

ANALYSIS ON FLOW CHARACTERISTICS OF COMBINATION AIR VALVE PROTOTYPE USING CFD TECHNIQUE: A 2D-VOF MODEL

Machimontorn Promtong^{*}, Sirawit Namchanthra, Akadath Leangruksa, Wanlop Choosri, Jetsadaporn Priyadumkol and Chakrid Vanasiri

Received: August 30, 2021; Revised: March 07, 2022; Accepted: March 26, 2022

Abstract

Due to vacuum and safety issues, combination air valves are necessarily introduced to evacuate the entrapped air before damaging piping system components. This research was aimed to introduce the Computational Fluid Dynamic (CFD) technique to investigate the flows in a combination air valve, especially while releasing the air under various operating conditions. A two-dimensional model was proposed to perform the simulation study to save the computing time. The boundary and conditions were derived from the actual experiment; specifically, the Volume of Fluids (VOF) method was employed to capture the interfacial motion of the air-water flows. The turbulent model, Realizable k - ϵ , was adopted to calculate the Reynolds Stress term. As a result, the predicted transient pressures, and the calculated air release times of both the cases installing and not installing the SARC agreed with the experiments; the maximum differences were 8.8% and 16.4%, respectively. According to the simulation works, it was found that the crucial factors to consider to evaluate air valve performance are transient pressure and velocity fields in the air valve and the transient force acting on the ball valve. In future work, a more comprehensive study will be conducted to find the critical criteria for the combination air valve design, especially to enhance the air removal performance.

Keywords: Computational Fluid Dynamics (CFD), Volume of Fluid Method (VOF), Volumn tracking modelling, Combination air valve, Air releasing performanc

Introduction

Air entrapped in a water pipe system can lead to problems that are invisible to the eye, and these unseen issues can range from minor issues to problems that could cause severe damage, such as poor control of flow and pressure, or the water hammer may damage pumps or even break the pipeline (Water and WastesDigest, 2016). Air

pockets in a pipeline can also result in head loss, dangerous high-velocity currents, and low system efficiency (Pozos-Estrada, 2007; Pozos-Estrada, 2018). Typically, these occur where operating options such as using different standpipes with varying manual or automatic shutoff valves (or check valves), are introduced (Gardell, 2013;

Department of Mechanical Engineering, Faculty of Engineering, Mahidol University, Thailand. E-mail: machimontorn.pro@mahidol.edu

^{} Corresponding author*

McElhaney, 2000). Nevertheless, the mentioned method may not be sufficient and can create additional problems if the operation is uncontrolled (Thorley, 1989). The most efficient method of controlling the air is to introduce standard air valves that allow the flow of a large amount of air continuously in a smooth and pleasant manner (ValveMagazine, 2017). However, to maximize efficiency, there is a requirement to understand the standard application of each type of air valve and understanding the installation (ValveMagazine, 2018). Furthermore, deep insight into the fundamental physics of air-release through the air valve is critical if a new design with improved efficiency is required (Darby, 2013; Aldeeb *et al.*, 2014; Darby and Aldeeb, 2014).

Many researchers have conducted experiments into the peak transient pressure parameters for the installation of air valves in a pressurized pipeline system (Zhou *et al.*, 2002a; Lee, 2005; Zhou *et al.*, 2013; Valdés *et al.*, 2014; Wu *et al.*, 2015). Tran (Tran, 2017) studied possible air-valve models to determine the pressure when differences of the filling rate were considered along with the air-valve characteristics such as the orifice size, the discharge coefficient, the mass, the density of float, and the maximum travel). Similarly, Zhou (Zhou *et al.*, 2002b) found that adjusting the filling rate and changing the shape of the orifice affected the pressure transient readings from the air valve outlet pipe. As Wu (Wu *et al.*, 2015), mentioned, the two stages of closing the valves reduce the pressure on the emergence of the water hammer.

Similarly, the pressure transient depends on the size of the orifice. However, it is noteworthy that the model without an orifice has a longer period in which to enter a steady state than the type with an orifice. Different air valves have can evacuate air at different rates, which means they have a suitable operating range. Therefore, a study on the influence of internal components of the air valve such as the inlet and outlet size, the orifice, and the internal shape must be considered.

