Backflow Air and Pressure Analysis in Emptying Pipeline Containing Entrapped Air Pocket

Mohsen Besharat [a]*, Oscar E. Coronado-Hernández[b,c], Vicente S. Fuertes-Miquel[c], Maria Teresa Viseu[d], Helena M. Ramos[a]

[a]*Department of Civil Engineering, Architecture and Georesources, CERIS, Instituto Superior Técnico, University of Lisbon, Av. Rovisco Pais, 1, 1049-001, Lisbon, Portugal; mohsen.besharat@tecnico.ulisboa.pt (M.B.); helena.ramos@tecnico.ulisboa.pt (H.M.R.)*

[b]*Facultad de Ingeniería, Universidad Tecnológica de Bolívar, Parque Industrial y Tecnológico Carlos Vélez Pombo Km 1 Vía Turbaco, 131001, Cartagena, Colombia; ocoronado@unitecnologica.edu.co*

[c]*Departamento de Ingenieríıa Hidráulica y Medio Ambiente, Universitat Politècnica de València, Camino de Vera S/N CP 46022, Valencia, Spain;*
vfuertes@upv.es (V.S.F-M.);

[d]*Laboratório Nacional de Engenharia Civil (LNEC), Av. do Brasil, 101, 1700-066, Lisbon, Portugal; tviseu@lnec.pt*

*Corresponding author:

Email: mohsen.besharat@tecnico.ulisboa.pt; Tel: 00351 218 418 155

Backflow Air and Pressure Analysis in Emptying Pipeline Containing Entrapped Air Pocket

The prediction of the pressure inside the air pocket in water pipelines has been the topic for a lot of research works. Several aspects in this field have been discussed, such as the filling and the emptying procedures. The emptying process can affect the safety and the efficiency of water systems. Current research presents an analysis of the emptying process using experimental and computational results. The phenomenon is simulated using the two-dimensional computational fluid dynamics (2D CFD) and the one-dimensional mathematical (1D) models. A backflow air analysis also is provided based on CFD simulations. The developed models show good ability in the prediction of the sub-atmospheric pressure and the flow velocity in the system. In most of the cases, the 1D and 2D CFD models show similar performance in the prediction of the pressure and the velocity results. The backflow air development can be accurately explained using the CFD model.

Keywords: Emptying process; Transient two-phase flow; Backflow air

Introduction

The air pocket presence in pipe systems is a practical problem due to the lack of design and operation knowledge or equipment malfunction. For those reasons, designers frequently neglect the air pocket existence that may lead to harmful consequences (Besharat et al. 2016a). This application occurs in the majority of cases in water distribution networks, firefighting systems, fluid-transport pipelines, storm-water, drainage and sewage systems, tunnels, syphons and urban water networks (Pozos et al. 2010; Laanearu et al. 2012). The air can appear in pipelines due to pressure drop or insufficient submergence in the water intakes. The filling of pipe systems can also be another source of the air accumulation. During this process, if the air is not removed completely, a volume of air will accumulate at downstream. The entrapped air in a pipeline can cause different problems including sub-atmospheric pressure appearance, disruption of the flow regime, reduction of the pump and turbine efficiency, fatigue or rupture of pipe materials and pipeline structures, changes in fluid properties, faults in instrumentation readings and environmental contamination (Besharat et al. 2016a; Ramezani et al. 2016; Escarameia 2007; Laanearu et al. 2015). Bowker et al. (1992) reported the influence of the air over the material of a pipe due to the erosion effect. The effect of an air pocket on the head loss was studied by Escarameia (2007) showing a loss of 35% for the analysed system which induces higher pump load leading to higher electricity consumption and depreciation. An air pocket decreases the effective cross-section of a pipe and the carrying capacity called air binding that may cause unexpected pressure surges (Richards 1962; Edmunds 1979; Besharat et al. 2017). Some aspects of the entrapped air in water filling pipelines have been studied using CFD models such as studying the filling process in a single pipe using 3D CFD simulation (Martins et al. 2017), using the 2D and 3D CFD models for rapid filling cases (Zhou et al. 2011), and understanding the dynamic behaviour of trapped air pocket in filling case using a 3D CFD model have been studied before (Zhou et al. 2018).

