Experimental And Numerical Investigation Of The Flow Analysis Of The Water-Saving Safety Valve

Muhammed Safa Kamer, Ahmet Kaya, Abdullah Sisman

Abstract: In this study, an auto-mechanical safety valve was designed and manufactured in order to prevent possible wastage of water and water raid after instantaneous water cuts during water usage in places where water use is widespread. Safety valve is activated and it switches off the line when water is cut off (when mains pressure is equal to atmospheric pressure), and as it does not allow water to pass when it comes back, it saves water and prevents the formation of raids. An experiment set was conducted in order to measure the pressure drop between the inlet and outlet of the safety valve and it was found that with the increased flow rate the pressure drop increases. The three-dimensional flow analysis of the safety valve was carried out with Ansys-Fluent software package and the results obtained were compared with experimental data, and a good harmony was achieved.

Index Terms: Ansys-Fluent, Computational Fluid Dynamics, Experimental Study, Pressure Drop, Safety Valve.

1 INTRODUCTION

WATER requirement is rapidly increasing together with the growing population and booming economy. Using clean water sources without wasting is the biggest goal in managing water economically. One of the key factors that lead to wasted water resources is water losses in the network, and this is one of the important research topics in our country as well as in other countries [1]. In cases of interruption of mains water, open-forgotten valves cause water loss. To prevent water losses in the network and systems, to reduce water consumption in instruments where water is used, and to prevent water raids caused by various water leaks, various studies have been conducted from the past to the present. Kavurmacioglu and Karadogan [2] introduced pessure regulating valves and their characteristics, and identified some regulator characteristics of the pressure regulator valves in the laboratory settings. By adjusting pressure using automatic control valves, they determined that it is possible to reduce water leakage from all sizes of installations. Celikag [3] determined the pressure loss coefficient for ball valves experimentally and numerically. He used the Fluent program, a finite volume method, in numerical methods. Amirante et al. [4] made the flow analyzes using the Fluent software package in order to improve the performance of hydraulic proportional valve and make design improvements. Chern et al. [5] studied the performance of the ball valve, the flow characteristics and the cavitation effect experimentally. To determine the performance of valve, they determined necessary coefficients for different volumetric flow and different pressures. Duymaz [6] calculated the local head loss coefficient of a butterfly valve in the nominal diameter of DN 40 quantitatively and experimentally. She used the Fluent software package, a finite volume method, in numerical analysis. Koyunbaba [7] analyzed the flow characteristic curves of the ball and butterfly valves which are valve types widely used in industry with the help of ANSYS program and compared with catalog data of the valve producing companies. Pirincciler [8] designed a single body distribution unit used in the distribution of water in pressurized irrigation system and investigated flow analysis experimentally and numerically. Experimentally obtained pressure losses were compared with the results obtained from numerical methods (FLUENT). Yuksel [9] calculated pressure loss coefficient in gate valves depending on the valve openings experimentally and numerically. Kaya [10] investigated the performance of a centrifugal pump which is horizontal shaft, single-staged, and end-suctioned numerically. He used the FLUENT program, a finite volume method, in his numerical study. Deng et al. [11] determined the flow analysis and local temperature distribution of oil which flows at high speed, around the notches in a spool-valve by the Ansys-Fluent program based on the finite volume method and ABAQUS program. Sandalci [12] determined pressure losses of two different valves sized DN65 and DN80 at different flow rates and different opening angles (in disk angels) experimentally. Yang et al. [13] investigated the flow analysis of a 3-D stop valve for different volumetric flows numerically. By applying the RNG k-ε turbulence model, they solved the Navier-Stokes equations by the Fluent software package. Chattopadhyay et al. [14] studied the flow through the pressure control valve using computational fluid dynamics methods. For turbulence conditions, they solved the Navier-Stokes equations using the Fluent software packages. They used the standard k-ε and realizable k-ε models in numerical analysis. Lisowski and Rajda [15] studied the reduction of the pressure drop in a hydraulic system, using directional control valve, experimentally and numerically. In the experimental study, an experiment was set up and the pressure drop occurring in different volumetric flows was determined. In the numerical study, on the other hand, using the standard k-ε turbulence model in the Ansys-Fluent software package, analysis were made and they were compared with experimental study results. Kamer [16] designed a mechanical valve in order to save water. By doing flow analysis in this designed valve, he determined the pressure drop occurring at various flow rates, experimentally and numerically. In this study, in order to prevent possible waste and water raid with the arrival of the mains water from the taps forgotten open during instantaneous water cuts in public water

