







---

RESEARCH ARTICLE

---

EVALUATING THE PERFORMANCE OF A SIMPLE DEVICE FOR REDUCING  
PRESSURE SURGE EFFECTS USING EXPERIMENTAL AND NUMERICAL METHODS

Volkan KIRICCI <sup>1,\*</sup> , Ahmet Ozan CELIK <sup>1,2</sup> , Canberk INSEL <sup>2</sup> , Metin KAYA <sup>3</sup> 

<sup>1</sup> Department of Civil Engineering, Eskişehir Technical University, 26555 Eskişehir, Turkey

<sup>2</sup> Teqflow R&D Inc., Eskişehir, Turkey

<sup>3</sup> Arcelik A.S., Refrigerator Plant, Eskişehir, Turkey

ABSTRACT

This paper presents results from experimental and numerical investigations of a new device to prevent damaging effects of pressure surge phenomenon occurring in a refrigerator water dispenser system. The device, which will be presented in detail, is a low cost, small size, manufacturable, pressure surge damper and flow regulator made out of non-movable single plastic bodies.

CFD simulations were used in order to elaborate on the physics behind the feasibility of the new concept, and also to systematically guide the design procedure. One of the two shortlisted design models was also tested physically to have more reliable results and to evaluate accurately the performance. Both numerical simulations and experiment results show that the damper device significantly reduces the pressure surge induced pressure peak (effectively 25% and up to 37%) and protects the refrigerator water dispenser system.

Given the specifics of the problem, such as the size, the material (plastic) type and cost limitations of the components to be used in the pipe system; the ranges of flow rate and the pressures in the pipe flow as well as the resulting remedies and the presented product are rather case specific. However, the damper presented here, makes the idea expandable to different scales and conditions of similar pressure surge related problems. This is due to its simple features, manufacturability and effectiveness.

**Keywords:** CFD, Experimental fluid mechanics, Pressure surge, Pressure damper

---

1. INTRODUCTION

Pressure surge and water hammer are transient problems occurring in various types of engineering applications. These range from big scale fluid transmission lines to small scale home appliances such as refrigerators. In our case, when the water pressure in the pipe systems abruptly changes, the pipes, water tank or other parts of the water supply circuit in the refrigerator can be damaged. Such damages pose a severe risk to the safety of the device and the user. The abrupt changes in the water pressure are due to pressure waves (also known as pressure surge) which occur when the water intake valve of a water conducting appliance is suddenly closed.

It is common practice to install a pressure surge cushioning/damping device into the water network to protect the water conducting appliances against the abrupt changes in pressure. The pressure surge preventing devices generally comprise a tube with an inlet; an outlet; a flow passage, which connects the inlet to the outlet; and an expandable/contractible air chamber which is connected with the flow passage. While being highly effective, cost of such devices also get high.

A large number of resources in the literature exist on the theory of water hammer and pressure surge [1, 2, 3] which will not be repeated here. Common methods of protection from pressure surge and water hammer are given thoroughly in various classic sources [4, 5]. A considerable number of patents also

exist for water hammer and pressure surge related devices (nearly 10000 from a single Google Scholar search with the keyword “pressure surge”). These are usually different forms of check-valves or arrestors with sophisticated mechanisms of moving parts.

Several methods exist for analyzing pressure surge and water hammer phenomena numerically. Conventionally, explicit Method of Characteristics (MOC) has been preferred by many researchers due to the simplicity and the convenience it provides for one dimensional flow conditions. Method of implicit (MOI) is a more complex technique developed to cope with the accuracy and convergence problems [6]. In addition, modified and coupled versions of MOC and MOI are also tested and compared to improve efficiency of transient flow analyses to examine the wave oscillation period in pressurized steel-piping systems [7]. Tabelsi and Triki (2019) [8] studied a dual technique based branching strategy by splitting the single plastic short-penstock into dual plastic sub-short penstocks placed upstream of the original steel piping system. Dual technique is suggested as a useful tool to soften both first hydraulic-head peak and crest. In addition, it is shown that the efficiency of the conventional one is increased in terms of wave oscillation periods while providing acceptable trade-off between hydraulic-head peaks or crests attenuation. In another study, an algorithm was suggested to efficiently predict maximum allowed pressure in oil pipelines and an automatic control method for suppressing the back-propagating pressure waves [9]. Lee et al. (2001) [10], presented a method for the control of the pressure transients caused by a relatively fast valve or actuator stoppage in a simple hydraulic system. Different orifices were used to reduce pressure transients. Similarly, Asiaban and Fathi-Moghaddam (2018) [11], carried out an experimental study to compare the performance of a porous structure and different orifice throttles at the inlet of pressurized and open surge tanks. Results confirm considerable advantages of porous throttling over customary orifice throttling for faster transient dampening. In an analysis of 3D Computational Fluid Dynamics (CFD) water hammer simulations using “SIMPLE” algorithm, Jinping et al. (2010) [12] found that numerical results agree well with physical model tests. The next section however focuses on the literature with emphasis on both numerical modeling of the water hammer phenomenon and also the existing design of simplistic but effective protective/damping/arresting devices. Al-Khomairi (2005) [13] obtained similar high performances by installing a perpendicular pipe to the main line with air-filled balls. In another study Al-Khomairi (2010) [14] used short PVC pipe sections with diameters larger than that of the pipes in the rest of the system and found that these large diameter pipe sections are able to reduce the magnitude of the water hammer by up to 80%. Choon et al. (2012) [15] was able to reduce the water hammer by 1/3 employing a by-pass line with a non-return valve. While these efforts represent experimental work and also simple and new design, Meniconi et al. (2012) [16] carried out both experimental and numerical investigation particularly on determining the effect of abrupt geometrical changes in a pipe system on the water hammer. They concluded that simple benchmark tests may not be of great importance as they do not include the combined (and total) result of all different components in a pipe system. Nikpour et al. (2014) [17] validated their CFD model with physical tests and concluded that commercial codes are able to capture the water hammer phenomenon with reasonably well accuracy.

