conductele centralelor hidroelectrice si ale statiilor de pompare (Contributions fo the Two-Phase Flow Study in the Pipelines of Hydropower Plants and Pumping stations) Politehnica University of Bucharest, Bucharest, Romania, 2011.

[5] *J. Ocasio*, "Pressure surging associated with pressurization of pipelines containing entrapped air," School of Civil Engineering, Georgia Institut of Technology, Georgia, USA, 1976.

[6] *D. Edwards and G. Farmer*, "A Study of Piston-Water Impact in an Impulsive Water Cannon," in Seventh International Conference on Jet Cutting Technology, Ottawa, Canada, 1984.

[7] *J. Agudelo*, Entrapped Air in Water Pipelines, Georgia, USA: School of Civil Engineering, Georgia Institute of Technology, 1988.

[8] *E. Steinmark*, On Multiphase Flow Models in ANSYS CFD Software, Chalmers University of Technology, Goteborg, Sweden, 2013.

[9] ANSYS Theory Guide.