Some researchers have focused on using the Computational Fluid Dynamics (CFD) technique to investigate flow characteristics and what happens when pressure surges occur while draining the entrapped air using the air valves (Davis and Stewart, 2002; Turesson, 2011; Lai *et al.*, 2017; García-Todolí *et al.*, 2018, Lai *et al.*, 2018; Kim and Jeong, 2021). Kim (Kim and Jeong, 2021) stated that the CFD method required a dynamic mesh technique (moving mesh) to observe the flow field due to the transient and instability issues. Moreover, García-Todolí *et al.* (García-Todolí *et al.*, 2018) employed a CFD program named ANSYS Fluent to study and capture the motion of entrapped air

entering and draining out of the single air valve. However, as many reserchers mentioned (Promtong and Tekasakul, 2007; Promtong *et al.*, 2016; Suvanjumrat, 2017a; Menéndez-Blanco *et al.*, 2019, Promtong *et al.*, 2020, Suvanjumrat and Loksupsaiboon, 2020), the validation and mesh independence studies are mandatory tasks before this technique is applied to redesign the air valve or similar system performance. Although works have been completed into using CFD to elucidate the air valves, there has been no work investigating the flow characteristics inside combination air valves during air release.

In other words, the critical factors that can explain the performance of this air valve type have not yet been discussed, especially in terms of its structure. Therefore, this research will focus on studying the two stages of closing the air valve (called the combination air valve). The objectives of this research are:

- 1) to observe the flow characteristics inside two different types of combination air valve under actual operation conditions.
- 2) to investigate the flow parameters used to describe the performance of the combination air valve numerically.
- 3) to address possible alternatives that could enhance current performance, especially the critical factors that affect the drainage rate of the entrapped air.

The main discussion regards the flow characteristics that occur in the combination air valves. By comparing the key parameters the transient pressure, the times for the ball moving (air releasing time), the velocity and pressure contours, and the air performance of the combination air valves can be evaluated.

Fluid Flow Equations and Volume of Fluid (VOF) Method

The conservative continuity and momentum equations can be used to describe air and liquid flow characteristics in pipes. However, in this simulation, the conservative energy equation can be ignored because there is no interest in the temperature effect. Due to the turbulent flow, there is a requirement to have the turbulence closure for calculating the kinetic viscosity term.

Mass Equation

The mass equation can express the mass changes over time within a control volume (as shown in Equation (1)). Because the mass velocity is continuous, this partial differential equation can be called the continuity equation. It should be noted that if the fluid flow is in a steady-state, then the first term can be ignored.

$$\frac{\partial \rho}{\partial t} + \rho \frac{\partial (\bar{u}_i)}{\partial x_i} = 0 \quad (1)$$

Momentum Equation

Similar to the mass equation, the momentum transfers within fluid control volume must be conserved; hence the momentum equation can be described using Equation 2.

$$\rho \frac{\partial \bar{u}_i}{\partial t} + \rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \rho \overline{u_i u_j} \right] - \rho g_i \quad (2)$$

Where

$$\tau_{ij} = -\rho \overline{u_i u_j} = \mu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho k \quad (3)$$

In Equations (1) and (2), ρ is the fluid density and $\bar{p}$ is the system pressure. $\bar{u}_i, \bar{u}_j$ are the average velocity components and $\overline{u_i u_j}$ is the velocity fluctuation. The x_i, x_j terms are the coordinate axis and the g_j and μ terms represent the gravitational acceleration vector and kinetic viscosity, respectively. Equation 3 shows the *Reynolds-Stress term* ($-\rho \overline{u_i u_j}$) and this equation can be resolved by turbulent models.

Turbulence models are devised to represent high-order moments of velocity fluctuations in terms of the lower-order moments. This can be completed directly, as in the case of the Eddy-Viscosity models, or indirectly, as in the case of models based on the solution of additional partial differential equations.

Different turbulence models are available to determine the turbulent viscosity (μ_t) (Launder and Spalding, 1974; Menter, 1994) and recently some turbulence models were evaluated by researchers the prediction capability using the open source CFD program (Suvanjumrat, 2017b; Yuan *et al.*, 2017).