There are studies comparing several different approaches of numerically modeling the pressure surge and water hammer concluding that methods such as finite volume with various orders and method of characteristics for instance, yield fairly close results [18, 19, 20]. According to Reidelmeier (2014) [19], CFD with Shear Stress Turbulence (SST) model tends to overpredict the magnitude of the water hammer related pulses whereas 1D models without the complexities of a CFD model can achieve the same results. Wu et al. (2015) [21] on the other hand was able to couple a 1D model with a 3D CFD scheme to, in a way, include the interaction between a pump and pressure surge due to sudden valve closure. Zhang et al. (2016) [22] explained the main advantages of CFD over 1D models and also provided optimum parameters for the mesh, compressibility and time step (for transient solutions).

While the literature review can be extended, one obvious outcome is that if applied properly, CFD models can simulate the pressure surge phenomenon with accuracy enough for design stages, even for

cases with rotating machinery. On the other hand, case specific problems still exist as the one investigated here, which may not be addressed with the existing solutions also where expensive and sophisticated devices may not be applicable.

The aim is use the existing 3D-CFD methods proven to capture the dynamics of the phenomenon to evaluate the performance of a new and simple pressure surge damping and flow regulating device. The prototype of the device was first produced and tested physically using a highly sensitive pressure transducer. The experimental and numerical results were then compared. The results suggest a nearly 25% reduction in the pressure magnitude due to pressure surge in a system with the new device defined in this study.

## 2. DAMPER DESIGN

The design procedure started by setting the design limitations. The limits are concerning the space, material and the cost. Subsequently a concept design pool, all of which is not presented here was generated. The main goal was to create rooms in the fluid domain which would potentially arrest the pressure waves temporarily when the solenoid valve was closed. This way, the energy losses due to local capturing of the pressure waves would lead to reduction in the magnitude of the pressure surge in the system faster than it should otherwise. Another concern was that such a fixture shouldn't interrupt the flow when the dispenser system is in operation. Flow interruption is a phenomenon in the flow direction (downstream) while the pressure surge is a travelling elastic energy wave in upstream direction (Figure 1).

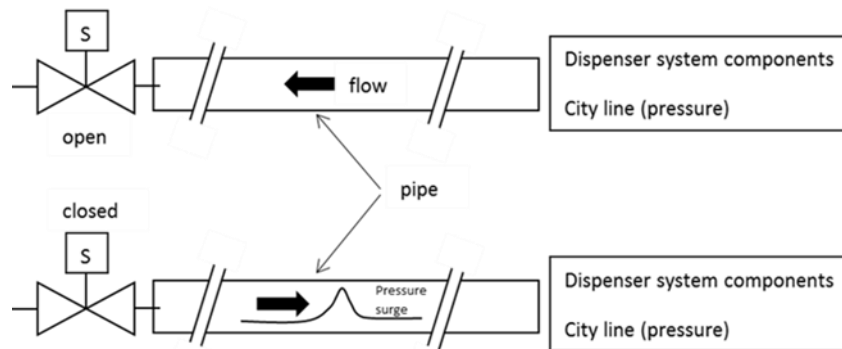


Figure 1. Representative drawing of the pipe system.

A short listed design pool was obtained using different shapes and orientations of box shaped rooms with inclined walls through the flow direction. Short cones with different orientations were used to form the room walls in the flow domain (pipe). CFD runs were consequently used to modify the geometry (the number of walls, their orientations with respect to each other and the edges of the room walls) to increase the positive effects.

## 3. EXPERIMENTAL AND NUMERICAL APPROACHES

### 3.1. Experiments

The fixture which was briefly mentioned in the previous section was 3D printed for physical tests with 50-micron resolution and 1:1 scale. Details of the damper device design and results of the experiment will be given in the following sections. Experiments were conducted in the Hydraulics Laboratory, Department of Civil Engineering, at Eskişehir Technical University. 6 mm (outer) diameter high density polyethylene pipes and matching quick-fits were used to create the pipe system. This arrangement,

although with a relatively short pipe length, is identical to that in a typical refrigerator with a water dispenser where different types of damages were also observed in the past. A solenoid valve was employed downstream of the system to generate the sudden blockage and the pressure surge. The system was fed from the water line in the laboratory with pressures near  $4 \times 10^5$  Pa (gage). The water tap pressure was monitored with a Bourdon pressure gauge. Omega PX409-1 pressure sensor with a 1000 psi (6894.76 kPa) full range, 1 kHz sampling frequency and 0.05% accuracy was used to capture the pressure surge in the pipe. A straight tube of 4 mm diameter (with 2 mm inner diameter) was used to connect the sensor with the 6 mm pipe along with a T-shaped 6mm-6mm-4mm quick-fit (location II). Pressure data were measured at a sampling rate of 1000 Hz. Two pressure sensor locations were considered downstream and upstream of the pipe, shown as locations I and II respectively in Figure 2 both 50 cm away from the damper.