An emptying process with no admitted air at upstream is subject to a slow down-surge wave which may create a very low sub-atmospheric pressure value. The sub-atmospheric pressure in the pipe system can cause a suction effect in the defective joints or valves leading the entrance of the air into the pipe system (Wisner et al. 1975) or, eventually, inducing the buckling or crushing of the pipe (Coronado-Hernández et al. 2017). A down-surge wave situation can be controlled by means of different methods such as air vessel, surge tank, bypass, in-line polymeric short-section and the air valve (Triki, 2017). However, a common solution for removing air and/or to avoid the sub-atmospheric pressure occurrence is installing air valves in high points of the pipe profile. An air valve conveys air at the atmospheric pressure into the pipeline or reliefs the accumulated air to regulate the pressure. Understanding the dynamic behaviour of the air and the backflow air phenomenon will facilitate the selection and design of a proper controlling method.

There are few studies about the emptying developments focusing on the complexity of the phenomenon in practical applications with irregular profiles (Laanearu et al. 2012; Tijsseling et al. 2016). The complex nature of the air-water tail, the movement of the air pocket and the air intrusion and oscillation are some topics that previously have been studied both experimentally and numerically by other researchers in emptying of simple systems (Zukoski 1966; Benjamin 1968; Vasconcelos and Wright 2008; Laanearu et al. 2015). The current study took advantage of a well-equipped experimental apparatus to measure the pressure in an air pocket and the flow velocity in the pipe system. The study focused on the two-phase emptying process with various air pocket sizes and valve actuation situations. It is essential to have the ability to accurately calculate the low-pressure conditions and understand the relation between backflow air and pressure changes for safety reasons and design rules definition. Hence, numerical simulations using the one-dimensional mathematical (1D) and the two-dimensional computational fluid dynamic (2D CFD) models were developed to show the ability of these methods for calculating the major parameters.

In general, the main objectives of the current research are to understand the dynamic behaviour of the air pocket during the emptying process, to examine the accuracy of provided models in prediction of major parameters and also to understand the backflow air effect on the emptying process. It is observed that the water columns pressurized initially but during the emptying process, a free surface flow is created with a distinct air-water interface. A stratified air-water tail is determined similar to the previous study developed by Laanearu et al. (2012).

Methodology

An undulating experimental apparatus made of transparent PVC pipes with a nominal diameter of 63 mm and equipped with two ball valves located at the downstream of the pipes as shown in Figure 1 was used to examine the drainage process. To test the two-phase emptying condition, a confined air pocket was located at the highest level of the profile before starting of each test. The water column is under static pressure initially. The drainage takes place by the partial and total opening of the ball valves simultaneously creating a down-surge wave which propagates towards the pipe ends. The ball valves actuation was carried out manually. As shown in Table 1, the length of air pockets in left and right pipe branches can be either equal or unequal. Different tests were carried out by changing the air pocket size and the valve opening situation namely partial or total opening as in Table 1. The valve is opened by 6% in the partial opening tests. The volume fraction ratio (VFR) for each test is provided in Table 1 as the volume of air over the volume of the pipeline. The measurements of pressure in the highest point of the pipeline profile and the flow velocity in horizontal pipes were carried out using a pressure transducer and an ultrasonic doppler velocimetry (UDV). The installed pressure transducer measures the absolute pressure up to 25 bar. The maximum measurement error has been reported 0.5% by the manufacturer. The output signal from the pressure transducers was an analogue signal with an electric current between 4 to 20 mA. After amplifying the signals, they were collected by an electronic oscilloscope to process the pressure signals, convert them to digital signals, remove the noises and finally record them on the computer. Also, the UDV transmits a short emission of ultrasound, which travels along the pipe at an angle of 20^{o} with the vertical axis. All the velocity measurements took place in the specific points at the downstream of the pipeline as shown in Figure 1. Measurements of each test carried out twice for more confidence. For experiments with suspicious data, more tests were carried out to verify the measurements. An average experimental data was calculated based on all the measurements and used for comparison with numerical data. Calculation of experimental data was done by a 1D mathematical model and a 2D CFD model.