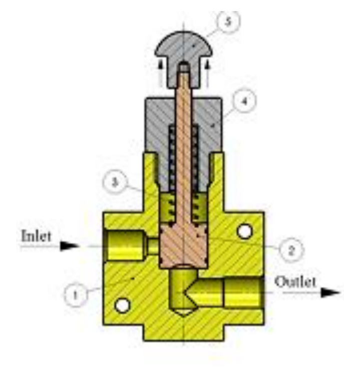
- *Muhammed Safa KAMER is PhD Student in Department of Mechanical Engineering in Kahramanmaras Sutcu Imam University, Turkey.*
- *E-mail:msafakamer@hotmail.com, msafakamer@ksu.edu.tr*
- *Ahmet KAYA is Assoc. Prof. in Department of Mechanical Engineering in Kahramanmaras Sutcu Imam University, Turkey. E-mail: kaya38@ksu.edu.tr*
- *Abdullah SISMAN is Assist. Prof. in Department of Mechanical Engineering in Kahramanmaras Sutcu Imam University, Turkey. E-mail: asisman@ksu.edu.tr*

consumption areas (hospitals, schools, etc.), a mechanically-automatically operated security valve was designed and manufactured. The flow properties (velocity and pressure distribution and pressure drop-mass flow relation) of this designed and manufactured safety valve were examined experimentally and numerically.

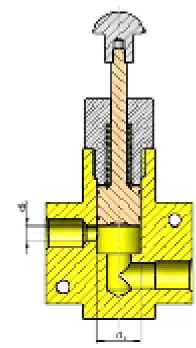
2 MATERYAL AND METHODS

2.1 Experimental Study

In order to prevent possible waste and water raid with the arrival of the mains water from the taps forgotten open during instantaneous water cuts by mechanically- automatically operated mechanisms, a new valve (Hand Operated Safety Valve) was designed and manufactured. The Hand Operated Safety Valve consists of, as shown in Figure 1, the outer body (1), the switching slider (2), the closing spring (3), the apertured sliding plug (4) and the opening knob (5). In the first case, as the switching slider (2) is in the switched position due to the force of the closing spring (3), even if there is tap water pressure in water supply installations, the Hand Operated Safety Valve does not allow the water to pass to the 'Exit' line. In cases when there is tap water pressure in water supply installations, the release knob (5) is pulled by manpower in the direction shown in Figure 1a, and the switching slide (3) is switched on and left as such. Thus, water passage is supplied to the outlet line of the Hand Operated Safety Valve. The force of the switching slider (2) of the closing spring (3) in the open position is designed in a way that it can not defeat the force formed by the minimum supply pressure which affects the switching slider (2) (the pressure affecting the switching slider (2) in cases when last user valve / valves are completely open). Therefore, in cases where there is mains pressure in the mains water installations, once the Hand Operated Safety Valve is switched on, it does not get into the off position until the mains water is cut off. In addition, even if the mains water is cut off, in cases when any user valves are not open, the valve still does not go off because in order for the valve to go off, in addition to the removal of the pressure affecting the switching slider (2), the water in the valve should be drained and it should not prevent the movement of the switching slide (2) towards the switched position. When mains water is cut off, as the mains pressure affecting switching slider (2) is removed, under the influence of forces of the closing spring (3) the switching slide (2) automatically goes off (For this, it requires that any user valve is open or the water in the valve should be drained somehow). At this stage, if the switching slider (2) is desired to be turned on by pulling the release knob (5) by manpower in the direction shown in Figure 1a, since there is no system pressure in water supply installations, again under the influence of forces of the closing spring (3) the switching slider (2) automatically goes off. In the event that mains pressure again comes to the mains water systems, until the switching slider (2) is switched on by pulling the release knob (5) in the direction shown in Figure 1a by manpower, the Hand Operated Safety Valve does not allow the passage of water to the 'Exit' line. Thus, in case of water supply interruption, even if the water valve / valves on the installations after the Hand Operated Safety Valve are forgotten open, no water flows from this valve / these valves if the tap water comes again.



Close position of the valve (a)



Open position of the valve (b)

Figure 1. *Cross-Sectional Views of Hand Operated Safety Valve*

When it is realized that there is no flow of water from water valve / valves, the Hand Operated Safety Valve is set to open with the help of manpower and so water passage is ensured to the last user valves. After water passage is ensured to the last user valves, it is noticed that the water flows from the valves left open and these valves are switched off. This valve which is designed and manufactured is shown in Figure 2. An experiment was designed in order to test this improved hand-operated safety valve and determine the pressure drop rates at different flows (Figure 3). The designed experimental set principally consists of 100 liter tank, circulation pump, closed expansion tank, pressure regulator, ball valve, electromagnetic flowmeter, digital pressure gauge, safety valve, long faucets and electrical panel elements.

