Mansoura Engineering Journal

Volume 48 | Issue 3 Article 2

2023

Computational Fluid Dynamics Investigation of Flow through Pneumatic Control Valve

Wael Elmayyah
Department of Mechatronics, Military Technical College, Cairo, Egypt, wael.elmayyah@mtc.edu.eg

Recommended Citation

Elmayyah, Wael (2023) "Computational Fluid Dynamics Investigation of Flow through Pneumatic Control Valve," *Mansoura Engineering Journal*: Vol. 48 : Iss. 3 , Article 2.
Available at: https://doi.org/10.58491/2735-4202.3042

ORIGINAL STUDY

Computational Fluid Dynamics Investigation of Flow Through Pneumatic Control Valve

Wael Elmayyah

Department of Mechatronics, Military Technical College, Egyptian Ministry of Defense, Cairo, Egypt

Abstract

Low-cost on-off pneumatic directional control valves are widely used with digital control circuits to control the position or the speed of pneumatic actuators. These valves have a limited flow capacity that hinders the system fast response. Therefore, deeper understanding of the internal air flow through the valve and its interaction with valve geometry will allow further performance improvement according to the required application.

In this paper, a Computational Fluid Dynamics (CFD) model for a low-cost internal pilot, electrically operated pneumatic 3/2 directional control valve has been developed to investigate the effect of the valve's geometry on the valve's outlet flow rate. The flow behaviour and the critical flow areas have been discussed. The computational model has been validated by comparing the predicted results by published experimental results in addition to results obtained by an analytical simplified model. The results have shown that improving the valve geometry by rounding the sharp edges of the valve at the critical flow areas would led to an increase of the valve's outlet mass flow rate. At the same inlet pressure ranging from 2 to 12 bar, the mass flow rate increase was 79.1% at pressure 2 bar and 103.6% at 12 bar. This could increase the pneumatic actuator response and enhance the accuracy of the position control in pneumatic circuits that uses modified low-cost valves.

Keywords: Computational Fluid dynamics (CFD), Reynolds Averaged Navier Stocks (RANS), Control Valves, Internal Flow, Pneumatics

1. Introduction

Computational Fluid Dynamics (CFD) has been used by many researchers to investigate external and internal air flow (Draz et al., 2020; Mostafa et al., 2020). Pneumatic directional control valves have a complicated geometry to allow the control valve function without air leakage (Beater, 2007) They are mainly used to control the direction of air flow between different components of a pneumatic circuit. Their complicated internal geometry results in internal resistance that throttles the air flow, which is undesirable outcome. Directional control valves have flow characteristics that defines the relationship between spool displacement and mass flow rate under constant pressure conditions. The valve displacement is the distance travelled by the valve's spool or piston from its normal position. One of the most critical areas in any valve is the valve throat, which is the minimum area between the valve plug or spool and the valve seat or land through which air passes (Dempster et al., 2006; Dempster and Elmayyah, 2013; Elmayyah and Dempster, 2013). The air flow rate is dependent on this area and is directly proportional to it. However, if the flow is choked the mass flow rate will be independent on the downstream conditions. Therefore, for an ideal gas, its mass flow rate could be calculated from the upstream pressure and the throat area (Lenzing et al., 1998; Schmidt et al., 2009). Understanding the flow behaviour and predicting this area will allow opportunities to valve performance improvement according to the required application. Many researches were carried out to investigate the internal flow behaviour using CFD; Said et al. (2020) has investigated a butterfly valve to predict its flow coefficients. The CFD results showed good agreement with the experimental

Received 29 December 2022; accepted 30 March 2023.
Available online 12 June 2023
E-mail address: wael.elmayyah@mtc.edu.eg.

https://doi.org/10.58491/2735-4202.3042

results. On the other hand, Whitehead et al. (2007) studied a pneumatic control valve to enhance its performance by optimizing its geometry. The valve geometry was improved by rounding edges and removing dead spaces to increase the mass flow rate at same upstream pressure with a negligible force change on the spool. Davis and Stewart, 2002a, 2002b predicted a globe valve flow characteristics using a simplified two-dimensional axisymmetric CFD model and validated the computational model experimentally. It was shown that, the CFD model could accurately predict the characteristic of control valves such as the discharge coefficient along a wide range of the valve opening. Another study on globe valve was carried out by Cho et al. (2007). The pressure distribution on the valve moving element was predicted using CFD and by solving the force dynamic equation. The computational model results agreed with the experimental results, and they addressed the major area of force balance on the valve moving element. To pave the way for further development and engineering investigation, Qian et al. (2014) used CFD to investigate a pilot control globe valve. The authors investigated the pressure drop through the valve, the utilization of energy and its refection on the valve opening/closing and the spring selection. The recommended work conditions were proposed to exert optimum performance and a prolonged service life. In a similar study of optimization, CFD has been used to optimize a diffuser by Djebedjian et al. (Djebedjian, 2021). Yousry et al. (Youssry et al., 2016a, 2016b, 2020) implemented a pneumatic position control system using low-cost on-of pneumatic directional control valve. The pneumatic parameters such as the air mass flow rate, the upstream pressure and the pneumatic actuator back pressure have been used to regulate the actuator response to improve the system accuracy. Experimental study to validate a Simscape model has been carried out. The experimental measurements has been used in the Simscape model to calculate the mass flow rate and the transient time to control the position of a pneumatic actuator. All these studies and many others such as (Yang et al., 2011; Peng et al., 2012; Schmidt and Egan, 2009; Lisowski et al., 2013; Stewart et al., 2018) proved that using CFD to predict the flow parameters through pneumatic valves is effective and can give significant accuracy.

This allows researchers and designers to improve certain performance of valves according to the required function and the conditions of the operation. The above mentioned researches give good understanding of the internal incompressible flow. However, only few researches investigated pneumatic valves, in addition that these studies didn't give enough details on the effect of valve's geometry on pneumatic valve performance. Therefore, using the low-cost pneumatic valves in position control needs more detailed investigation. This leads to improve the valve design to match certain operation requirements.