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (4)$$

In this study, the turbulent model adopted is the standard $k - \varepsilon$ model. The details of the kinetic energy of turbulence (k) and the dissipation rate (ε) are given below:

- k Equation:

$$\rho u_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + P + G - \rho \varepsilon \quad (5)$$

- ε Equation:

$$\rho u_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_1 \frac{\varepsilon}{k} (P + C_3 G) - C_2 \rho \frac{\varepsilon^2}{k} \quad (6)$$

The model constants are

$$C_\mu = 0.09, C_1 = 1.44, C_2 = 1.92, C_3 = 1.0, \sigma_k = 1.0, \sigma_\varepsilon = 1.217$$

The last term in equation (6), $\rho \varepsilon$ represents the destruction rate, and P and G are the shear production buoyancy production terms respectively, which are denoted by:

$$P = \mu_t \frac{\partial u_i}{\partial x_j} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (7)$$

$$G = g_i \beta \frac{\mu_t}{\sigma_t} \left(\frac{\partial \bar{T}}{\partial x_i} \right) \quad (8)$$

Volume of Fluid (VOF) Method

The numerical technique, Volume of Fluid (VOF), is introduced to track the movement on free surfaces (Fluid-Surface area). This method is proposed based on the Eulerian method, where the mesh is stationary or moving in a predetermined manner.

While shape-changing, this numerical technique allows programmers to track the surface areas' present position and reshapes them according to the velocity direction. Thus, the numerical method is not an algorithm solving the flow equation (Equations 9-11) in itself; but the averaged Navier-Stokes equation also describes the fluid motion (where m represents the considered fluid phase). More details regarding the VOF method can be found in the work of Hirt and Aniszewski (Hirt and Nichols, 1981, Aniszewski *et al.*, 2014).

$$\frac{\partial C_m}{\partial t} + v \times \nabla C_m = 0 \quad (9)$$

$$\sum_{m=1}^n C_m \quad (10)$$

$$\rho = \sum_{m=1}^n \rho_m C_m \quad (11)$$

Experimental Details and Results

The experiments were conducted at Chaipong Engineering Co. Ltd. (Thailand), and the experimental setup can be seen in Figure 1. The parameters, including the transient pressures, volumetric flow rate, and the transient motion of the floating ball, were recorded using a pressure transducer, flow meter, and high-speed camera, respectively. The transient pressure was recorded every 0.15 seconds using a data logger.

During the experiment, the 3D printing model of the combination air valve was replaced with an actual one due to the requirement to capture the ball motion draining the air visually.

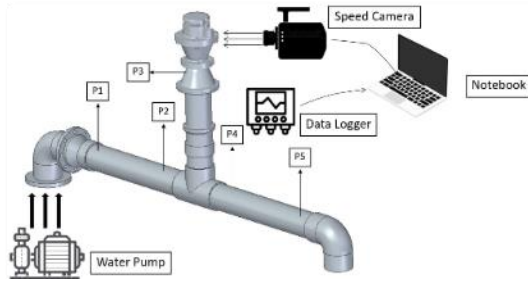


Figure 1. Schematic Chart of the Experimental Setup

Six different flow rate conditions were investigated but only two sets were considered for the simulation study. The results of the experiment, including the transient pressure and the ball moving time are presented and discussed, before benchmarking using the simulation results can be actioned.

Figure 2 shows the results of transient pressure at five different measuring locations in the piping system. The two graphs show results with and without the Secondary Air Release Chamber (SARC) installed. Without the SARC, the secondary outlet was closed intentionally so that it would have just a single air valve. These two cases have similar transient pressure changes, except for when the ball moving was (No. 3). The case with the SARC spent less time (3.30 sec) while draining the air than the case without the SARC (5.20 sec). Operating under the current flow rate, the maximum pressure found at the region under the ball valve (P3) was 0.27 Bars, and a slightly lower rate was found with the SARC.

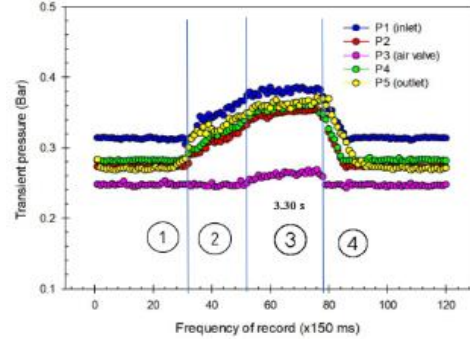
The results show that the combination air valve (with the SARC installed) is more effective in terms of air evacuation than the case with no SARC. Also, it can be implied that if there is a large volume of air in the pipeline, installing a combination air valve could allow better drainage than a single air valve.