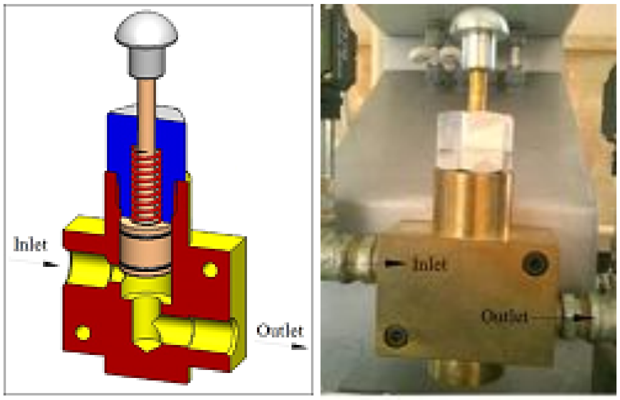


Figure 2. *The Designed Hand Operated Safety Valve*

The clean water filled in the tank is sent to the system through the circulation pump system. The clean water reaches the long tap touring the installation elements and is transferred to the tank from the tap again. The system operation continues in closed cycle in this manner. The clean water sucked from the tank with the aid of the circulation pump is divided into two paths at the pump outlet. In the first one of these outlets, the clean water reaches the long faucet after touring various installation elements. In the second outlet, on the other hand, the clean water is transferred to the warehouse after it tours the safety valve and a variety of installation elements. The clean water following the first outlet, here, first enters the pressure regulator. The clean water coming out of the the pressure regulator enters the ball valve in order to get the flow adjustment.

1	100 liter tank
2	Circulation pump
3	Closed expansion tank
4	Pressure regulator
5	Ball valve
6	Electromagnetic flowmeter
7	Digital pressure gauge
8	Safety valve
9	Hand-operated safety valve
10	Long tap
11	Electrical panel

Figure 3. *The Schematic Diagram of The Experiment Set Used in Pressure Drop Measurement*

The flow measurement of the clean water coming out of ball valves is made by electromagnetic flowmetre. The clean water which goes through electromagnetic flowmeter enters the Hand Operated Safety Valve (Figure 2). The clean water coming out of the Hand Operated Safety Valve first enters the long tap and the water coming out of it flows back to the tank. A digital pressure gauge is placed to the inlet and outlet of the Hand Operated Safety Valve to measure pressure in these parts. In the event that the long tap on the system is closed or the flow transferred to the first outlet is reduced through the ball valve, to prevent damage to the pump, clean water is directed to the second outlet wherein it opens the safety valve and reaches the warehouse.

2.2 Numerical Study

Within the scope of numerical study, the three-dimensional solid model of the analysis area was modeled in the computer environment by taking measures on the experiment set. The drawings were performed considering the open position of the Hand Operated Safety Valve. In computer environments, in three-dimensional solid modeling operations the SolidWorks commercial software was used and in the flow analyses the Ansys-Fluent commercial software was used.The assumptions made in the numerical calculations;

- In the valve inlet-outlet, steel pipes and iron fittings materials were used and the roughness values of these materials are taken as 1.6 µm as a fixed value for steel pipes and 5.12 µm for iron fittings materials. Valve is, on the other hand, made of bronze and its roughness value is taken as 3.2 µm as a fixed value [17].
- Numerical calculations, for turbulent, three-dimensional, continuous, forced convection, are achieved by using Ansys Fluent computer program which is often used in CFD applications. In solutions, the segregated solver and the SIMPLE algorithm was used [18]. Continuity and momentum conservation equations [11];

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{u}\right) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}\left(\rho \vec{u}\right) + \nabla \cdot \left(\rho \vec{u}\vec{u}\right) = -\nabla p + \nabla \cdot \left(\vec{\tau}\right) \tag{2}$$

Here, p is static pressure; $\vec{\tau}$ is expansion tensor.

$$\vec{\tau} = \mu\left[\left(\nabla \vec{u} + \nabla \vec{u}^{T}\right) - \frac{2}{3}\nabla \cdot \vec{u} I\right] \tag{3}$$

Here, µ is the molecular viscosity and is used as the effective viscosity in the turbulent flow ($\mu_{eff} = \mu + \mu_t$, here μ_t is the turbulent viscosity). For three-dimensional numerical analysis, the following boundary conditions have been identified;

i. Inlet boundary condition: "mass-flow-inlet" definitions are made, and mass flow rates are taken in the range from 0.2149 kg/s to 0.2915 kg/s.
ii. Outlet boundary condition: defined as "pressure-outlet". The experimentally determined numerical value of the valve outlet pressure is entered.
iii. Pipe and valve walls are defined as "wall".

In numerical analysis, Standard k-ε and realizable k-ε turbulence models are used. In k-ε models, for the turbulent kinetic energy (k) and the rate of loss "dissipation" (ε), two transport equations are solved in addition to the Navier-Stokes equations. In standard k-ε turbulence model; the turbulence kinetic energy (k) and loss rate (ε) are solved using the following equations [11; 15; 19].