In this study, CFD is used to investigate the flow through the valve. The pneumatic DCV has been investigated in (Youssry et al., 2016b, 2020) theoretically and experimentally by solving the dynamic equations using Simscape in Matlab (Azahar et al., 2020). However, solving the dynamic equations is limiting the understanding of the flow behaviour. Therefore many researches such as (Dempster et al., 2006; Elmayyah and Dempster, 2013; Schmidt et al., 2009; Said et al., 2020; Greshniakov et al., 2018; Qi et al., 2019) investigated flow through valves using CFD. These researches proved that the RANS equations with the k-ϵ turbulence model can effectively predict the supersonic compressible flow through valves. The CFD model has been validated by comparing the predicted CFD results with

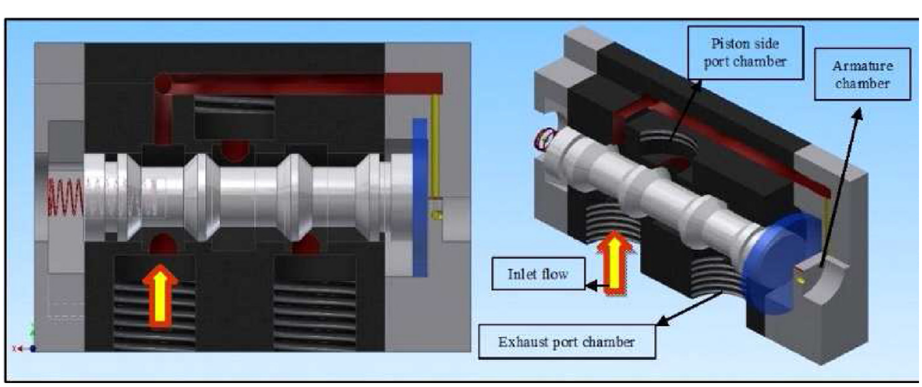

Fig. 1. Pneumatic valve construction.

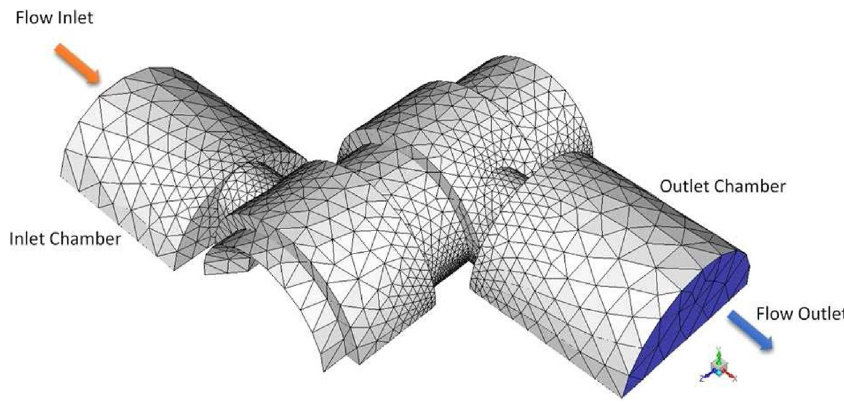

Fig. 2. The discretised computational domain of Model A.

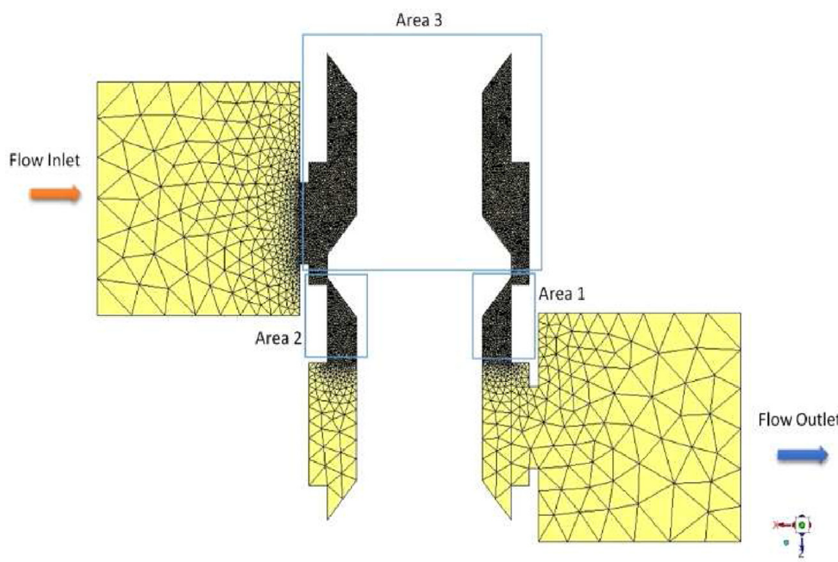

Fig. 3. The discretised computational domain of Model A at the symmetry plane.

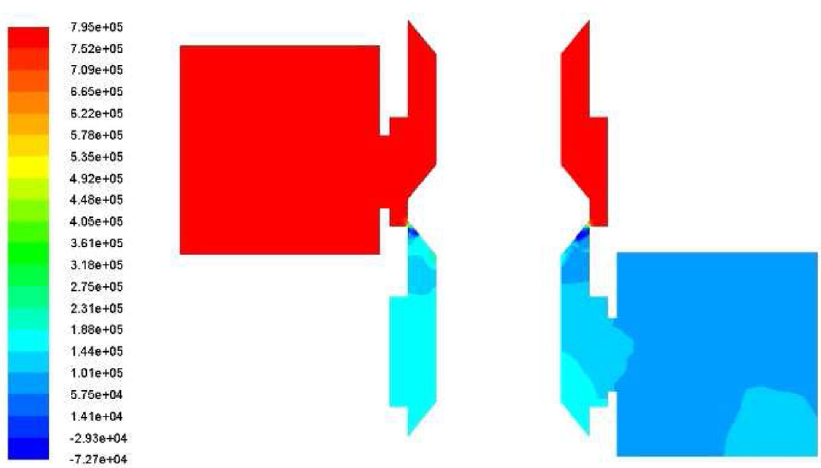

Fig. 4. The Static Pressure Contours at inlet Pressure of 8 barg.

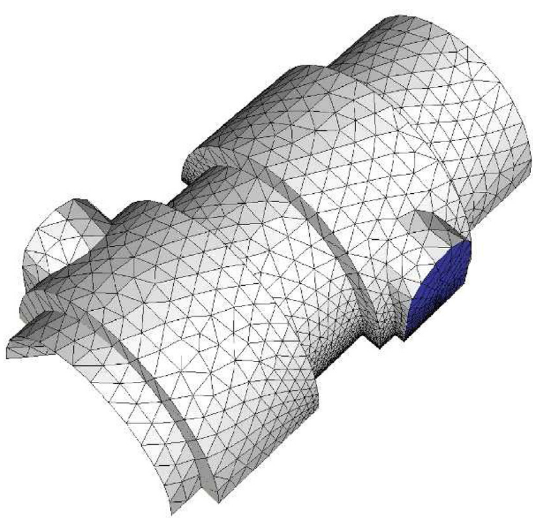

Fig. 5. The discretised computational domain of Model B.

experimental results carried out by Youssry et al., 2016a, 2016b in addition to an analytical simplified model. The critical flow area have been identified. Modifications to the valve's geometry at the critical flow areas has been carried out. This could lay the foundations for further optimization of valves design to meet the requirement of different applications at low-cost. The accuracy of the position control in systems that use low-cost on-off valves, could be increased by optimizing the valve design (Mathworks, 2007; Wallace et al., 2004; Dempster and Elmayyah, 2008).