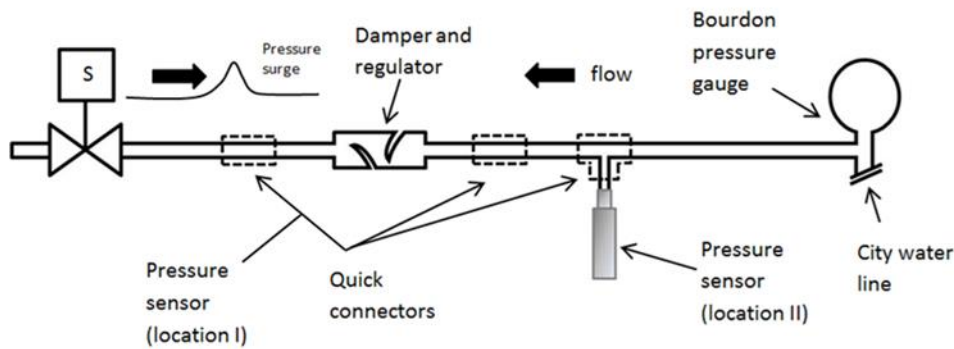


Figure 2. Physical model test configuration.

### 3.2. CFD Model

ANSYS CFX solver was preferred in this study for performing analysis to simulate the experimentally tested performance of the prototype. CFX is a highly capable software in many type of CFD analyses as an element-based finite volume solver. CFD method depends on solving governing fluid mechanics equations (continuity, momentum, energy, turbulence) numerically. Details of CFD theory and the equations will not be given here, however further details may be obtained in listed studies [23-25] and in ANSYS CFX Manual [26].

In order to determine the effect of a new damper design, a reference CFD simulation of the pipe without any damper design was performed. 4.5 m long straight pipe with a diameter of 6 mm was used for both physical and numerical model tests. The term “WD” will be used here to refer to the “without damper” case. Details of the numerical model domain geometry and boundary conditions are shown in Figure 3.

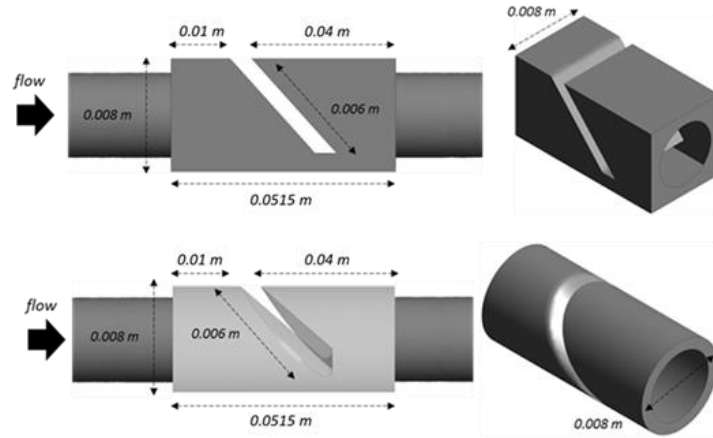


Figure 3. Geometric configuration of the solution domain.

Simulations were performed under the same configuration and conditions for both cases (with and without damper) to monitor the performance of damper devices. First, steady state simulations were carried out to help convergence and prepare stable initial conditions for time dependent solutions. Upstream of the pipe was defined as  $4 \times 10^5$  Pa (city water network pressure) pressurized inlet and the downstream was defined as mass flow-outlet with  $2.1 \times 10^{-5}$  m<sup>3</sup>/s discharge which is obtained from physical tests as the initial boundary conditions. The solid surfaces of the domain were defined as no

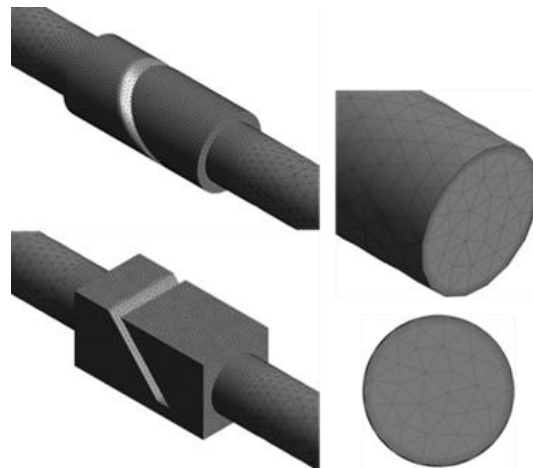
slip rough wall with appropriate roughness height to match that of the actual high density polyethylene pipes (Fig. 3). Steady-state simulations were completed with the given initial and boundary conditions and following transient analyses were performed to simulate the pressure surge. To control the closing time of the imaginary valve in CFD analysis, an expression was defined at the mass flow outlet as a function of time. Closing time was applied as 0.025 s to match the physical tests. In CFD simulations, time steps of 0.001 s was used to comply with the sampling rate of the pressure sensor used in the physical tests. High resolution advection scheme and second order backward Euler transient scheme were preferred to obtain more accurate solutions. Although compressibility of the water against high and low pressure waves being almost negligible, a variable density expression that changes according to pressure for water was defined and used in the whole domain.

After the simulation of the WD case, two different damper designs were simulated with single and double room cases. Damper designs were inspired from gun silencer concept which traps and absorbs sound waves by means of chambers. A box shaped damper designs with an inclined wall opposite to the pressure wave direction coming from downstream and a cylindrical damper design that consist a short conic shape interior wall were located in the pipeline 1.5 m away from the outlet. Throughout this paper, “BD1” and “CD1” terms will be used to refer to the box shaped and cylindrical damper designs with one room respectively. Details of the BD1 and CD1 geometries are presented in Figure 4.