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon \tag{4}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}G_k-C_{2e}\rho\frac{\varepsilon^2}{k} \quad (5)$$

In these equations, G_k is the turbulence kinetic energy production caused by the average rate gradient. μ_t is the viscosity of turbulence and is determined by the equation of $\mu_t=\left(\rho C_\mu k^2\right)/\varepsilon$. Constants in turbulence model are given below [11; 15; 20]. The coefficient C_μ which is fixed in the "Standard k-ε" model takes a dynamic form in the realizable k-ε turbulence model and is obtained by the equation of $C_\mu=1/\left[A_0+\left(A_s\,k\,S_{ij}/\varepsilon\right)\right]$ [19].

Here; $A_0=4.04$, $A_S=\sqrt{6\cos(\phi)}$ and $\phi=\frac{1}{3}\cos^{-1}\left(\sqrt{6W}\right)$ are constants. The deformation tensor S_{ij} is calculated as follows:

$$S_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \quad (6)$$

The relation between the term W and S_{ij} is as [14];

$$W=\frac{S_{ij}S_{ij}}{\sqrt{S_{ij}S_{ij}}} \quad (7)$$

In the realizable k-ε turbulence model, to determine the turbulent kinetic energy (k) and loss rate (ε) the following equations are used.

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k-\rho\varepsilon \quad (8)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_j)}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+\rho C_1 S_\varepsilon-\rho C_2\frac{\varepsilon^2}{k+\sqrt{v\varepsilon}} \quad (9)$$

The constants of the "Realizable k-ε" turbulence model have the following values [20].

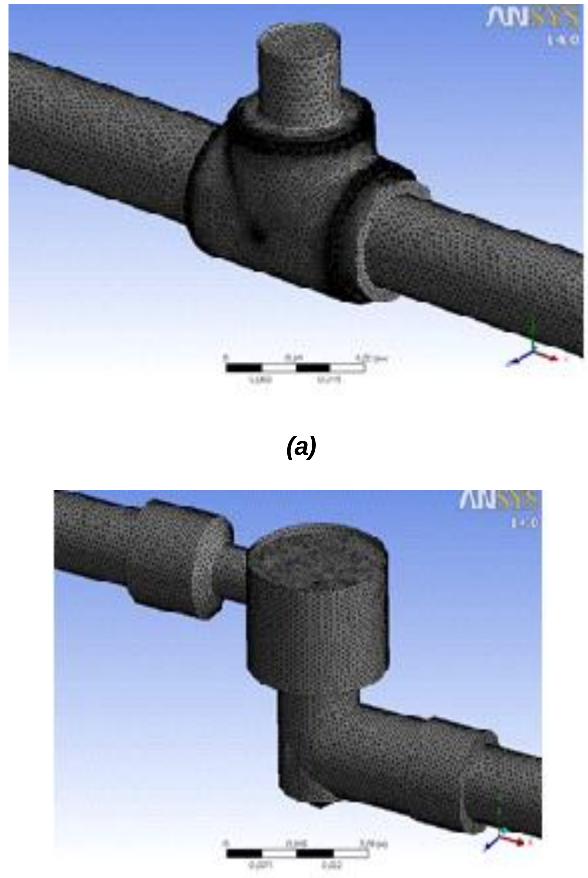
$$C_2=1.9 \;,\; \sigma_k=1.0 \;,\; \sigma_\varepsilon=1.2$$

2.3 Mesh Structure

In the quantitative solution of the Hand-operated safety valve, the network structure shown in Figure 4 was used. A heavy network was used to get a better view of the speed and pressure changes at the corner points. For the valve solution area, the network structure numbered $1.25\text{x}10^6$ was used.

2.4 Error Analysis

In experimental studies, it is important to determine the number of errors which affect the accuracy of the results. In experiments carried out in an experiment setting established in accordance with the standards specified in the literature, when the data obtained are evaluated, errors arise in two different ways. They are the inevitable errors arising from the structure of the experimental set and measurement instruments and errors caused by the negligence of the person making the experiment. The error analysis made for the evaluation of these errors are very important for interpretation of results [21]. After a certain number of experiments, to determine the error rates of these experiments several methods have been developed in practice. One of the most widely used of these is the "rational approach" method. In this type of error analysis, it is accepted that all instruments in measuring system make maximum errors at the same time [22]. Considering this method, the analysis of the error between the experimentally and quantitatively calculated pressure drop (ΔP) results can be performed. Within the scope of experimental study, the pressure drop values (ΔP) were calculated by subtracting the pressure values measured in the output line (P_O) from the pressure values separately measured in the Hand Operated Safety Valve inlet line (P_i) for each experiment. The measurement precisions (w_B) of digital pressure gauges used in the measurement of these pressures is ±%0.5, and these pressure gauges can measure only as the bar unit. Assuming that the pressure gauges make the maximum and minimum errors at the inlet and outlet at the same time, for each experiment the maximum pressure drop (ΔP_{max}) and the minimum pressure drop (ΔP_{min}) values were calculated taking into account the sensitivity of the measuring instruments. In calculations (11) and (12) numbered equations were used.