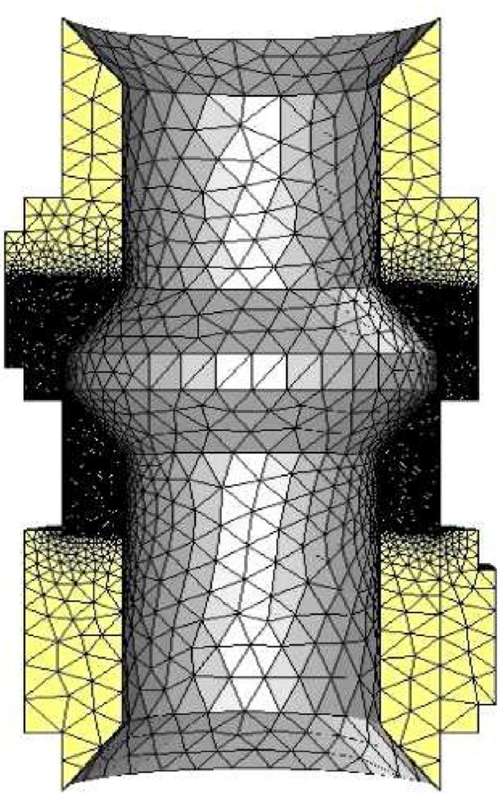

Fig. 6. The discretised computational domain of Model B at the symmetry plane.

1.1. Valve construction and operation

The pneumatic DCV used in this study is a normally closed (NC), three ports, and two position (3/2) solenoid operated DCV with internal pilot. A section in the 3D CAD model of the valve is shown in Fig. 1. The solenoid plunger allows the pilot signal stage; as it permits the compressed air to move the valve spool. The spool returned to its normal position by the act of a compression spring. The actuating solenoid is electrically operated by a supply voltage 24 dc Volt. The DCV is operated by 0.7 bar minimum to overcome the inertia of spool and spring force and up to 10 bar maximum pressure.

2. Methodolgy

2.1. Computational work

Air is modelled as ideal gas. RANS equations with the two-equation k-ϵ turbulence model has been used. The model has been generated in ANSYS19 and Fluent is used to solve the equations. To allow computational efficiencies a half 3D model has been developed to represent the valve geometry. The model is developed at the full opening of the spool when the flow is directed to the piston side port. The first used model 'Model A' is shown in Fig. 2, which holds all the geometry from the valve inlet to the valve outlet to the piston side port. 'Model A' has a computational mesh of 487,714 tetrahedral cells. High density mesh at the critical flow area between the valve body and the spool lands has been generated to capture the flow details. Fig. 3 shows Model A mesh with the high dense mesh on Areas 1,2 and Area 3; Where Area 1 is the smallest flow area between the spool and the valve body at the outlet side; Area 2 is the smallest flow area between the spool and the valve body at the inlet side; Area 3 is the flow area between the spool and the valve body upstream of Area 1 and Area 2.

Fig. 4 shows the static pressure distribution at inlet pressure of 8 barg. However, it could be noticed that the large inlet and outlet chambers have these two chambers and reduce the computational budget. For more simplification, only Area 1 and Area 2 have the dense mesh. The simplified computational Model 'Model B' has a computational mesh of a total of 252,055 tetrahedral cells, shown in Fig. 5, distributed

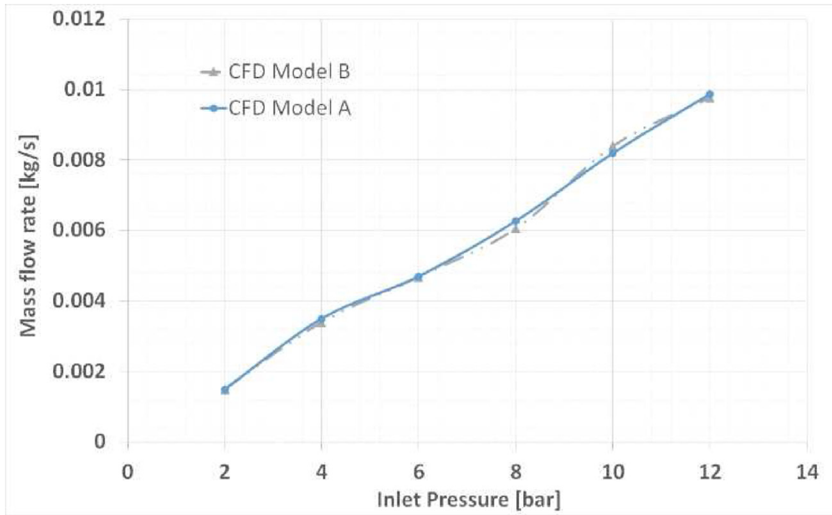

Fig. 7. Mass flow rate predicted by Model A and Model B.

giving a high density mesh at the critical flow area Area 1 and Area 2, shown in Fig. 6. A denser mesh of 354,781 tetrahedral cells has been used to examine the grid independence, with no significant improvement for the solution. The difference in air fow rate between inlet and outlet was 0.000001 kg/s so the cell number was kept about 252,055 in all cases. Fig. 7 shows the predicted mass flow rate by Model A and Model B at different inlet pressures. Therefore, Model B will be used in the rest of the study.

2.1.1. Boundary conditions and solution

The proper setting of the boundary conditions is an essential step to obtain accurate CFD results. The boundary conditions are applied at the valve entrance, valve outlet and valve walls. Valve walls were defined as stationary walls. At the inlet boundary, the stagnation pressure, static pressure and stagnation temperature are applied; in addition an initial value for the turbulence intensity and the hydraulic diameter are introduced. Hence, an initial air mass flow rate is determined at the inlet area then is recalculated from the downstream conditions at the choking plane. At the outlet boundary conditions the static pressure and the stagnation temperature are applied. However, the flow calculation 115 is independent of the outlet boundary condition if the flow is choked. The discertization scheme used for the continuity, momentum, energy, turbulent kinetic and turbulent dissipation energy equations was

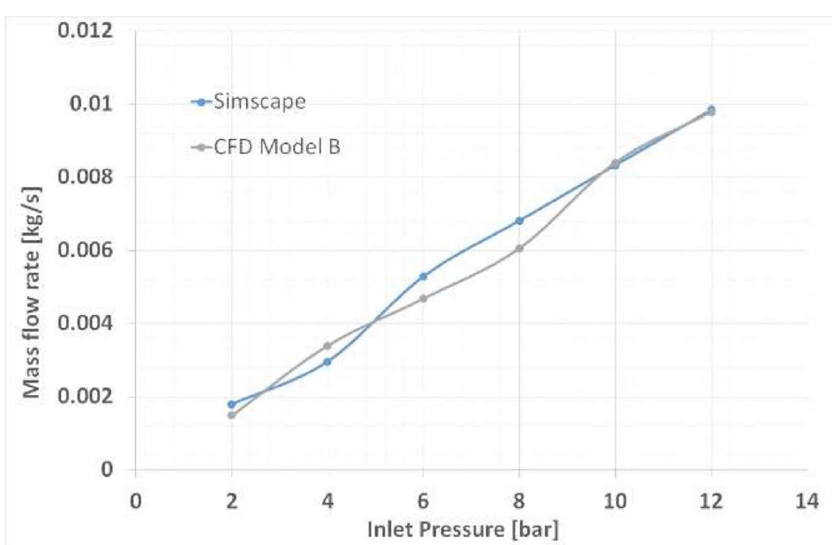

Fig. 8. The predicted and measured Air flow rate.