Two-Dimensional Simulation

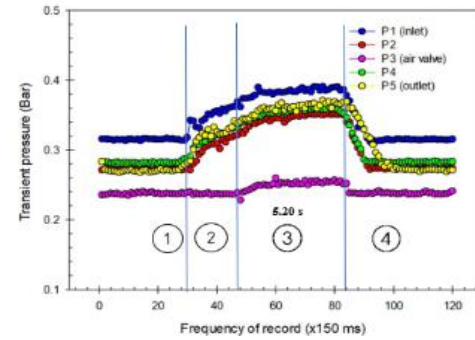
The simulation studies, which are described in this section, include numerical details, the mesh independence study, and a discussion of the key parameters results. As the piping system is symmetrical along the x-y plane (Figure 1), the flow domain for the simulation can only be considered on the cross-sectional plane, making it a two-dimensional study.

To capture the flow characteristics of the combination air valve, the actual operating conditions, such as the inlet pressure and inlet volumetric flowrate, were adopted for the two different combination air valves (with/without SARC). Special attention was given to the

elucidation of the flow characteristics inside the combination air valves while releasing the air.



(a) With SARC Installed



(b) Without SARC

Figure 2. Transient pressure occurring during the operation at considered flow rate ($Q=9.64 \times 10^{-3} \text{ m}^3/\text{s}$)

Numerical Description

In this simulation, the fluid flow in the pipeline was used to create the combination air valve, which was performed using the CFD program named ANSYS Fluent 2021 R1. To tracking the interfacial surface between the air and water, the implicit VOF method was introduced.

In addition, the introduction of the Dynamic Mesh (Moving Mesh) technique can enhance the motion of the ball valve when the force is applied as it makes it smoother. For the turbulence model, the realizable k- ϵ model was adopted for the Reynold Stress term calculation.

During the transient investigation, five locations (P₁-P₅) were used for collecting the transient pressures in the piping system. The ball motion during lifting and the touching of the orifice (air outlet) was also captured. For the benchmarking process, the crucial parameters from the simulations, including the monitored pressures, and

the ball motion time, were benchmarked with the experiment results. Moreover, the pressure contours and velocity vectors surrounding the ball valve were taken for comparison and discussion alongside the cases with and without the SARC.

Mesh Independence Study

For computational efficiency purposes, the smallest number of meshes that could give a satisfactory numerical accuracy was investigated. Also, the mesh refinement was specified for the sensitive area, and as aforementioned, it was situated near the ball valve region.

Table 1 illustrates the predicted pressures from the mesh numbers compared to the time when results converged and yielded the acceptable percentage figure (should be less than 5% difference).

Table 1. Difference of Predicted Pressure Readings with Different Element Meshes

Element Size (m)	Number of Elements	Pressure (P_3) (Pa)	Result Difference (%)
0.0070	21,974	1,039.76	62.48
0.0050	45,065	2,164.71	21.88
0.0040	55,513	2,876.25	3.79
0.0015	390,068	2,855.40	3.04
0.0010	845,734	2,771.16	-

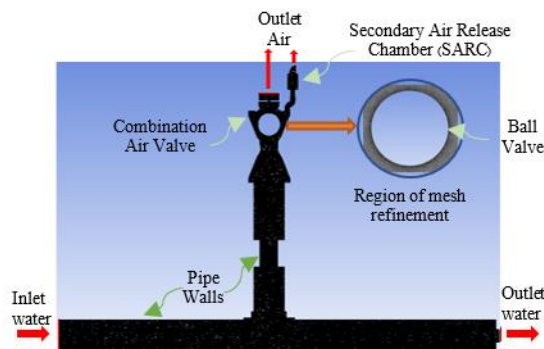


Figure 3. The Flow Domain After Meshing and the Refinement Zone Near to the Ball Valve

The Geometry and Local Mesh Details

The geometries of the pipeline and combination air valve were separated into small elements because of the meshing process. According to the mesh independence study, the total elements used in the simulation numbered 55,513.