(a)

(b)

Figure 4. *Network Structure Used in Numerical Analysis*

$$\Delta P=P_i-P_o \quad (10)$$

$$\Delta P_{max}=\left[P_i+\left(P_i\cdot w_B\right)\right]-\left[P_o-\left(P_o\cdot w_B\right)\right] \quad (11)$$

$$\Delta P_{min}=\left[P_i-\left(P_i\cdot w_B\right)\right]-\left[P_o+\left(P_o\cdot w_B\right)\right] \quad (12)$$

3 Results and Discussion

The purpose of this study is to design and manufacture a mechanically- automatically operated security valve in order to prevent possible waste and water raid with the arrival of the mains water from the taps forgotten open during instantaneous water cuts in public water consumption areas (hospitals, schools, etc.). The flow properties (velocity and pressure distribution and pressure drop-mass flow relation) of this safety valve which is designed and manufactured were examined experimentally and numerically.

3.1 Analysis of Experimental Data

Within the scope of experimental study, before taking the experimental data, for the system to reach equilibrium, the electromagnetic flow meter, digital pressure gauges and circulation pump were run, and the Hand Operated Safety Valve and the long tap were set open. All experiments were performed while the system was in this way. After flow adjustment, by waiting at least 20 minutes for each experiment it is ensured that the system reaches the regime. Each experiment was repeated three times and the average values were taken. In the experimental study, pressure drop changes were determined in six different flow rates. Using the experimental results, the valve pressure drop (ΔP) and the ratio (%ΔP_0) of the pressure drop (ΔP) to the inlet pressure (P_i) were calculated for 6 different flow rates and the results obtained are given in Table 1.

$$\%\Delta P_0 = \left(\Delta P / P_i\right) \cdot 100 \qquad (13)$$

As the mass flow of fluid passing through the valve increases, the pressure drop that occurs between the inlet and outlet also increases. It is known that this increase is caused by cases, such as pipe surface roughness, friction, and sudden contraction-expansion. When Table 1 is analyzed, it is seen that the pressure drop rates (%ΔP_0) of all experiments appear to be very close to each other. That these rates are close to each other shows that mass flow ($\dot{m}$) change and pressure drop (ΔP) change are almost linear. By applying error analysis to experimental results, it is determined that measured values can vary at which intervals and they are shown in Table 2.

3.2 Analysis of Numerical Results

Within the scope of numerical study, the flow analysis of the valve whose three- dimensional solid model was created was carried out by the Ansys-Fluent software package. To be used in numerical analysis, some features of the fluid (water) were identified. The thermodynamic properties of water circulating in the experimental set were presented in Table 3 according to the the temperature values during the experiments and these features were used in numerical studies. Reynolds number was calculated according to the maximum cross section (d_p given in Figure 1.b) and the minimum cross sections (d_G given in Figure 1.b) of the flow area and it was determined that the flow is turbulent in all circumstances. In the numerical study conducted, the used volume element sizes (the distance between two nodes) have a considerable effect on the results. Therefore, elements network infrastructure must be small enough to give accurate results. In this study, for both used turbulence models the effect of the number of cells in the network structure on the experimental results were examined and given in Table 4. According to the examination conducted, in the event that the cell number in network structure is 1.250.000, it was observed that the results obtained are very close to the results of the experimental results and in the realizable k-ε turbulence model, the error rate is much lower (0.6%) and this turbulence model was used in the studies.

***Table 1**. Mass Flow Change of The Pressure Drop that Occurs in The Valve Input-Output*

Experiment No	Temperature	Mass Flow	Inlet Pressure	Outlet Pressure	Pressure Drop	
	***T* [K]**	**$\dot{m}$ [kg/s]**	**P_i [Pa]**	**P_o [Pa]**	**ΔP [Pa]**	**%ΔP_0 [%]**
1	301.15	0.2915	185000	174000	11000	5.946
2	303.15	0.2816	173000	163000	10000	5.780
3	304.15	0.2663	156000	147000	9000	5.769
4	305.15	0.2483	136000	128000	8000	5.882
5	306.15	0.2315	118000	111000	7000	5.932
6	307.15	0.2149	102000	96000	6000	5.882