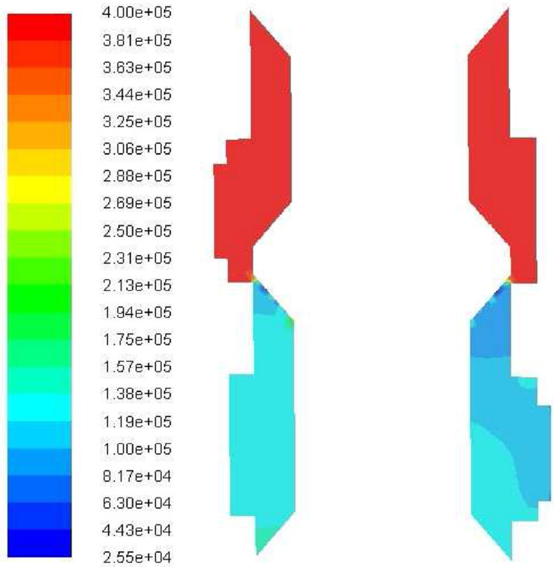

Fig. 9. Static Pressure Contours on the Symmetry Plan at inlet pressure 4 barg.

second order upwind for the convection terms and second order central difference for the diffusion terms. The convergence criterion was based on the residual values of the calculated variables, i.e., mass, velocity components, energy, turbulent kinetic energy and turbulent dissipation energy. The threshold values were absolute with magnitudes of $1 \times 10-3$ for all variables, except for the energy equation where it was $1 \times 10-6$. The pressures range used was 2–12 barg.

2.2. Analytical simplified model

One of the traditional ways of calculating the mass flow rate through orifices or valve's orifices is to use the simplified models. Equation (1) shows a simplified equations to describe the air flow through the valve orifice. Air is considered as an ideal gas passes through a sharp edge orifice with no heat losses. Youssry et al. (2016a) uses Simscap in Matlab to solve this model and predict the mass flow rate of the valve. However, it is very challenging to predict the real 3D annuls flow area (A) and the proper discharge coefficient (Cd.A) due to the complexity of the flow parameters and the uncertainty of the critical flow area in particular in the case of choked flow. The (Cd.A) value has been estimated from the experimental work and used in the Simscape model. Therefore, CFD can provide much more detailed information on the flow behaviour through the orifice and predict much more accurate mass flow rate than the simplified model without the help of the expensive and exhausting experimental work. To validate the CFD model, a comparison between the predicted mass flow rate by the CFD Model B and the results of the experimental results provided by (Youssry et al., 2016b) and the results of the analytical simplified model of equation (1) is shown in Fig. 8. The comparison shows good agreement which gives more confidence in the CFD model. The difference of the predicted mass flow rate is ranging from 0.7% to 7.5%.

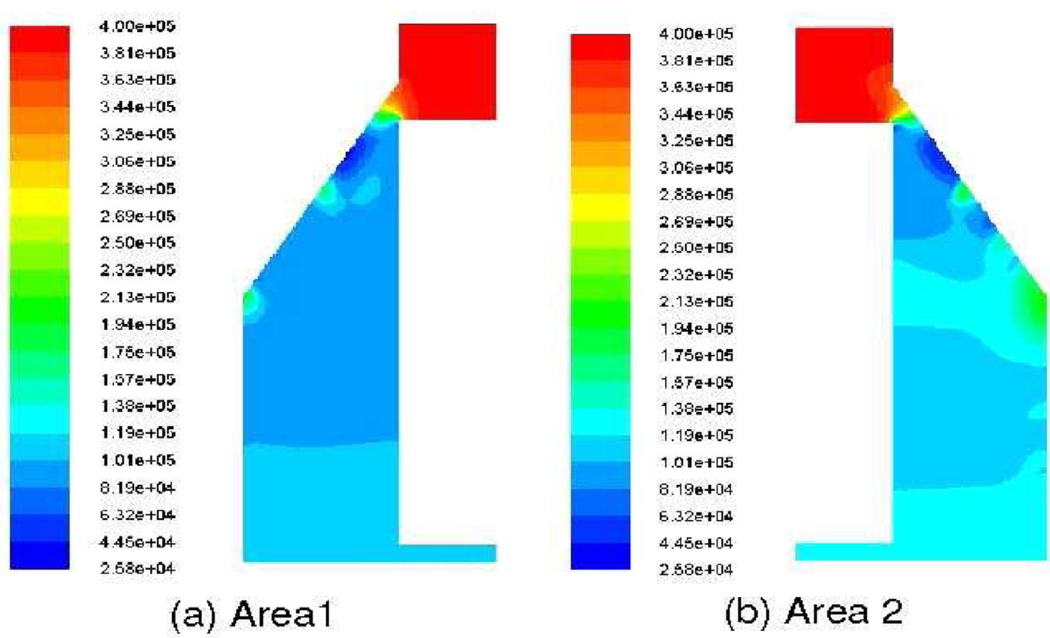

Fig. 10. Static Pressure Contours on the Symmetry Plan at inlet pressure 4 bar at Area 1 and Area 2.

$$\dot{m}=\begin{cases} A.C_d.P_1\sqrt{\dfrac{2k}{RT(k-1)}\left(\left(\dfrac{P_2}{P_1}\right)^{\frac{2}{k}}-\left(\dfrac{P_2}{P_1}\right)^{\frac{k+1}{k}}\right)}\ \text{Subsonic Flow}\left(\dfrac{\mathrm{P}_2}{\mathrm{P}_1}\right)>0.528 \\ A.C_d.P_1\left(\dfrac{2}{k+1}\right)^{\frac{1}{k-1}}\left(\sqrt{\dfrac{2k}{RT(k+1)}}\right)\ \text{choked flow}\left(\dfrac{\mathrm{P}_2}{\mathrm{P}_1}\right)\le 0.528 \end{cases} \tag{1}$$

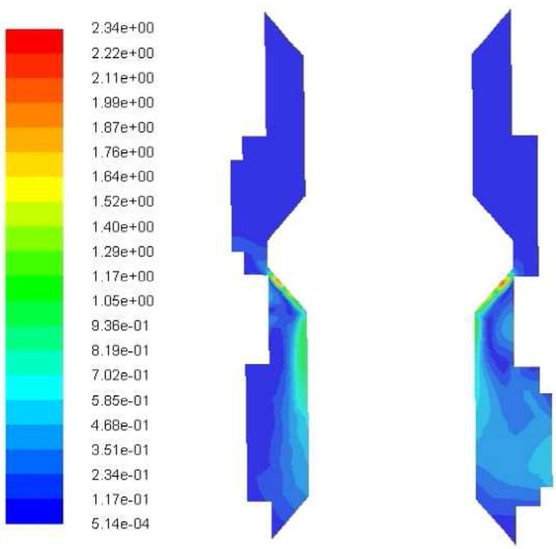

Fig. 11. Mach Number Contours on the Symmetry Plan at inlet pressure 4 bar at Area 1 and Area 2, Model B.