**Figure 4.** Details of BD1 (top) and CD1 (bottom) geometries.

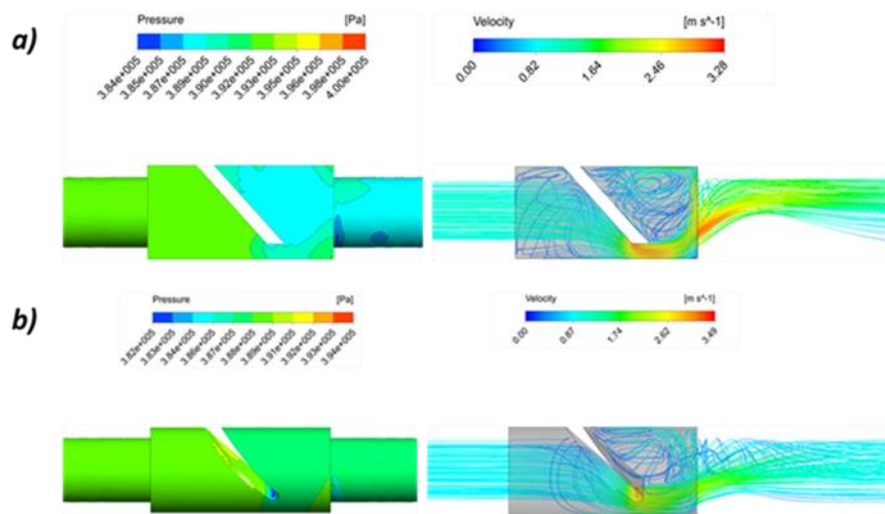
After the geometry of the BD1 and CD1 design were created, a proper mesh was generated for the solution domain. CFD simulation needs to meet certain mesh quality requirements such as geometrical shape and size of the mesh elements [27]. A preliminary mesh independency effort suggested that at least 1 million cells are needed for stable gross flow parameters. An average of 2.5 million mesh elements was used in each case to resolve accurately the complex geometry of damper design and the solution domain. The same configurations were used for all CFD simulations. Mesh element sizes were dictated not to exceed 1 mm. Refinement near the damper zone was applied to reduce the mesh size down to 0.5 mm to avoid low quality, skewed elements. Inflation layers were also applied near the solid surfaces to achieve accurate solutions near the viscous layer. Generated mesh for both BD1 and CD1 are given in Figure 5.



**Figure 5.** Mesh of the CD1 (top), BD1 (bottom) and the inflation at the walls.

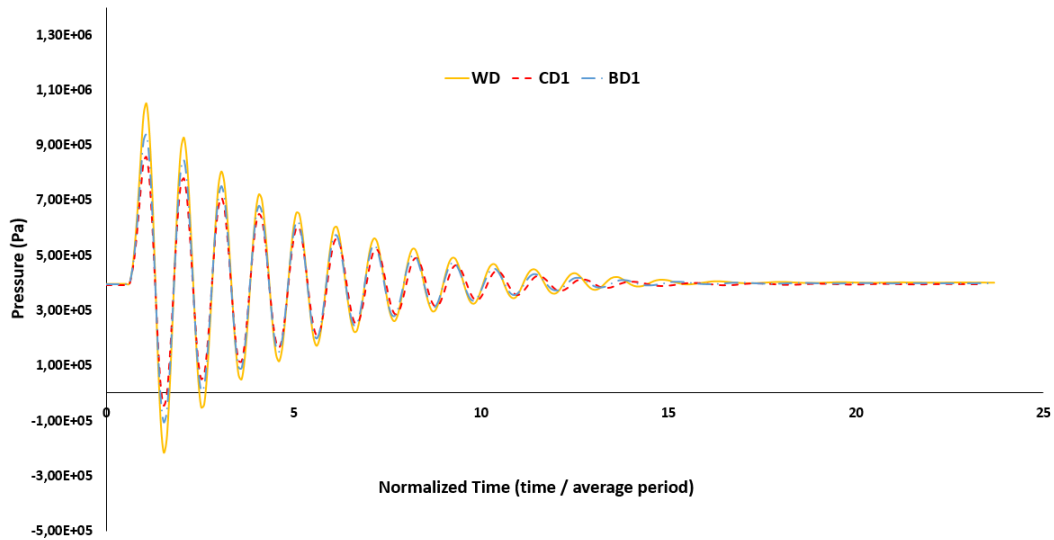
#### 4. RESULTS

Transient CFD solutions were continued until the pressure oscillations due to pressure surge settled down. As previously stated, both two damper designs were expected to trap and absorb the pressure waves and reduce the surge magnitude. In order to demonstrate the damper design performances, pressure contours and velocity streamlines were analyzed.



**Figure 6.** a) Pressure contours around BD1 and velocity streamlines, b) Pressure contours around CD1 and velocity streamlines.

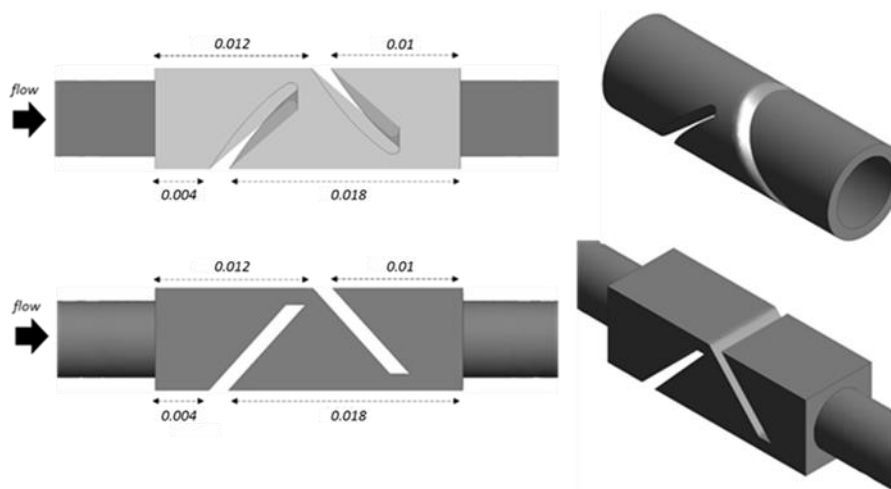
Both damper designs are breaking the energy of the flow while the valve is open and reducing the pressure just at the point where chambers are separated by damper wall (Figure 6 a-b). Streamlines show that the velocity of the flow increases due to the contraction and dead zones occur behind the inclined walls of dampers as expected. The local energy losses due to the damper in physical tests were observed to be negligible. Additionally, it should be noted that cavitation is not a significant concern for the water dispenser system presented here.