***Table 2.** Error Analysis-Applied Experimental Results*

Experiment No	Mass Flow	Inlet Pressure	Outlet Pressure	Maximum Pressure Drop	Measured Pressure Drop	Minimum Pressure Drop
	$\dot{m}$ [kg/s]	**P_G [Pa]**	**$P_Ç$ [Pa]**	**ΔP_{max} [Pa]**	**ΔP [Pa]**	**ΔP_{min} [Pa]**
1	0.2915	185000	174000	**12795**	11000	**9205**
2	0.2816	173000	163000	**11680**	10000	**8320**
3	0.2663	156000	147000	**10515**	9000	**7485**
4	0.2483	136000	128000	**9320**	8000	**6680**
5	0.2315	118000	111000	**8145**	7000	**5855**
6	0.2149	102000	96000	**6990**	6000	**5010**

Numerical studies were conducted using six different experimental data, and the solutions obtained by two different turbulence models were compared with experimental data and shown in Table 5. It was found that the results obtained using realizable k-ε turbulence model are in good agreement with experimental data. Therefore, by using the results of the numerical analysis conducted by the realizable k-ε turbulence model, pressure drops (ΔP) and the pressure drop rates ($\%\Delta P_0$) were calculated. Pressure drops (ΔP) varying according to different mass flow ($\dot{m}$) are obtained numerically and they are shown in Figure 5 by comparing them to the experimental results. Obtained experimental and numerical results, the change in mass flow rate-pressure drop was found to be quite compatible. It was determined that the numerically obtained results are among the data obtained by applying error analysis to the experimental data. With the increase in mass flow, it was determined that the pressure drop occurring between the inlet and outlet of the safety valve increases, and this increase was found to be almost linear. Similar changes were also found in the studies conducted for different valve mechanisms in literature [6; 12; 14; 15; 23]. That the experimental and numerical results are compatible showed that when experimental system was established in this and other similar studies, complex problems that may create high costs can be solved by Ansys-Fluent software package at a minimal cost and in a short time

. Table 3. Properties of Saturated Water [24]

Experiment No	Temperature	Density	Specific Heat	Heat Conduction Coefficient	Dynamic Viscosity	Kinematic Viscosity	Reynolds Number	
	T [K]	ρ [kg/m³]	C_P [J/kg.K]	*K* [W/m.K]	*μ* [kg/m.s]	*υ* x 10^{-7} [m²/s]	Minimum Diameter, d_G	Maximum Diameter, d_p
1	301.15	996.400	4178.800	0.612	0.000835	8.38218	44460.917	14820.306
2	303.15	996.000	4178.000	0.615	0.000798	8.01205	44953.147	14984.382
3	304.15	995.600	4178.000	0.617	0.000782	7.85858	43358.342	14452.781
4	305.15	995.200	4178.000	0.618	0.000767	7.70498	41250.096	13750.032
5	306.15	994.800	4178.000	0.620	0.000751	7.55127	39257.782	13085.927
6	307.15	994.400	4178.000	0.621	0.000736	7.39743	37215.601	12405.200

Table 4. *The Impact of The Number of Cell Elements on The Solution*

Analysis No	Number of elements	Mass Flow $\dot{m}$	Inlet Pressure, P_i			Error Rate	
			Experimental	Numerical			
				Standart *k-ε*	Realizable *k-ε*	Standart *k-ε*	Realizable *k-ε*
		kg/s	Pa	Pa	Pa	%	%
1	600,000			190227	190084	2.748	2.675
2	700,000			186681	186398	0.900	0.750
3	800,000	0.2915	185000	186878	186425	1.005	0.764
4	1,000,000			187054	186518	1.098	0.814
5	**1,250,000**			**186374**	**185258**	**0.737**	**0.600**

Table 5. *Analysis of The Quantitative Results with Different Turbulence Models*

Experiment – Analysis No	Inlet Pressure, P_i			Error Rate	
	Experimental	Sayısal			
		Standart *k - ε*	Realizable *k - ε*	Standart *k - ε*	Realizable *k - ε*
	Pa	Pa	Pa	%	%
1	185000	186702	186116	0.912	0.600
2	173000	175081	174155	1.189	0.663
3	156000	157803	156779	1.143	0.497
4	136000	137262	136600	0.919	0.439
5	118000	118699	118448	0.589	0.378
6	102000	102714	102472	0.695	0.461

The fluid in the system is pressurized with the help of circulation pump. As the fluid moves forward in the system, it will be subjected to pressure drop due to losses, such as friction, valves and fittings and so on. Therefore, it is expected that the pressure at the inlet valve (P_i) is more than the pressure at the outlet valve (P_O). Pressure distribution in the flow is shown in Figure 6, and that the pressure in the inlet region is higher is identified. It is expected that there is a sudden contraction and the pressure is expected to increase in the regions where the flow splashes (Figure 6). Velocity distribution was obtained in the flow area and it was identified that the speed increases in the narrowest sections (Figure 7).