3. Results and discussion

CFD gives a good opportunity to investigate the fow behaviour and the fow parameters distribution. These investigations could give deeper understanding of the critical parameters that could control the valve for better performance. From the static pressure and Mach number distribution through the valve, the critical fow area can be predicted. The valve outlet mass fow rate directly afects the actuator speed and response. In this study, all results are predicted at the full opening position as this valve is only on-of valve. Fig. 9 presents the static pressure distribution on the valve symmetry plane at inlet pressure of 4 bar. The figure shows the static pressure distribution from the inlet at 4 barg to the outlet at 0 bar, while Fig. 10 shows the static pressure contours on Area 1 and Area 2. The static pressure value in the inlet chamber is nearly constant and equals to the valves' pressure inlet value. A pressure build up takes place on the spool walls at Area 1. This

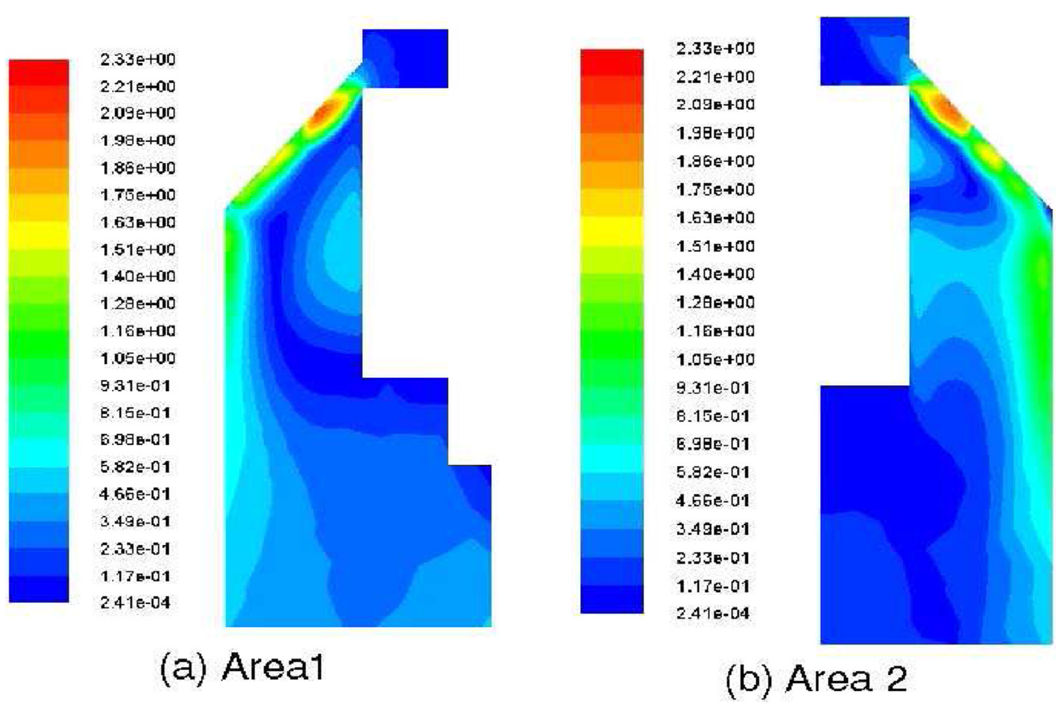

Fig. 12. Mach Number Contours on the Symmetry Plan at inlet pressure 4 bar at Area 1 and Area 2, Model B.

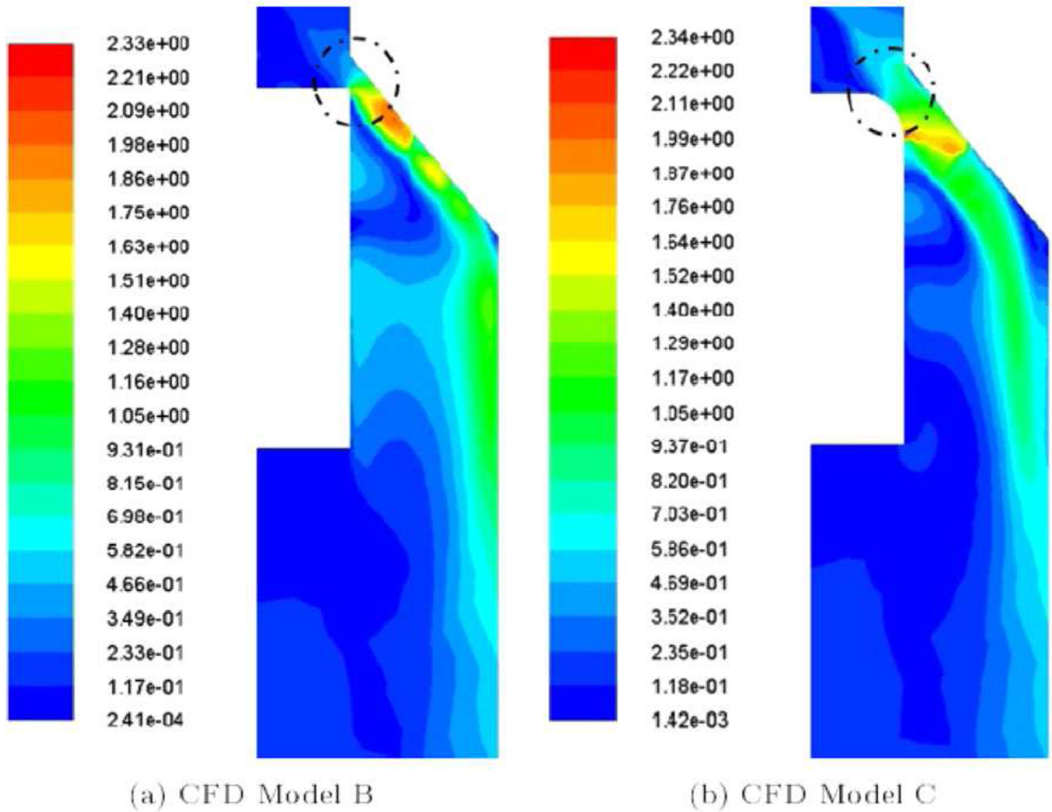

Fig. 13. Mach Number Contours on the Symmetry Plan at inlet pressure 4 bar at Area 1, Model B and Model C.