Details of the mesh size and mesh refinement can be seen in Figure 3, and the boundaries of the

flow domain, including the inlet, the outlets, the ball valve, and the pipe walls, are displayed in the figure.

Results and Discussion

In this section, the simulation results from the two different case studies are presented and discussed along with the experimental data. First, the simulated flow stage is explained to provide the fundamental physics of the fluid flow in the piping system and the changes in the transient pressure.

Second, the essential parameters, including the transient pressure, the ball moving time, velocity, and pressure profiles, are discussed. It should be noted that these results were from the simulations with different forms of the combination air valves (with and without the SARC installed) under the same flow conditions.

Flow Scenario Descriptions

The flow simulation can be divided into four main scenarios, which all correlate to actual flow behaviors. Due to the transient consideration, the predicted pressure of each stage can be compared with the experimental data. However, only the first three stages of the flow simulation results are explained in this discussion, as displayed in Figure 4.

Stage 1: The water continues to fill the horizontal pipe. (Figure 4(a)) shows the start of this stage, when there is only air in the piping system. Without this information from the inlet, this stage may provide unacceptable simulation results.

Stage 2: The water rises in the vertical pipe after the horizontal pipe is completely filled with water (Figure 4b). If the horizontal pipe is not filled this stage will not happen.

Stage 3: The water pushed the ball to remove the air from the piping system, which is called the "air releasing time" period (Figure 4(c)).

Stage 4: The ball drops due when the water supply is discontinued. Although this stage is shown in the experiments, it is difficult to capture this flow scenario due to the flow complexities in the simulation.

Table 2 shows the predicted transient pressure for the position under the ball valve (P_3) and the air releasing time (during Stage 3) compared to the results from the experiments. When SARC is installed, the predicted pressure and the air releasing time figures were overestimated at 8.8% and 5.2%, respectively.

In contrast, for when SARC is absent (working as a single air valve), both predicted valve figures were underestimated at 1.5% and 16.4%,

respectively. In general, simulation cases could give similar transient pressures and the air releasing times with the experiments.

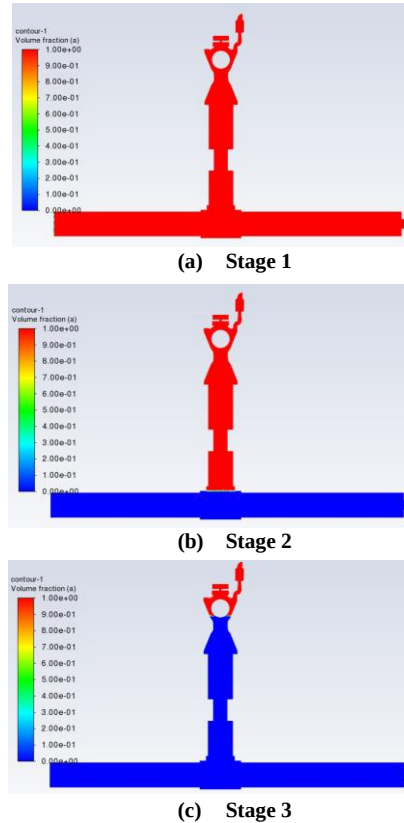


Figure 4. Volume Fractions of the Water-Air Flow in Pipeline for Stage 1, 2, and 3

Table 2. Comparison of the Results between the Simulation and Experiments

Type	Source	Maximum Transient Pressure (P_3) (Pa)	Air releasing time (s) (Stage 3)
With SARC	Simulation	2,340	1.692
Installed	Experiment	(8.8%) 2,151	(5.2%) 1.784
Without SARC	Simulation	2,392	2.251
	Experiment	(1.5%) 2,428	(16.4%) 1.933

Instead of using the VOF, different modern front-tracking methods (like, coupling with the level set and a hybrid with particle technique) might be presented to improve the numerical accuracy. Also, the case of no SARC installed gives both the predicted transient pressure and the air releasing time higher than the case of the SARC installed. According to these results, it could be implied that having the SARC installed for the combination air

valve would give better performance in terms of air removal.