1D Model

The one-dimensional model developed by Coronado-Hernández et al. (2018) for undulating profiles is used to simulate the emptying process. The mathematical model is based on a rigid water column formulation (RWCF) taking a moving air-water interface using a polytropic expression to simulate an entrapped air pocket (León et al. 2010; Martins et al. 2015). The RWCF considers a higher elasticity of entrapped air than the elasticity of both water and pipe. Zhou et al. (2013), Zhou et al. (2002) and Izquierdo et al. (1999) demonstrated that elastic water hammer formulations bring similar solutions compared to RWCF when an air pocket is trapped. Neglecting water and pipe elasticity, the momentum equation is expressed as:

$$\frac{dv_w}{dt} = \frac{p_a^* - p_{atm}^*}{\rho_w L_w} + g\frac{\Delta z}{L_w} - f\frac{v_w |v_w|}{2D} - \frac{R_v g A^2 Q_{T,w} |Q_{T,w}|}{L_w} \tag{1}$$

in which v_w is the water velocity, *t* is the temporal coordinate, p_a^* is the air pocket absolute pressure, p_{atm}^* is the atmospheric pressure, ρ_w is the water density, L_w is the length of the water column, *g* is the gravity acceleration, Δz is the difference elevation of the water column ends, *f* is the Darcy-Weisbach friction coefficient, *D* is the internal pipe diameter, R_v is the resistance coefficient of the drain valve, *A* is the cross-sectional area and $Q_{T,w}$ is the total water flow to be drained by a drain valve.

A piston flow model was considered to simulate a moving air-water interface.

$$\frac{dL_w}{dt} = -v_w \tag{2}$$

A polytropic expression is used to simulate the air pocket behaviour using the polytropic equation, i.e., $p_a^* \forall_a^n = cte$, which relates the air pocket pressure with the air volume ($\forall_a^n$). The polytropic exponent or polytropic index (n) can vary from 1 for the isothermal process to 1.4 for adiabatic process depending on temperature change and heat transfer (Besharat and Ramos 2015). An algebraic-differential system (ADS) describes the entire process, which is composed by the momentum equation, a moving interface air-water position, and the polytropic model. The resolution of the ADS gives the information of the hydraulic variables (air pocket pressure, water velocity, and length of the water column). To solve the ADS a constant friction coefficient was considered (f=0.018) with a non-variable polytropic coefficient of n=1.1. The minor loss coefficients were calibrated based on the experiments. The Simulink tool in Matlab was used to solve the algebraic-differential equations system.

2D CFD Model

The accuracy and more comprehensive results of CFD simulations attract researchers to use them rather than 1D models. The selection of appropriate simulation models, such as a pertinent turbulence model, will increase the efficiency and accuracy of CFD simulations. Turbulence models may be categorized as low-Reynolds and high-Reynolds models (Cebeci, 2004). The low-Reynolds models aim to identify laminar sub-layers needing very low-height cells near the wall and high calculation load. For that reason, the high-Reynolds models are commonly used. There are many high-Reynolds turbulence models, ranging from one-equation models to robust two-equation models. As a two-equation model, the k-ε model is used frequently due to good performance, acceptable accuracy and low computational time. This model uses two transport equations for turbulence kinetic energy (k) and its dissipation rate (ε). The transport equation of k is derived from exact formulations, while the dissipation rate is derived from an empirical formulation (Launder and Spalding, 1972; Cebeci, 2004) as:

$$\rho\frac{\partial k}{\partial t}+\rho\bar{u}_j\frac{\partial k}{\partial x_j}=\tau_{ij}\frac{\partial \bar{u}_i}{\partial x_j}-\rho\varepsilon+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] \tag{3}$$

$$\rho\frac{\partial \varepsilon}{\partial t}+\rho\bar{u}_j\frac{\partial \varepsilon}{\partial x_j}=C_{\varepsilon_1}\frac{\varepsilon}{k}\tau_{ij}\frac{\partial \bar{u}_i}{\partial x_j}-C_{\varepsilon_2}\frac{\varepsilon^2}{k}+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] \tag{4}$$

in which μ_t is the turbulent viscosity given by:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \tag{5}$$