pressure increase is high risk of forming a higher intense shockwave. The risk is higher in case of higher inlet pressure and large actuator. The formation of shockwaves in safety relief valves has been investigated by (Elmayyah, 2013) and further investigations of the shockwave in this pneumatic DCV will take place in a future study. Fig. 11 illustrate the Mach number contours at inlet pressure of 4 bar. It

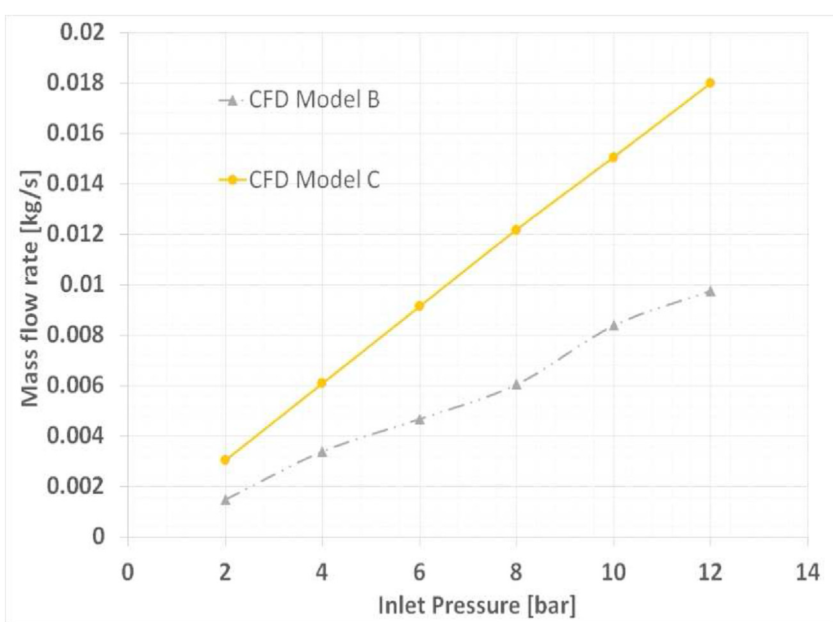

Fig. 14. Air Mass Flow at different inlet pressures.

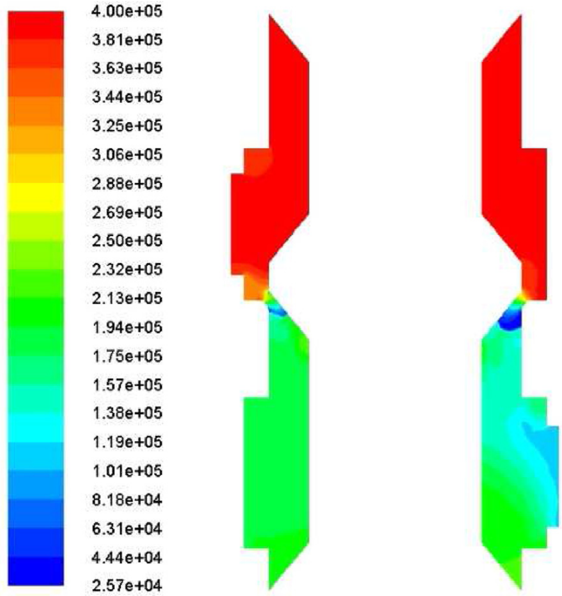

Fig. 15. Static Pressure Contours on the Symmetry Plan at inlet pressure 4 bar, Model C.

could be noticed that the flow is nearly static in the inlet chamber and starts to accelerate at Area 1 and Area 2. Fig. 12 shows the Mach number contours at Area 1 and Area 2. If the flow is subsonic, the mass flow rate depends on the flow area and the pressure difference of the pressure upstream and downstream the valve. If the flow is sonic or supersonic the mass flow depends on the critical area and the upstream pressure value. The critical area is characterised by chocked flow i.e. $M = 1$. Therefore, the critical area in Fig. 11 that could be recognised by the Mach number value of 1. It is easily observed that the critical flow area here at $M = 1$ is not a regular 3D shape which makes it harder to be calculated for one-dimensional models or the simplified models without the aid of the experimental setup. The valve under study here is a low-cost on-off valve which has been used to control the position of a pneumatic cylinder. To improve a low-cost pneumatic circuit response, its needed to increase the mass flow rate delivered to the actuator through the pneumatic DCV. Therefore, increasing the valve outlet mass flow rate at the same pressure inlet will assist the circuit response and control. A slight modification in the critical flow area of the valve will significantly change the flow parameters. A modified model of the valve has been developed denoted by Model C; in which the sharp edge of the internal land area of the valve at Area 1 and Area 2 has been rounded to be of a radius of 0.5 mm. Fig. 13 shows the Mach number distribution at Area 1 predicted by Model B and Model C; the

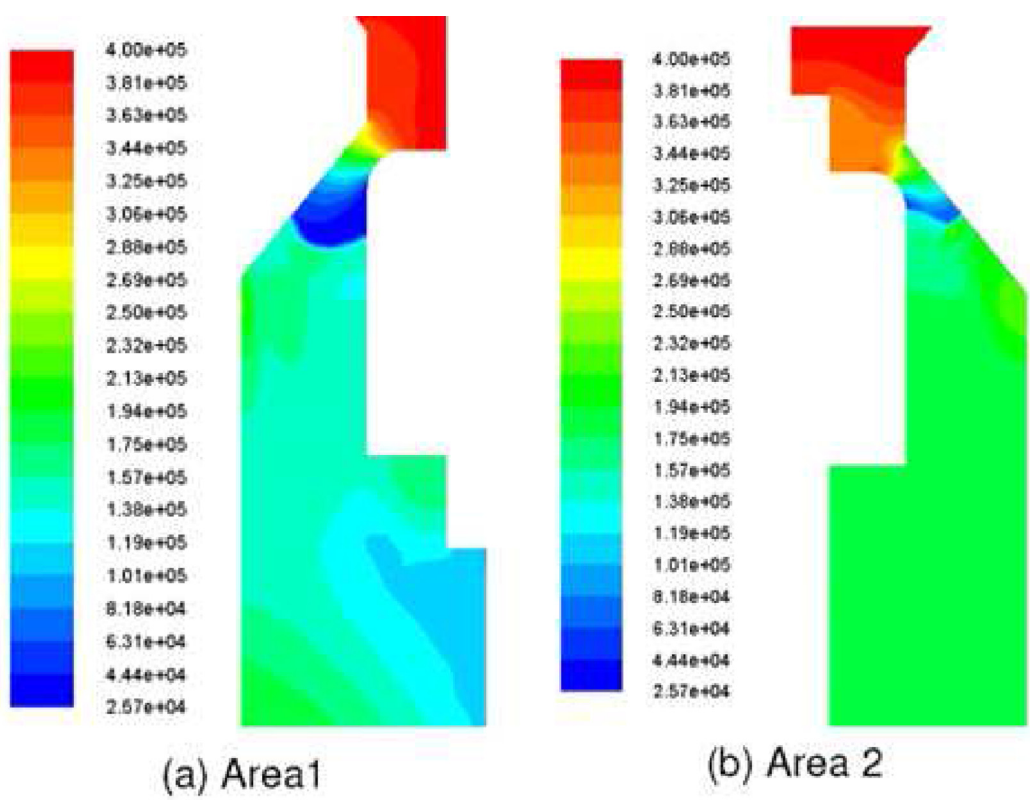

Fig. 16. Static Pressure Contours on the Symmetry Plan at inlet pressure 4 bar at Area 1 and Area 2, Model C.