**Figure 7.** Pressure wave form at the midpoint of the upstream of pipe for WD, BD1 and CD1.

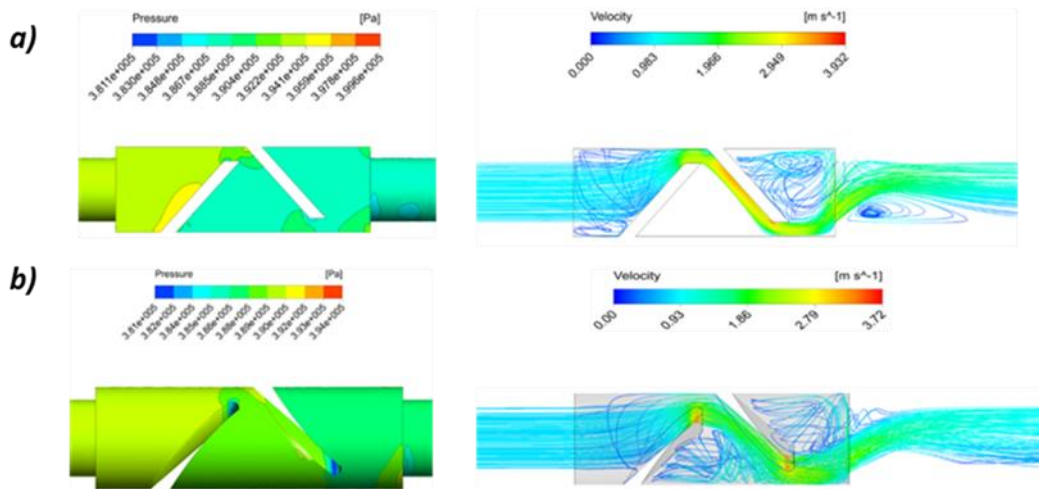
Magnitude and period of pressure waves induced by sudden valve closure is dominated by closing time of the valve, length and material of the pipe and the downstream pressure. However, as mentioned before, the set-up used here was built with the purpose of representing the refrigerator component, not an ideal benchmark case. Results of WD, BD1 and CD1 simulations are compared in terms of duration (average period: time between the detectible peaks which corresponds to 0.013 s for all experimental and numerical data) and the magnitude of the surge pressure at the midpoint of the pipes upstream section shown in Figure 7. Time axis is normalized by the average period of pressure oscillations to exclude pipe length effect which is directly related with the magnitude of periods. As expected, the pressure peaks are reduced for both BD1 and CD1 cases. In WD case, pressure peaks reach almost  $11 \times 10^5$  Pa. This value is reduced to  $9.4 \times 10^5$  Pa for the BD1 and gets even a lesser value of  $8.55 \times 10^5$  Pa for the CD1 case. This observation makes cylindrical design with conic wall a better choice for reducing the magnitude of the pressure surge.

Two additional CFD simulations were performed to understand the effect of the number of chambers for both BD1 and CD1. New damper designs consist of two staggered rooms and will be named as BD2 (box damper with two rooms) and CD2 (cylindrical damper with two rooms, Figure 8).



**Figure 8.** Details of BD2 (top) and CD2 (bottom) geometries (length in m).

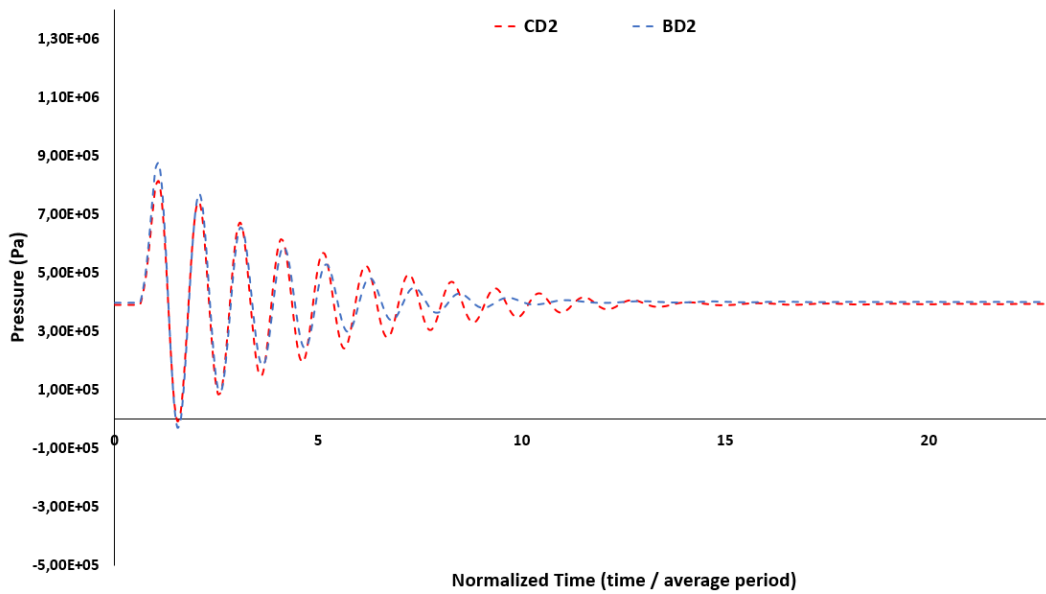
Although CD1 design has a better performance than the BD1 in first set of simulations, both are simulated for the case with two rooms to observe effect of staggered rooms clearly. Analyzing Figures 9 a) and b), it is observed that in BD2 case damping of pressure begins near the first wall of damper while this effect starts behind the second wall in CD2 case. This can be explained by the smooth contraction provided by conic shaped walls of CD2 design which is clearly shown in velocity streamline figures. The behavior of the flow around the devices is no indication to the damping performance. However, the flow regulation of the devices is also an important factor that needs to be evaluated along with the damping effects. This is particularly important for the comfort of the end user. If the continuous losses through the pipe system is neglected, a 3.6% local loss ratio is observed for the BD2 design and it is only 2.4% for the CD2 design as expected due to its geometry. What is interesting in this comparison is the higher performance of the CD2 design against pressure surge as shown in Figure 10 while breaking lesser flow energy.