Rest of parameters are constants defined from experimental results as $C_\mu=0.09$, $C_{\varepsilon_1}=1.44$, $C_{\varepsilon_2}=1.92$, $\sigma_k=1.00$ and $\sigma_\varepsilon=1.30$ (Wilcox, 2006). However, implementation of the two-equation turbulence models may lead to unsuitable results near the wall. This flaw comes from the inability of the model in the calculation of constant, B accurately in the law of the wall equation, i.e., $u=u_*\left[\ln(u_*y\;/\;\upsilon)\,/\,\kappa+B\right]$ where κ and B are constants (Besharat et al. 2016b). To enhance the accuracy, wall functions or enhanced wall treatment (EWT) method may be used. The EWT method uses a two-layer approach which divides the near wall domain into two layers namely viscous sub-layer and turbulent sub-layer for calculating the ε and the μ_t in the near wall domain. The EWT method combines linear and the logarithmic law of the wall by a function from Kader (1993) which uses a blending function as $\Gamma=-0.01(y^+)^4/(1+5y^+)$ to achieve a single wall law for entire near wall domain in the momentum and energy equations as shown in Equation 6 (ANSYS FLUENT R19.0).

$$u^+=e^{\Gamma}\,u^+_{lam}+e^{1/\Gamma}\,u^+_{tur} \tag{6}$$

The ANSYS Fluent R19.0 academic was used as the CFD simulation tool. A transient pressure-based solver resolved the coupled equations. This solver is able to handle the coupled equations between pressure, velocity and density in the compressible flows (Wang et al. 2016). The volume of fluid (VOF) multiphase model with the explicit formulation was implemented for discretization. The VOF model for determining the deformations of the interface is used extensively due to simplicity and low required computational load (Besharat et al. 2016b). Two components of air and water were considered in this simulation as separate fluids in the two-phase simulation.

In order to simulate the valve actuation in the CFD model, a moving cell zone condition was defined using the sliding mesh method and a user-defined function (UDF) with the desired angular velocity of each test. All boundary conditions except the outlet ends of the system are non-slip wall boundary conditions. For the two outlet ends, the pressure outlet boundary was defined. The CFD model takes advantage of the pressure implicit with the splitting of operators (PISO) method for coupling the pressure-velocity formulations. The spatial discretization was carried out using a finite volume approach. The pressure staggering option (PRESTO) discretization scheme is used for the pressure, while for other spatial discretization a second order upwind scheme is adopted. For the interface tracking method, the compressive scheme was used. This scheme is a second order reconstruction scheme based on a slope limiter value (ANSYS FLUENT R19.0).

$$\phi_f = \phi_d + \beta \nabla \phi_d \tag{7}$$

where the ϕ_f is the face VOF value, ϕ_d is the donor cell VOF value, β is the slope limiter value between 0 and 2 and $\nabla \phi_d$ is the donor cell VOF gradient value.

All the mentioned mesh information was selected based on the previous research and mesh independent analysis in a similar system (Besharat et al. 2016b). The mesh for the CFD model consists of 108,919 cells in all the simulations having the minimum and maximum face areas of 0.0001m^2 and 0.006 m^2, respectively. A triangular unstructured mesh was considered for the main body of the model with 10 rectangular inflation layers at the near-wall zone. The simulation was executed on a desktop computer: Intel(R) Core(TM) i7-4790 CPU @ 3.60GHz with an installed memory of 16 GB with a time step equal to 0.001 s.