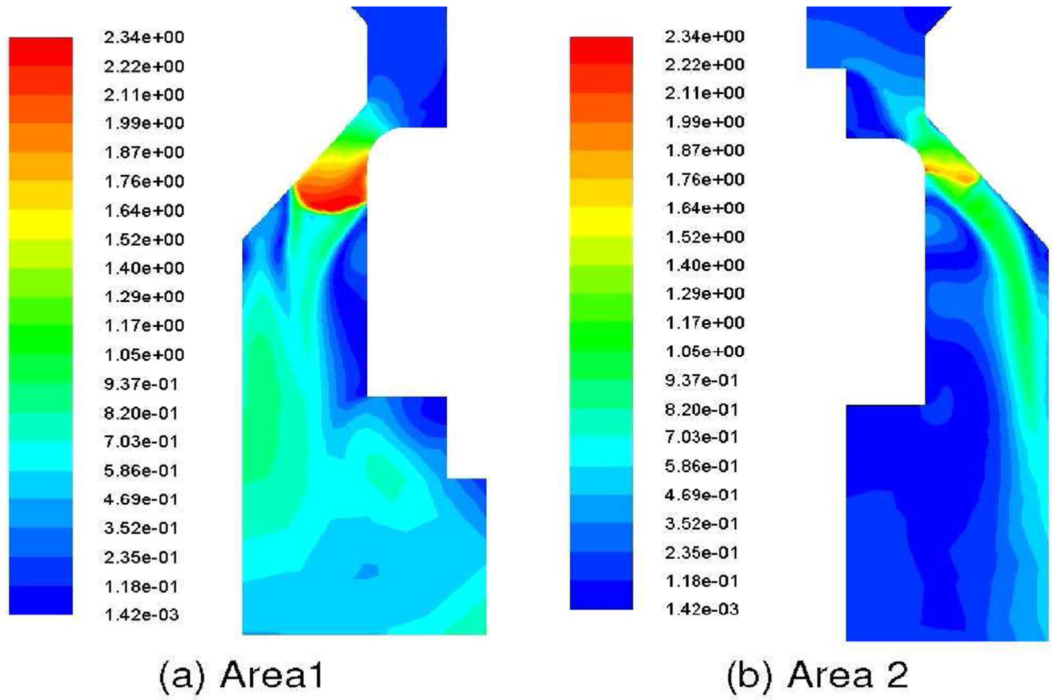

Fig. 17. Mach Number Contours on the Symmetry Plan at inlet pressure 4 bar, Model C.

sharp edge that has been modified is denoted by the dotted circle.

Fig. 14 shows the mass flow rate predicted by the CFD Model B (original valve design) and Model C (the modified rounded sharp edges). The figure shows a significant increase of the mass flow rate at the same pressure inlet value ranging from 79.1% to 103.62%. Where the mass flow rate increase was

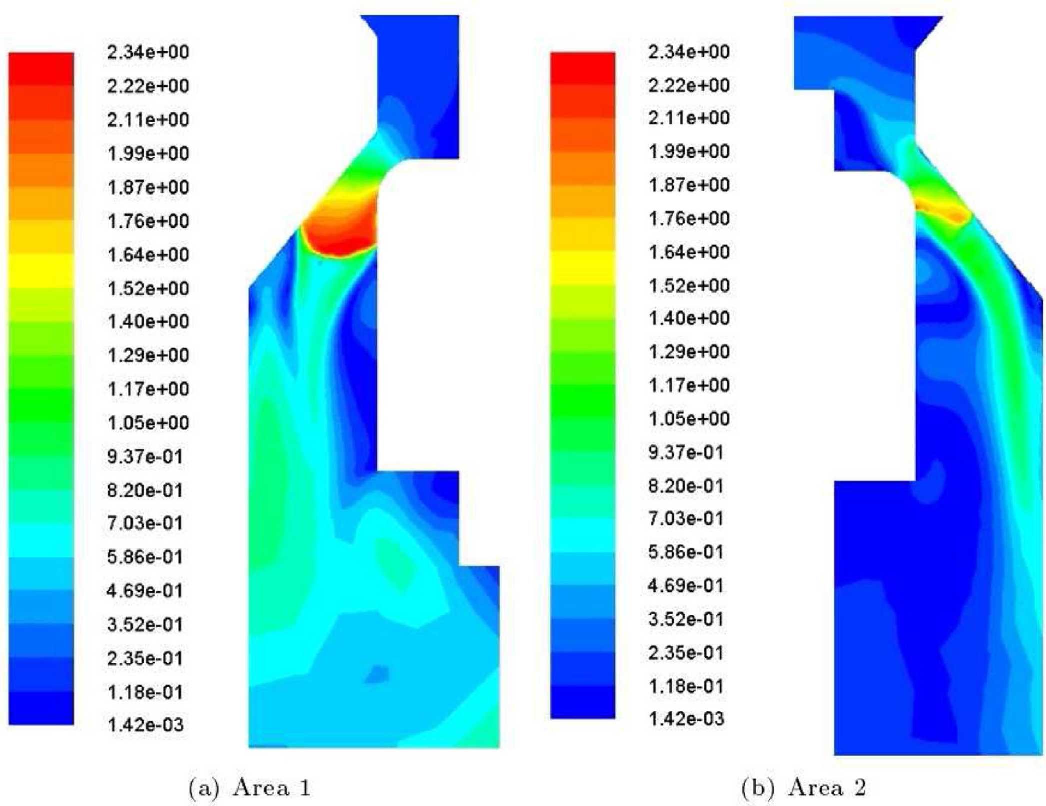

Fig. 18. Mach Number Contours on the Symmetry Plan at inlet pressure 4 bar at Area 1 and Area 2, Model C.

79.1% at inlet pressure of 2 bar, while it is 103.62% at 12 bar. Fig. 15 shows the static pressure contours of Model C at inlet pressure 4 bar. While Fig. 16 shows the static pressure contours on Area 1 and Area 2. A pressure build up takes place on the spool walls at Area 1 and Area 2. This pressure increase is high risk of forming a higher intense shockwave than Model B. The Mach number contours on the valve and Area 1 and Area 2 shown in Figs. 17 and 18 can confirm the formation of Shockwave at the Mach number sharp decrease line. This critical flow area can dramatically control the valve performance and it needs further investigation for a proper optimization according to the required application.