**Figure 9.** a) Pressure contours at the walls for BD2 and velocity streamlines b) Pressure contours at the walls for CD2 and velocity streamlines.

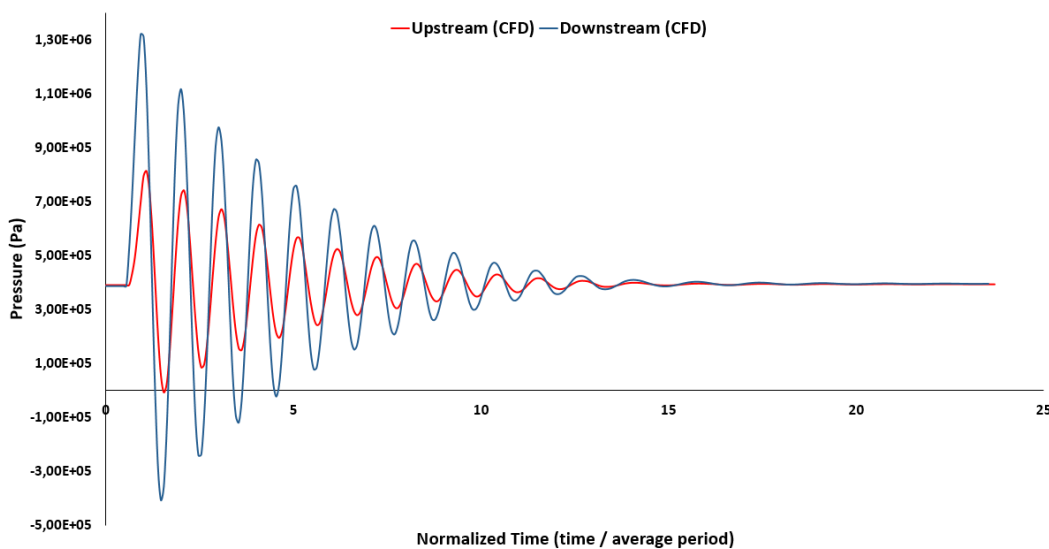
In Figure 10, pressure wave oscillations are compared between BD2 and CD2. As in the first comparison between BD1 and CD1, cylindrical design is superior to the box design in reducing pressure peak values. This corresponds to 7% improvement (reduction) in performance for box design and 4% for cylindrical design in maximum peak pressure value for the one room cases. Another point that should be considered here is the oscillation pattern of the BD2 and CD2. While CD2 provides lesser pressure peak magnitude, BD2 causes absorption of pressure oscillation in a shorter period of time.





**Figure 10.** Pressure oscillation at the midpoint of the upstream of pipe for BD2 and CD2.

CFD results indicate that the cylindrical damper design shows better performance in reducing the magnitude of the pressure waves travelling upstream. The finalized design consists of two staggered short conic shape walls in a short straight pipe section. Narrow mouths of the conics are located facing downstream of the pipe in order to trap and absorb (by momentarily travelling extra distance and hitting the walls) pressure surge in the room between staggered conics immediately after the valve is shut. To have a better assessment of the physics behind the damper performance, pressure waves before the damper (upstream) and after the damper (downstream) were also compared in Figure 11 for CD2 design.