Comparison of Results and Emptying Patterns

The prediction of pressure and velocity values, as two important parameters for pipe operation, is fulfilled in both 1D and 2D CFD models and the results are compared to experimental observations. All the results discussed in the coming lines are presented until 3 s of the simulation when all the major changes occur. Figure 2 shows the changes in pressure during the emptying process demonstrating different variation trends. Also, the root mean square error (RMSE) for all the pressure simulations have been calculated as $\mathrm{RMSE} = [\sum_{i=1}^{n} (\exp_i - \mathrm{num}_i)^2 / n]^{1/2}$ where $\exp_i$ and num_i are experimental and numerical data respectively and n is the data size. RMSEs are presented in Table 2. The RMSEs declare a close performance of the 1D and 2D CFD models. Based on the emptying behaviour, the tests are categorized into three patterns:

- Pattern 1; equal left and right air pocket lengths with a partial valve opening (i.e., tests 1, 2 and 3). The valve opening percentage is very small, so there is a low and uniform change at the air-water interface. A very slow but constant descending of water level occurs. No backflow air occurs and the pressure is changing gradually with no oscillation. The results from 2D CFD simulations are very close to those from the 1D model, both presenting good accordance with the measured data (Figure 2 a, b and c).

- Pattern 2; equal left and right air pocket lengths with a total valve opening (i.e., tests 4, 5 and 6). In this pattern, the air-water interface is oscillating during the emptying process considerably. For low VFR case (test 4), the pressure is highly oscillating ending up with fast dissipation (Figure 2d) as the pressure amplitude for each wavelength proves by showing bigger changes when compared to other cases. In this case, the 2D CFD simulation predicts the pressure in first concavity higher than the experimental value. However, it predicts the pressure trend better than the 1D model. The wavelength for test 4 is considerably smaller when compared to tests with larger air pockets. Accordingly, increasing the air pocket size in pattern 2 increases the period and decreases the amplitude of oscillating pressure wave. Both models are able to predict the pressure measurements properly as shown in Figure 2 e and f.
- Pattern 3, unequal left and right air pocket lengths with a total valve opening (i.e., tests 7 and 8). During the emptying, in the longer air pocket branch, the water level ascends steadily and the water level in the shorter air pocket branch starts a uniform descending due to the higher applied gravity force. The air pocket reaches the highest expansion state at time 0.40 s and a reverse flow takes place in longer air pocket branch immediately after that. Following the reverse flow occurrence, the air intrudes from the downstream of the longer air pocket branch at t=0.50 s. This tendency continues until arising an equilibrium between inertia and gravity forces. From this point on, the gravity force in the longer air pocket branch becomes higher and then the movement starts in the opposite direction, i.e., ascending the water level in the shorter air pocket branch and descending in the longer air pocket branch. The pressure variation in pattern 3 involves a faster dissipation as shown in Figure 2 g and h. Due to the air-water interface movement,

the pressure simulation is more challenging and the pressure predictions after time 1.50 s are not very accurate.

The velocity has been measured at a distance of 1 m from pipe ends using the UDV. The measurement of the flow velocity inside the pipe was not possible in the pattern 1 due to the very low velocities. The velocity measurements were carried out in the right branch for pattern 2 and in both right and left branches for pattern 3. Both 2D CFD and 1D models are able to predict the general trend of the mean velocity variations during the time (Figure 3). However, the simulation of the experimental mean velocity values is not accurate. This lack may result from the difficulty to measure the velocity with the UDV in existing two-phase transient conditions with the backflow air occurrence. The amount of backflow air intrusion directly affects the accuracy of the measurements. For that reason, the simulation results are better in Tests 4 and 5, Figure 3 when less backflow air intrudes. Higher backflow air intrusion in Tests 7 and 8, Figure 3 leads to bigger difference between the measured and calculated data. Since the 1D model is not able to predict the backflow air accurately, a big difference is observed as well between the 1D and 2D CFD calculated data.

Backflow Air Analysis

The backflow air prediction provides significant information towards understanding the emptying process. In pattern 1, no backflow air has been observed due to partial opening and very slow valve actuation. For patterns 2 and 3, the air volume fraction contour is presented in Figure 4 at time 0.50 s showing the backflow air occurrences and the flow velocity vectors. For all the cases, the backflow air appears soon after opening the valve. Referring to Figure 2d (test 4), the lower pressure has been already attained at time 0.40 s prior to starting of air intrusion at time 0.50 s. In fact, the air pocket for test 4 has faced the maximum expansion at time 0.40 s and after that despite opening the valve more, a contraction starts in the air pocket which leads to a reverse flow. Immediately after the reverse flow, the air intrusion starts (Figure 4a). For higher VFRs at pattern 2 (tests 5 and 6), the reverse flow and backflow air occurrences are delayed. However, at time 3 s in Figure 5, the same backflow air intrusion occurred for tests 4, 5 and 6 related to pattern 2 leading to the formation of an air binding.