Other Interested Parameters

One advantage of the simulation method is that it provides variables that may be difficult to experiment for. For instance, the parameters include the pressure contours, velocity vectors, and the force acting on the ball valve while moving. Figure 5 shows the fluid flow characteristics around the ball valve for two different conditions, which are when the ball valve is almost closed at the orifice hole (the air outlet).

As shown in Figure 5(a), the pressure contour of the two cases was highest at the center of the ball. This is because of the large impact force acting on that region during the air release out of the orifice. Comparing the cases with and without the SARC installed, on average the pressure distribution inside the air valve where SARC is more uniform (especially the haft top region) than without SARC installed. This result was due to the influence of the velocity distribution.

Figure 5(b) presents the velocity profiles for both cases with the same operating conditions. On average, having the SARC installed creates a lower velocity than without the SARC because the air drains from the air valve through the secondary chamber.

Nevertheless, the case with no flow passing through the SARC shows high velocities going through the orifice from both sides of the ball valve. According to the figure, the maximum predicted velocities for both cases were between 5.30-7.42 m/s.

As shown in Figure 6, the transient force is very high when the water impacts the ball valve (the case without SARC installed). This could damage the internal components of the air valve; later, the force becomes a constant at 80 N. This causes the ball to float and push the distributed pressure surrounding it.

Reducing the time taken to stabilize the transient force may also decrease the air release time, improving the air drainage time. Monitoring the transient force with different floating weights and orifice hole diameters could give an idea to redesign the air valves.

Conclusion and Future Work

The experiments and numerical works to elucidate the flow characteristics during the air releasing period inside a piping system with two combination air valve conditions have been performed

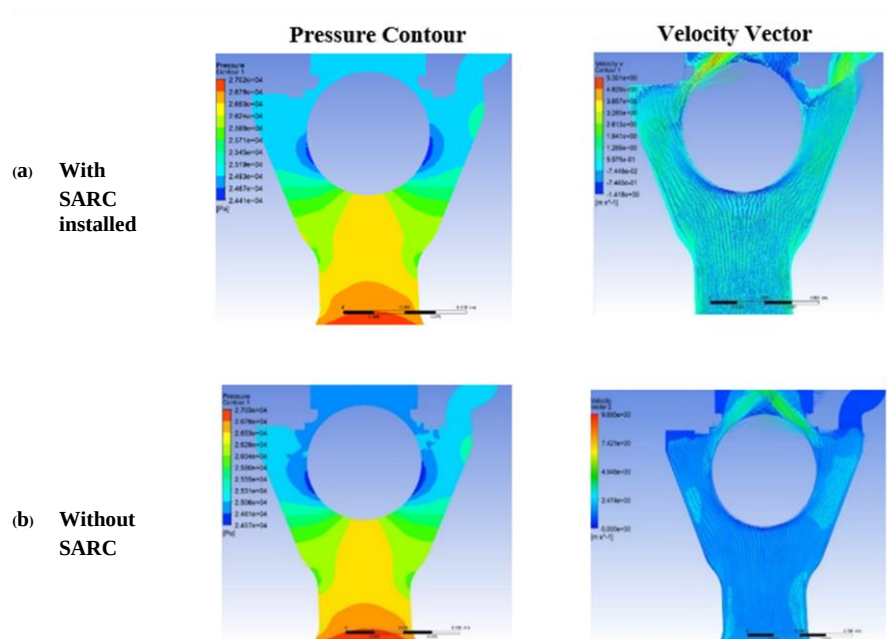


Figure 5. Pressure Contour and Velocity Vector of Stage 3; (a) With SARC installed (Top Row) and (b) Without SARC (Bottom Row)

successfully. After adopting the actual operating conditions as the initial and boundary condition of the simulation, the predicted pressure and the predicted air release time were found to have a good agreement with the experiment data. Overall, the simulations could capture the physical conditions of the flow in the air valve system. Also, some crucial parameters regarding the performance can be discussed, and conclusions can be drawn as follows:

When comparing both cases, the predicted transient pressures (P_3) and the air release times agreed with the experiments; the maximum differences were 8.8% and 16.4%, respectively.

The air release time with SARC installed was less than without SARC installed. Therefore, having the SARC improves the release of the air.

The pressure contour and velocity vectors inside the air valve reflect the influence of the inner air valve structure, so monitoring the transient force acting on the ball valve could be important to investigate the air removal performance.