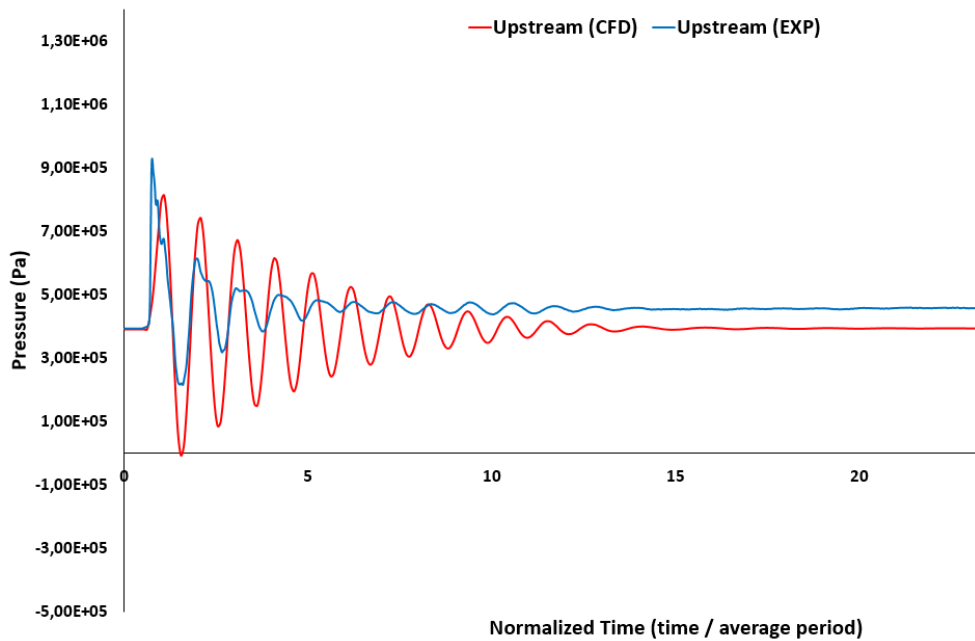


**Figure 11.** Pressure oscillation before the damper (upstream) and after the damper (downstream) for CD2.

Sudden closing of the valve is generating high pressure waves at the downstream and the maximum pressure peak reaches about  $13 \times 10^5$  Pa. The same peak was observed to be  $8.15 \times 10^5$  Pa, a drop of 37

% which is a significant reduction effort for protecting the refrigerator water dispenser system from pressure surge damage.

Figure 12 presents the comparison of the experimental and CFD pressure values at the midpoint of the pipe upstream results under  $4 \times 10^5$  Pa inlet pressure condition. In experimental results, maximum pressure peak value at the upstream point reaches to  $9.22 \times 10^5$  Pa and it is 11% higher than the results from the CFD model. Although verified, with proper validation, using benchmark set-ups rather than an industrial application, a much more accurate numerical representation can be achieved. The offset between the steady state pressure values in Fig. 12 (those after normalized time of 12) is an indication of solver capabilities of incompressible flow interaction with flexible solids.



**Figure 12.** Comparison of maximum pressure peak values at the midpoint of the upstream (with damper case) between CFD and experimental results

## 5. CONCLUSIONS AND DISCUSSION

The purpose of the current study was to investigate damaging effects of pressure surge phenomenon occurring in a refrigerators water dispenser system and develop a damper device design using 3D CFD simulations and also confirm the effectiveness by physical experiments. CFD model was particularly used to understand the characteristics of the flow in the dampers. Different model designs were considered according to the industrial application (producibility, size, and cost) limitations. Preliminary CFD runs were utilized to guide the design procedure. A plastic damper with a cylindrical shape consisting of two conic walls inside was chosen as the most effective design. Both numerical simulations and experiment results show that the damper device significantly reduces the pressure peaks and protects the refrigerator water dispenser system. The results also suggest that the gun silencer logic works well for pressure surge problems under the initial and boundary conditions described here. It should be noted that no cavitation risk was detected within the fixtures tested here (implied from CFD results).

Another point that should be considered is the role of CFD simulations in industrial product design processes. Although an accurate CFD model requires many physical parameters which should be adjusted diligently if a validation study is intended, for a more realistic numerical analysis, elastic deformation of the pipe wall due to pressure surge should be included by a two-way Fluid-Structure-

Interaction (FSI) approach. Otherwise energy of pressure wave will be reflected without any reduction due to deformation. Although CFD and experimental results do not perfectly agree in terms of magnitudes, CFD simulation provides a cost and time effective pre-design process before and after prototyping and physical test stage. It is also useful for demonstrating the physics behind the proof of concept. This is particularly important as many features of the flow reflecting the feasibility of the application may not be detected easily by physical experiments.

Both experimental and numerical results indicate that a very low cost, small size, manufacturable, single plastic body can reduce the high pressure waves and protect the refrigerator dispenser system from pressure surge effects. The presented work is part of an ongoing collaborative project and the products produced in the fashion described here are expected to reduce the cost of effective solutions to the pressure surge problem in home appliances.

## ACKNOWLEDGEMENTS

The authors thank Mr. Burhan Sahin for the valuable discussions on the design constraints presented in this study. The research grants no: 1702F052 and 1705F405 by Eskişehir Technical University is gratefully acknowledged.

## CONFLICT OF INTEREST

The authors stated that there are no conflicts of interest regarding the publication of this article.

## REFERENCES

- [1] Záruba J. Developments in Water Science: Water Hammer in Pipe-Line Systems. Elsevier Science, 1993.
- [2] Ghidaoui MS, Zhao M, McInnis DA, Axworthy DH. A review of water hammer theory and practice. Applied Mechanics Reviews, 2005; 58 (1) 49-76. doi: 10.1115/1.1828050
- [3] Bergant A, Simpson AR, Tijsseling AS. Water hammer with column separation: A historical review. Journal of Fluids and Structures., 2006; 22 (2) 135-171. doi: <https://doi.org/10.1016/j.jfluidstructs.2005.08.008>
- [4] Stephenson D. Simple Guide for Design of Air Vessels for Water Hammer Protection of Pumping Lines. J. Hydraul. Eng., 2002; 128(8) 792-797. doi: [https://doi.org/10.1061/\(ASCE\)0733-9429\(2002\)128:8\(792\)](https://doi.org/10.1061/(ASCE)0733-9429(2002)128:8(792))
- [5] Boulos PF, Karney BW, Wood DJ, Lingireddy S. Hydraulic Transient Guidelines for Protecting Water Distribution Systems. Journal American Water Works Association. 2005; 97 (5) 111-124. doi: <https://doi.org/10.1002/j.1551-8833.2005.tb10892.x>
- [6] Afshar MH, Rohani M. Water hammer simulation by implicit method of characteristic. Int. J. Press. Vessels Pip. 2008; 85 (12) 851–859. doi: <https://doi.org/10.1016/j.ijpvp.2008.08.006>
- [7] Wang C, Yang J. Water Hammer Simulation Using Explicit–Implicit Coupling Methods. J. Hydraul. Eng. 2015; 141(4) 1-12. doi: [https://doi.org/10.1061/\(ASCE\)HY.1943-7900.0000979](https://doi.org/10.1061/(ASCE)HY.1943-7900.0000979)