For pattern 3 with different left and right air pocket lengths, the air starts intruding from the longer air pocket branch. A reverse flow immediately emerges in the branch with longer air pocket size (the right branch in tests 7 and 8, Figure 4d and e). In the branch with shorter air pocket length (longer water column), the emptying occurs sooner due to higher gravity force. Also, the volume of backflow air in pattern 3 is mainly greater than pattern 2 at the same time as Figures 4 and 5 demonstrate. At pattern 2, dominant two-phase flow patterns of stratified flow or wave flow occur (Figures 4 and 5) in the horizontal pipes during the emptying. Also, the backflow air volume in test 8 with highest VFR is more than that for test 7.

Figure 6 shows the air volume fraction of reverse flow in the right branch from time 0.50 s to 3.00 s. For all the tests, the majority of the pipe cross section is occupied by the air forming an air binding phenomenon. The backflow air grows in the upstream direction and upon reaching upstream of the valve, a stratified flow takes place in the downstream region. For pattern 2 (including tests 4, 5 and 6), increasing the VFR decreases the volume of the backflow air at the beginning of the emptying. But, for the rest of the process, the backflow air amount remains constant. For pattern 3 (including tests 7 and 8), increasing the VFR leads to higher air intrusion.

The worst sub-atmospheric pressures for tests 5 and 8 with very close VFR values are almost the same due to the fact that the pressure variation is mostly a function of VFR rather than the situation of the air pocket (i.e., equal or unequal left and right lengths). However, the backflow air is much higher in test 8 as shown in Figures 4, 5 and 6, which is a result of very high air-water interface movement due to different initial levels. These results prove that the worst sub-atmospheric pressure is mainly dependent on the VFR value rather than the backflow air magnitude or the situation of the air pocket. However, for equal VFRs, a higher backflow air leads to higher peak pressure which is evident comparing tests 5 and 8. The backflow air intrusion can increase the peak pressure considerably as it has occurred in test 7 when compared to test 4. The effect of the backflow air intrusion in a higher VFR test (test 7) has created a peak pressure equal to a low VFR test (test 4) which was unexpected.

Conclusions

The proposed 1D mathematical and 2D CFD models were used to simulate the pressure and velocity changes during an emptying process and detect the backflow air behaviour. Both models were able to adequately predict the pressure and velocity oscillations during the initial vibrations mostly until 1.5 s. The RMSEs presented in Table 2 show very close performance of both models. In general, the 1D model seems acting slightly better only in the prediction of pressure variations but impossible to show the air-water interface and the backflow air entrance. It was revealed that for a 6% partial opening percentage of the valve (pattern 1), the drainage occurs confidently with no backflow air until the end of the emptying process. So, no fluctuation in the air-water interface occurs and the pressure remains in a safe limit with a uniform change. For equal left and right air pocket sizes with a total valve opening (pattern 2), the backflow air entrance is notable which decreases the drainage capacity and, as a result, causes considerable fluctuation in the air-water interface. In this case, the pressure drops to a quite low sub-atmospheric value. For unequal left and right air pocket branches (pattern 3), the air entrance rate is much higher leading to robust fluctuations in the air-water interface and the pressure dampens faster. In this case, the backflow air amounts in two branches are not equal and more air enters the pipeline from the branch with longer air pocket size.

Results show that the sub-atmospheric pressure magnitude in the first concavity is not highly dependent on the backflow air amount. The sub-atmospheric pressure depends mostly on the air pocket size. But the peak pressure in the first pressure jump is dependent on the backflow air magnitude. The bigger the backflow air is the higher peak pressure occurs in the first pressure jump.