The experiments will be attempted further to obtain a more definite conclusion in future work. As well as considering more realistic situations in the simulation, for instance, adding the air from the inlet boundary ; thus, the numerical investigations would provide more validity and insights.

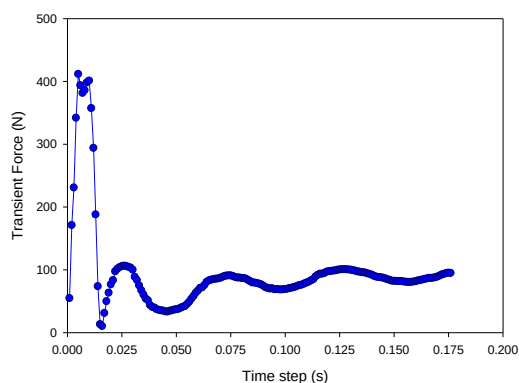


Figure 6. Predicted Transient Force Acting on Ball Valve while Elevating

Acknowledgments

This research has been supported by the Mechanical Engineering Department, Faculty of Engineering, Mahidol University.

Thanks to CADFEM SEA Pte. Ltd. - A Certified Elite Channel Partner to ANSYS - for providing support and access to the CFD program named ANSYS Fluent 2021 R1.

Also, Thanks to Chaipong Engineering Co., Ltd. for allowing to use the facilities.

Thanks to Robert McEvoy, from the Faculty of ICT at Mahidol University for editing and proofreading the manuscript.