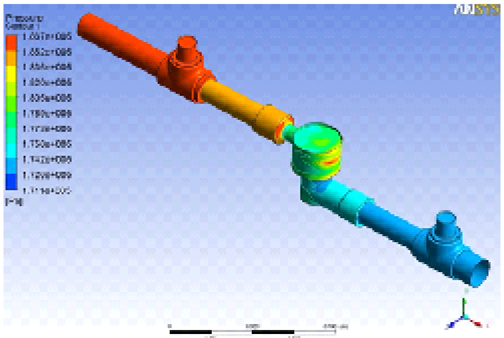


Figure 6. *Pressure Distribution of Flow Volume*

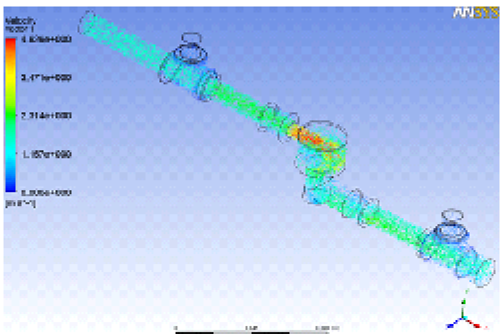


Figure 7. *Velocity Vectors of Flow Volume*

4 Conclusion

In this study, in order to prevent possible waste and water raid with the arrival of the mains water from the taps forgotten open during instantaneous water cuts in public water consumption areas (hospitals, schools, etc.), a mechanically- automatically operated security valve was designed and manufactured. The flow properties (velocity and pressure distribution and pressure drop-mass flow relation) of this safety valve which is designed and manufactured were examined experimentally and numerically. In order to determine the pressure drop in the safety valve, an experiment set was designed and pressure changes occurring in different mass flows were determined. Within the scope of the quantitative study, on the other hand, the safety valve was modeled as three-dimensional and the pressure drop between the inlet and outlet was determined by the Ansys Fluent program for each flow value. Experimental and numerical results obtained were found to be compatible with each other. The increase in the mass flow increased pressure drop which occurs in the safety valve, and this increase was determined to behave close to linear. Also, the fact that the results obtained from the numerical analysis program are compatible with the experimental data has shown that the numerical analysis can be used effectively in this type of studies.

Nomenclature

A_0, A_s : turbulence model constants
C_p : specific heat
C_1 , C_2 , $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ : turbulence model constants
D : characteristic length of the geometry, diameter for circular pipes (m)
d_G : cross-sectional diameter of the valve inlet line (minimum cross-sectional) (m)
d_p : inner diameter of the piston (maximum cross-sectional) (m)
G_k : turbulent kinetic energy production ($kg/m{\cdot}s^3$)
K : heat conduction coefficient (W/m·K)
k : turbulent kinetic energy (m^2/s^2)
$\dot{m}$: mass flow rate (kg/s)
P_o : outlet pressure (Pa)
P_i : inlet pressure (Pa)
Re : Reynolds number
S_ε : source term for ε (m^2/s^4)
T : temperature (K)
t : time (s)
u : velocity component in the horizontal direction (m/s)
υ : kinematic viscosity of the fluid (m^2/s)
V_{ort} : average flow velocity (m/s)
w_B : precision digital pressure gauge (%)
x : horizontal coordinate (m)
ΔP : pressure drop (Pa)
ΔP_{max} : maximum pressure drop (Pa)
ΔP_{min} : minimum pressure drop (Pa)
% ΔP_0 : pressure drop rate (%)
ε : turbulence dissipation (loss) rate (m^2/s^3)
μ : dynamic viscosity (kg/m·s)
μt : turbulence viscosity (kg/m·s)
ρ : density (kg/m^3)
σ_k : turbulent Prandtl number for k
σ_ε : turbulent Prandtl number for ε

Acknowledgment

The patent application related to the valve which is the subject of this study was supported by TUBITAK as ARDEB project No. 2013/1813 within the scope of TUBITAK 1008 - Patent Application Promotion and Support Programme.

References

[1] Pala, B., Latifoglu, A., “Water Losses Occurring in Water Distribution Networks - Case of Kayseri Province”, V. National Environmental Engineering Congress (in Turkish), Ankara, pp.1-7, 2003.

[2] Kavurmacioglu, L., Karadogan, H., “Water Leakage Reduction of the Installations Using Automatic Control Valves”, VI. National Installation Engineering Congress and Exhibition, Izmir, pp.635-644, 2003.

[3] Celikag, B., “Determining Valve Loss Factor in Ball Valves Experimentally and by Finite Volume Method”, Master’s Thesis (in Turkish), Institute of Science, Ege University, Izmir, p. 119, 2004.