The worst sub-atmospheric pressure is related to the lowest VFR and total opening of the valve. In this case, the sub-atmospheric pressure takes place when the valve is not opened considerably and no backflow air exists yet. So, for very low VFRs (very small air pockets), the emptying must be done with careful attention.

Authors suggest studying the effect of backflow air on emptying process in different pipe profiles for future works. Also, it would be valuable if the emptying process is evaluated in a controlled condition to hinder backflow air and compare results.

Acknowledgement

The authors acknowledge the support of the Civil Engineering, Research, and Innovation for Sustainability centre (CERIS) from Instituto Superior Técnico, University of Lisbon, Portugal, for providing the experimental apparatus and the financial support by Fundação para a Ciência e a Tecnologia (FCT), Portugal. Also, the authors want to thank the project REDAWN (Reducing Energy Dependency in Atlantic Area Water Networks) EAPA_198/2016 from INTERREG ATLANTIC AREA PROGRAMME 2014–2020.

References

ANSYS FLUENT R19.0 academic [Computer software]. ANSYS, Canonsburg, PA.

Benjamin, T.B. 1968. Gravity currents and related phenomena. *J. Fluid Mech.* 31(2): 209–248.

Besharat M., and H.M. Ramos. 2015. Theorical and experimental analysis of pressure surge in a two-phase compressed air vessel. 12th *International Conference on Pressure Surges*, BHR Group, Ireland, Dublin, 729-744.

Besharat, M., R. Tarinejad, and H.M. Ramos. 2016a. the effect of water hammer on a confined air pocket towards flow energy storage system. *Journal of Water Supply: Research and Technology - AQUA* 65 (2): 116-126.

Besharat, M., R. Tarinejad, M.T. Aalami, and H.M. Ramos. 2016b. Study of a compressed air vessel for controlling the pressure surge in water networks: CFD and experimental analysis. *Water Resources Manage*. 30(8): 2687-2702.

Besharat M., M.T. Viseu, and H.M. Ramos. 2017. Experimental study of air vessel sizing to either store energy or protect the system in the water hammer occurrence. *Water* 9(1): 63.

Bowker, R.P.G., G.A. Audibert, H.J. Shah, and N.A. Webster. 1992. Detection, control, and correction of hydrogen sulfide corrosion in existing wastewater systems, Office of Wastewater Enforcement and Compliance, Office of Water, Washington, DC.

Cebeci, T. 2004. Turbulence models and their applications. *Horizons Pub. Inc.*, Springer.

Coronado-Hernández, O.E., V.S. Fuertes-Miquel, M. Besharat, and H.M. Ramos. 2017. Experimental and numerical analysis of a water emptying pipeline using different air valves. *Water* 9(2): 98.

Coronado-Hernández, O.E., V.S. Fuertes-Miquel, M. Besharat, and H.M. Ramos. 2018. Subatmospheric pressure in a water draining pipeline with an air pocket. *Urban Water J*. (accepted).

Edmunds, R.C. 1979. Air binding in pipes. *J. Am. Water Works Associ*. 71(5): 273-277.

Escarameia, M. 2007. Investigating hydraulic removal of air from water pipelines. *Proc. Inst. Civ. Eng. Water Manage.* 160(1): 25-34.

Izquierdo, J., V. Fuertes, E. Cabrera, P. Iglesias, and J. Garcia-Serra. 1999. Pipeline start-up with entrapped air. *J. Hydraul. Res*. 37 (5), 579-590.

Laanearu, J., I. Annus, T. Koppel, A. Bergant, S. Vučkovič, Q. Hou, A.S. Tijsseling, A. Anderson, and J.M.C. van't Westende. 2012. Emptying of large-scale pipeline by pressurized air. *J. Hydraul. Eng*. 138(12): 1090-1100.

Laanearu, J., D.Q. Hou, and A.S. Tijsseling. 2015. Experimental and analytical study of the air-water interface kinematics during filling and emptying of a horizontal pipeline. 12th *Int. Conf. on Pressure Surges*, BHR Group, Ireland, Dublin, 625-637.

Launder, B.E., and D.B. Spalding. 1972. Lectures in mathematical models of turbulence. *Academic Press*, London, England.