- [8] Trabelsi M, Triki A. Dual control technique for mitigating water-hammer phenomenon in pressurized steel-piping systems. *Int. J. Press. Vessels Pip.*, 2019; 172 397-413. Doi: <https://doi.org/10.1016/j.ijpvp.2019.04.011>
- [9] Yuzhanin V, Popadko V, Koturbash T, Chernova V, Barashkin R. Predictive control and suppression of pressure surges in main oil pipelines with counter-running pressure waves. *Int. J. Press. Vessels Pip.* 2019; 172 42-47. doi: <https://doi.org/10.1016/j.ijpvp.2019.03.015>
- [10] Lee CS, Lee KB, Lee CG. An experimental study on the control of pressure transients using an orifice. *Int. J. Press. Vessels Pip.*, 2001; 78 (5) 337-341. doi: [https://doi.org/10.1016/S0308-0161\(01\)00046-1](https://doi.org/10.1016/S0308-0161(01)00046-1)
- [11] Asiaban P, Fathi-Moghaddam M. Flow throttling in surge tanks using porous structures. *Int. J. Press. Vessels Pip.*, 2018;168 301-309. doi: <https://doi.org/10.1016/j.ijpvp.2018.11.009>
- [12] Jinping L, Peng W, Jiandong Y. CFD Numerical simulation of water hammer in pipeline based on the Navier-Stokes equation. V European Conference on Computational Fluid Dynamics, Lisbon, Portugal, 2008 June .
- [13] Al-Khomairi AM, Ead S. Sizing of A Plastic Chamber with Air-filled Balls for Water Hammer Control. *WIT Transactions on Engineering Sciences*, 2005; 69 311–318.
- [14] Al-Khomairi AM. Plastic water hammer damper. *Australian Journal of Civil Engineering*; 2010; 8 (1) 73-81.
- [15] Choon TW, Aik LK, Aik LE, Hin TT. Investigation of water hammer effect through pipeline system. *International Journal on Advanced Science Engineering Information Technology*, 2012; 2 (3) 48-53. doi: <http://dx.doi.org/10.18517/ijaseit.2.3.196>
- [16] Meniconi S, Brunone B, Ferrante M. Water-hammer pressure waves interaction at cross-section changes in series in viscoelastic pipes. *Journal of Fluids and Structures*, 2012; 33, 44-58. doi: <https://doi.org/10.1016/j.jfluidstructs.2012.05.007>
- [17] Nikpour MR, Nazemi AH, Hosseinzadeh DA, Shoja F, Varjavand P. Experimental and Numerical Simulation of Water Hammer. *Arabian Journal for Science and Engineering*, 2014; 39 (4) 2669–2675. doi: <https://doi.org/10.1007/s13369-013-0942-1>
- [18] Zhao M, Ghidaoui M. Godunov-type solutions for water hammer flows. *J. Hydraul. Eng. ASCE.*, 2004; 130 (4) 341–348. doi: [https://doi.org/10.1061/\(ASCE\)0733-9429\(2004\)130:4\(341\)](https://doi.org/10.1061/(ASCE)0733-9429(2004)130:4(341))
- [19] Riasi A, Nourbakhsh A, Raisee M. Unsteady turbulent pipe flow due to water hammer using  $k-\theta$  turbulence model. *J. Hydraul. Res.* 2009; 46 (4) 429–437. doi: <https://doi.org/10.1080/00221686.2009.9522018>
- [20] Riedelmeier S, Becker S, Schlücker E. Damping of water hammer oscillations—comparison of 3D CFD and 1D calculations using two selected models for pipe friction. *Proc. Appl. Math. Mech.*, 2014; 14 705–706. doi: <https://doi.org/10.1002/pamm.201410335>
- [21] Wu D, Yang S, Wu P, Wang L. MOC-CFD coupled approach for the analysis of the fluid dynamic interaction between water hammer and pump. *J. Hydraul. Eng.* 2015; 141 (6) 1-8. doi: [https://doi.org/10.1061/\(ASCE\)HY.1943-7900.0001008](https://doi.org/10.1061/(ASCE)HY.1943-7900.0001008)

- [22] Zhang X, Cheng Y, Xia L, Yang J. CFD simulation of reverse water-hammer induced by collapse of draft-tube cavity in a model pump-turbine during runaway process. IOP Conference Series: Earth and Environmental Science , 2016; 49 (5) 05 2017.
- [23] Carrillo JM, García JT, Castillo LG. Experimental and Numerical Modelling of Bottom Intake Racks with Circular Bars. *Water*, 2018; 10 (5) 605. doi: <https://doi.org/10.3390/w10050605>
- [24] Versteeg HK, Malalasekera W. *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. 2nd ed., Pearson Prentice Hall: Englewood Cliffs, NJ, USA, 2007.
- [25] Hirsch C. *Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics*. 2nd ed., Butterworth-Heinemann, Elsevier: Oxford, UK, 2007.
- [26] ANSYS Inc. *ANSYS CFX. Solver Theory Guide*. Release 14.0, ANSYS, Inc.: Southpointe, Canonsburg, PA, USA, 2011.
- [27] Martins NMC, Carrico NJG, Ramos HM, Covas DIC. Velocity-distribution in pressurized pipe flow using CFD: Accuracy and mesh analysis. *Computers & Fluids*. 2014; 105 218–230. doi: <https://doi.org/10.1016/j.compfluid.2014.09.031>