[4] Amirante, R., Moscatelli, P.G., Catalano, L.A., “Evaluation Of The Flow Forces On A Direct (Single Stage) Proportional Valve By Means Of A Computational Fluid Dynamic Analysis”, Energy Conversion and Management 48, pp. 942-953, 2007.

[5] Chern, M.J., Wang, C.C., Ma, C.H., “Performance Test And Flow Visualization Of Ball Valve”, Experimental Thermal and Fluid Science 31, pp. 505-512, 2007.

[6] Duymaz, B.A., “Determining Valve Loss Factor in Butterfly Valves Experimentally and by Finite Volume Method”, Master’s Thesis (in Turkish), Institute of Science, Ege University, Izmir, p.103, 2008.

[7] Koyunbaba, E., “Determination of Flow Characteristics of the Valves by Computational Fluid Dynamics Applications”, Master's Thesis (in Turkish), Dokuz Eylul University, Institute of Science and Technology, Izmir, p.74, 2008.

[8] Pirincciler, M., “Valve Design with Hydraulic Remote Control”, Master’s Thesis (in Turkish), Ege University, Institute of Science and Technology, Izmir, p.81, 2008.

[9] Yuksel, B., “Determining Valve Loss Factor in Gate Valves Experimentally and by Finite Volume Method”, Master's Thesis (in Turkish), Ege University Institute of Science and Technology, Izmir, p.97, 2008.

[10] Kaya, M., “Numerical Analysis of Centrifugal Pump Performance”, Master’s Thesis (in Turkish), Istanbul Technical University, Institute of Energy, Istanbul, p.75, 2009.

[11] Deng, J., Shao, X.M., Fu, X., Zheng, Y., “Evaluation Of The Viscous Heating Induced Jam Fault Of Valve Spool By Fluid–Structure Coupled Simulations”, Energy Conversion and Management 50, pp. 947-954, 2009.

[12] Sandalci, M., “Experimental Investigation of the Performance Coefficient of Butterfly Valves”, Master’s Thesis (in Turkish), Marmara University Institute of Science, Istanbul, p.52, 2009.

[13] Yang, Q., Zhang, Z., Liu, M., Hu, J., “Numerical Simulation Of Fluid Flow Inside The Valve”, Procedia Engineering 23, pp. 543-550, 2011.

[14] Chattopadhyay, H., Kundu, A., Saha, B.K., Gangopadhyay, T., “Analysis Of Flow Structure Inside A Spool Type Pressure Regulating Valve”, Energy Conversion and Management 53, pp. 196-204, 2012.

[15] Lisowski, E., Rajda, J., “CFD Analysis Of Pressure Loss During Flow By Hydraulic Directional Control Valve Constructed From Logic Valves”, Energy Conversion and Management 65, pp. 285-291, 2013.

[16] Kamer, M.S., “The Method of Preventing the Water Loss from the Valves Forgotten Open during the Cut Off of the Mains Water”, Master’s Thesis (in Turkish), Institute of Science, Sutcu Imam University, Kahramanmaras, p.86, 2014.

[17] Bagci, M., “Technical Drawing”, Birsen Publishing Company (in Turkish), ISBN: 975-511-128-X, Istanbul, p. 286, 1998.

[18] Kahraman, N., Sekmen, U., Ceper, B., Akansu, S.O., “Quantitative Investigation of the Effects of Turbulators in Pipe Flows to the Heat Transfer”, Journal of Thermal Science and Technology 28(2), pp. 51-59, 2008.

[19] Simsek, O., “Experimental and Theoretical Analysis of Open Channel Flow Passing over Curved Large Cap Weir”, Master's Thesis (in Turkish), Cukurova University Institute of Science and Technology, Adana, p.109, 2011.

[20] Ansys Fluent 14.0 Theory Guide 2011, “Ansys Inc.”, http://cdlab2.fluid.tuwien.ac.at/LEHRE/TURB/Fluent.Inc/v140/flu_th.pdf. (URL Accessed on: 28.02.2014).

[21] Kaya, A., “Theoretical and Experimental Investigation of Heat and Mass Transfer in Drying”, PhD Thesis (in Turkish), Karadeniz Technical University, Institute of Science and Technology, Trabzon, p.153, 2008.

[22] Genceli, O.F., “Measurement Techniques”, Birsen Publishing Company (in Turkish), ISBN: 978-975-511-113-1, Istanbul, p.387, 2008.

[23] Winfield, D., Cross, M., Croft, N., Paddison, D., Craig, I., “Performance Comparison Of A Single And Triple Tangential Inlet Gas Separation Cyclone: A CFD Study”, Powder Technology 235, pp. 520-531, 2013.

[24] Cengel, Y.A., Cimbala, J.M., “Fluid Mechanics Fundamentals and Applications”, Izmir Guven Bookshop Limited (in Turkish), ISBN: 978-975-6240-18-2, Izmir, p. 938, 2012.