References

- Aldeeb, A.A., Darby, R., and Arndt, S. (2014). The dynamic response of pressure relief valves in vapor or gas service. Part II: Experimental investigation. *J. Loss Prev. Proc. Indust.*, 31:127-132.
- Aniszewski, W., Ménard, T., and Marek, M. (2014). Volume of Fluid (VOF) type advection methods in two-phase flow: A comparative study. *Comp. Flu.*, 97:52-73.
- Darby, R. (2013). The dynamic response of pressure relief valves in vapor or gas service, part I: Mathematical model. *J. Loss Prev. Proc. Indust.*, 26(6):1,262-1,268.
- Darby, R. and Aldeeb, A.A. (2014). The dynamic response of pressure relief valves in vapor or gas service. Part III: Model validation. *J. Loss Prev. Proc. Indust.*, 31:133-141.
- Davis, J.A. and Stewart, M. (2002). Predicting Globe Control Valve Performance-Part I: CFD Modeling. *J. Fluids Eng.*, 124(3):772-777.
- García-Todolí, S., Iglesias-Rey, P., Mora-Meliá, D., Martínez-Solano, F., and Fuentetaja, V. (2018). Computational Determination of Air Valves Capacity Using CFD Tech. *Water*, 10(10).
- Gardell, J. (2013). Benchmark of RELAP5 Check Valve Models against Experimental Data.
- Hirt, C.W. and Nichols, B.D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. *J. Comp. Phys.*, 39(1):201-225.
- Kim, N-S. and Jeong, Y-H. (2021). An investigation of pressure build-up effects due to check valve's closing characteristics using dynamic mesh techniques of CFD. *Ann. Nuclear Energy*, p. 152.
- Lai, Z., Karney, B., Yang, S., Wu, D., and Zhang, F. (2017). Transient performance of a dual disc check valve during the opening period. *Ann. Nuclear Energy*, 101:15-22.
- Lai, Z., Li, Q., Karney, B., Yang, S., Wu, D., and Zhang, F. (2018). Numerical Simulation of a Check Valve Closure Induced by Pump Shutdown. *J. Hyd. Eng.*, 144(12).
- Launder, B.E. and Spalding, D.B. (1974). The numerical computation of turbulent flows. *Computer Methods in Appl. Mech. Eng.*, 3(2):269-289.
- Lee, N.H. (2005). Effect of Pressurization and Expulsion of Entrapped Air in Pipelines. *J. Appl. Fluid Mech.*, 10(1):11-20.
- Mcelhaney, K.L. (2000). An analysis of check valve performance characteristics based on valve design. *Nuclear Eng. Design*.
- Menéndez-Blanco, A., Fernández Oro, J.M., and Meana-Fernández, A. (2019). Unsteady three-dimensional modeling of the Fluid-Structure Interaction in the check valves of diaphragm volumetric pumps. *J. Fluid Struct.*, 90:432-449.
- Menter, F.R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA J.*, 32(8): 1,598-1,605.
- Pozos-Estrada, O. (2007). Investigation on the Effects of Entrapped Air in Pipelines. In: Faculty of Civil and Environmental Engineering). Institut für Wasserbau der Universität Stuttgart
- Pozos-Estrada, O. (2018). Investigation of the combined effect of air pockets and air bubbles on fluid transients. *J. Hydro.*, 20(2):376-392.
- Promptong, M., Cheung, S., and Tu, J. (2016). Numerical modelling of subcooled boiling flow based on mechanistic approach: A validation study using wet steam (IAPWS) as working fluid properties. *Lecture notes in engineering and computer science*.
- Promptong, M., Khunsri, K., Teachapanitvittaya, K., Trakulkumle, T., Watechagit, S., and Suvanjumrat, C. (2020). Experimental and numerical investigations into the natural convection of hot gas in a vertical smoking oven: a validation study. In: The 34th Conference of the Mechanical Engineering Network of Thailand., 15-17 July 2020, Prachuap Khiri Khan, Thailand, p. 484-494.
- Promptong, M. and Tekasakul, P. (2007). CFD study of flow in natural rubber smoking-room: I. Validation with the present smoking-room. *Appl. Thermal Eng.*, 27(11-12):2,113-2,121.
- Suvanjumrat, C. (2017a). Comparison of Turbulence Models for Flow Past NACA0015 Airfoil Using OpenFOAM. *Eng. J.*, 21(3):207-221.
- Suvanjumrat, C. (2017b). Implementation and Validation of OpenFOAM for Thermal Convection of Airflow. *Eng. J.*, 21(5):225-241.
- Suvanjumrat, C. and Loksupsaiboon, K. (2020). Improvement of Thermal Distribution in the Rubber-Glove Former Conveyor Oven by OpenFOAM. *Eng. J.*, 24(2):109-120.
- Thorley, A.R.D. (1989). Check Valve Behavior Under Transient Flow Conditions: A State-of-the-Art Review. *J. Fluids Eng.*, 111(2):178-183.
- Tran, P.D. (2017). Pressure Transients Caused by Air-Valve Closure while Filling Pipelines. *J. of Hydraulic Eng.*, 143(2).
- Tureson, M. (2011). Dynamic simulation of check valve using CFD and evaluation of check valve model in RELAP5.
- Valdés, J.R., Rodríguez, J.M., Monge, R., Peña, J.C., and Pütz, T. (2014). Numerical simulation and experimental validation of the cavitating flow through a ball check valve. *Energy Conversion and Management*, 78:776-786.
- Valvemagazine (2017) Why Air Valves are Needed in Water Applications Valvemagazine (2018) Air Valves in Piping Systems.
- Water and Wastesdigest (2016) How to deal with the dangers of entrapped air in liquid piping systems.
- Wu, Y., Xu, Y., and Wang, C. (2015). Research on Air Valve of Water Supply Pipelines. *Procedia Engineering*, 119:884-891.
- Yuan, H.L., Kekina, P., Suvanjumrat, C., Agarwal, R.K., Tandon, P., and Wang, E.X. (2017). A Comparative Study on Turbulence Models for Simulation of Flow Past NACA 0015 Airfoil Using OpenFOAM. *MATEC Web of Conferences*. 95.
- Zhou, F., Hicks, F., and Steffler, P. (2002a). Observations of Air-Water Interaction in a Rapidly Filling Horizontal Pipe. *J. Hydraulic Engineering-ASCE - J HYDRAUL ENG-ASCE*, 128.
- Zhou, F., Hicks, F., and Steffler, P. (2002b). Transient Flow in a Rapidly Filling Horizontal Pipe Containing Trapped Air. *J. of Hydraulic Eng.*, 128(6):625-634.
- Zhou, L., Liu, D., and Karney, B. (2013). Investigation of Hydraulic Transients of Two Entrapped Air Pockets in a Water Pipeline. *J. of Hydraulic Eng.*, 139(9):949-959.