León, A., M. Ghidaoui, A. Schmidt, and M. Garcia. 2010. A robust two-equation model for transient-mixed ows. *J. Hydraul. Res.* 48 (1), 44-56.

Martins, N., J., Delgado, H.M., Ramos, and D., Covas. 2017. Maximum transient pressures in a rapidly filling pipeline with entrapped air using a CFD model. *J. Hydraul. Res*. 55(4): 506-519.

Martins, S.C., H.M. Ramos, and A.B. Almeida. 2015. Conceptual analogy for modelling entrapped air action in hydraulic systems. *J. Hydraul. Res.* 53 (5), 678-686.

Pozos, O., C.A. Gonzalez, J. Giesecke, W. Marx, and E.A. Rodal. 2010. Air entrapped in gravity pipeline systems. *J. Hydraul. Res*. 48(3): 338-347.

Ramezani, L., B. Karney, and A. Malekpour. 2016. Encouraging effective air management in water pipelines: A critical review. *J. Water Resour. Plann. Manage*. 142(12): 1-11.

Richards, R.T. 1962. Air binding in water pipelines. *J. AWWA*. 68(6): 719-730.

Tijsseling, A., Q. Hou, Z. Bozkus, and J. Laanearu. 2016. Improved one-dimensional models for rapid emptying and filling of pipelines. *J. Press. Vessel Technol*. 138: 031301.

Triki, A. 2016. Water-hammer control in pressurized-pipe flow using an in-line polymeric short-section. *Acta Mech.* 227, 777-793.

Vasconcelos, J.G., and S.J. Wright. 2008. Rapid flow startup in filled horizontal pipelines. *J. Hydraul. Eng*. 134(7): 984-992.

Wang, H., L. Zhou, D. Liu, B. Karney, P. Wang, L. Xia, J. Ma, and C. Xu. 2016. CFD approach for column separation in water pipelines. *J. Hydraul. Eng*. 142(10): 1-11.

Wilcox, D.C. 2006. Turbulence modeling for CFD. *DCW Industries*, 3rd edition.

Wisner, P.E., F.N. Mohsen, and N. Kouwen. 1975. Removal of air from water lines by hydraulic means. *J. Hydraul. Div*. 101(HY2): 243-257.

Zhou, F., M. Hicks, and P.M. Steffler. 2002. Transient flow in a rapidly filling horizontal pipe containing trapped air. *J. Hydraul. Eng*. 128(6), 625–634.

Zhou, L., D., Liu, and C., Ou. 2011. Simulation of flow transients in a water filling pipe containing entrapped air pocket with VOF model. Eng. Appl. Comp. Fluid Mech. 5(1): 127–140.

Zhou, L., D. Liu, B. Karney. 2013. Investigation of hydraulic transients of two entrapped air pockets in a water pipeline. *J. Hydraul. Eng*. 139 (9), 949-959.

Zhou, L., H., Wang, B., Karney, D., Liu, P., Wang, and S., Guo. 2018. Dynamic behavior of entrapped air pocket in a water filling pipeline. *J. Hydraul. Eng*., 144(8): 04018045

Zukoski, E.E. 1966. Influence of viscosity, surface tension, and inclination angle on motion of long bubbles in closed tubes. *J. Fluid Mech*. 25(4): 821–837.

Table 1. Experimental tests specifications

Table 2. RMSEs for simulation of the pressure data

Figure 1. Real physical and 2D CFD model; (a) model and mesh; (b) an open valve; (c) a partially open valve; (d) a closed valve

Figure 2. Pressure change during pattern 1 (a, b and c), pattern 2 (d, e and f) and pattern 3 (g and h)

Figure 3. Flow velocity in the pipe for pattern 2 (a and b) and pattern 3 (c and d)

Figure 4. Air volume fraction contour and velocity vectors for different tests of patterns 2 and 3 at time 0.50 s

Figure 5. Air volume fraction contour and velocity vectors for different tests of patterns 2 and 3 at time 3.00 s

Figure 6. Air volume fraction development during backflow air action