4. Conclusions

Computational Fluid Dynamics can be effectively used in understanding flow parameters through pneumatic valves. This allows further improvements in valve performance for different applications. A pneumatic (3/2) solenoid operated DCV with internal pilot has been investigated using CFD to study the internal flow. Three different computational half 3D models have been developed to optimise the computational effort and get the highest accuracy. The critical flow area between the spool and the valve body has been identified. This irregular shape flow area dramatically controls the valve mass flow rate and other flow parameters such as the pressure distribution and the formation of the shockwave. This irregular flow area cannot be easily predicted without the aid of Computational Fluid Dynamics. The model can predict the mass flow rate with acceptable accuracy and agreement with experimental results and an analytically solved simplified model. By rounding the valve body sharp edges at the critical flow area, the mass flow rate has been increased by 79.1%–103.6% at same valve inlet pressure. Where the mass flow rate increase was 79.1% at inlet pressure of 2 bar, while it is 103.62% at 12 bar. This increase in mass flow rate can significantly improve the circuit response and control. The risk of shockwave formation has been noticed and it needs further investigation. These results and deeper understanding of the flow through the valve will pave the way for further investigations to optimise the valve design.

Conflicts of interest

There is no potential conflicts of interest with respect to the research, authorship or publication of this article.

References

Azahar, M.I.P., Irawan, A., Tau_ka, R.M., Suid, M.H., 2020. Position control of pneumatic actuator using cascade fuzzy self-adaptive PID. In: Lecture Notes in Electrical Engineering. Springer, Singapore, pp. 3–14. https://doi.org/10.1007/978-981-15-2317-5_1.

Beater, P., 2007. Pneumatic Drives. Springer Berlin Heidelberg. https://doi.org/10.1007/978-3-540-69471-7.

Cho, T.-D., Yang, S.-M., Lee, H.-Y., Ko, S.-H., 2007. A study on the force balance of an unbalanced globe valve. J. Mech. Sci. Technol. 21, 814–820. https://doi.org/10.1007/bf02916360.

Davis, J.A., Stewart, M., 2002a. Predicting globe control valve performance_part i: CFD modeling. J. Fluid Eng. 124, 772–777. https://doi.org/10.1115/1.1490108.

Davis, J.A., Stewart, M., 2002b. Predicting globe control valve performance_part II: experimental verification. J. Fluid Eng. 124, 778–783. https://doi.org/10.1115/1.1490126.

Dempster, W., Elmayyah, W., 2008. A computational Fluid dynamics evaluation of a pneumatic safety relief valve. In: Proceeding of the 13th International Conference on Applied Mechanics and Mechanical Engineering (AMME). Military Technical College. https://doi.org/10.21608/amme.2008.39005.

Dempster, W., Elmayyah, W., 2013. Two phase discharge flow prediction in safety valves. Int J Press Vessels aPip 110, 61–65. https://doi.org/10.1016/j.ijpvp.2013.04.023.

Dempster, W., Lee, C.K., Deans, J., Jul. 2006. Prediction of the Flow and Force Characteristics of Safety Relief Valves. Proceedings of PVP2006-ICPVT-11 2006 ASME Pressure Vessels and Piping Division Conference. https://doi.org/10.1115/pvp2006-icpvt-11-93142.

Djebedjian, B., 2021. Difuser optimization using computational fuid dynamics and micro-genetic algorithms.(dept m). MEJ Mansoura Eng J 28, 15–34. https://doi.org/10.21608/bfemu.2021.142398. URL.

Draz, A., Saadany, H.E., Awad, M., Awady, W.E., 2020. Investigation of air fow over delta and cranked arrow delta wings. (dept. m.). MEJ Mansoura Eng J 45, 1–9. https://doi.org/10.21608/bfemu.2020.112323. URL.

Elmayyah, W., 2013. Computational fluid dynamics investigations of shock waves in safety relief valves. International Conference on Aerospace Sciences and 300 Aviation Technology 15, 1–11. https://doi.org/10.21608/asat.2013.22200 (AEROSPACE SCIENCES).

Elmayyah, W., Dempster, W., 2013. Prediction of two-phase flow through a safety relief valve, Proceedings of the Institution of Mechanical Engineers. Part E. J Process Mech Eng 227, 42–55. https://doi.org/10.1177/0954408912453407.

Greshniakov, P., Handroos, H., Sverbilov, V., 2018. Position control of pneumatic cylinder actuated by low-cost on/o_ valves pulse number modulation. In: 2018 Global Fluid Power Society PhD Symposium (GFPS), IEEE. https://doi.org/10.1109/gfps.2018.8472404.

Lenzing, T., Friedel, L., Alhusein, M., 1998. Critical mass flow rate in accordance with the omega-method of DIERS and the Homogeneous Equilibrium Model. J. Loss Prev. Process. Ind. 11, 391–395. https://doi.org/10.1016/s0950-4230(98)00022-9.

Lisowski, E., Czyycki, W., Rajda, J., 2013. Three dimensional CFD analysis and experimental test of flow force acting on the spool of solenoid operated directional control valve. Energy Convers. Manag. 70, 220–229. https://doi.org/10.1016/j.enconman.2013.02.016.

Mathworks, Simulink, 2007. Simulink. Oldenbourg Wissenschaftsverlag, pp. 1–48. https://doi.org/10.1524/9783486841244.1.

Mostafa, H., Awad, M.M., Sultan, G., El-ghonemy, A., Elbooz, A., 2020. Enhancing air side heat transfer coefcient of fat tube air cooled condensers.(dept m). MEJ Mansoura Eng J 31, 15–24. https://doi.org/10.21608/bfemu.2020.129417. URL.

Peng, Z.-f., Sun, C.-g., Yuan, R.-B., Zhang, P., 2012. The CFD analysis of main valve flow field and structural optimization for double-nozzle flapper servo valve. Process Eng. 31, 115–121. https://doi.org/10.1016/j.proeng.2012.01.1000.

Qi, H., Bone, G.M., Zhang, Y., 2019. Position control of pneumatic actuators using three-mode discrete-valued model predictive control. Actuators 8, 56. https://doi.org/10.3390/act8030056.

Qian, J.-Y., Wei, L., Jin, Z.-J., Wang, J.-K., Zhang, H., Lu, A.-L., 2014. CFD analysis on the dynamic flow characteristics of the pilot-control globe valve. Energy Convers. Manag. 87, 220–226. https://doi.org/10.1016/j.enconman.2014.07.018.

Said, M., AbdelMeguid, H., Rabie, L., 2020. A comparison study between 3-d CFD and experimental data of butterfly valve coefficients. (dept. m. (power)). Bull Faculty Eng Mansoura Univ 39, 10–22. https://doi.org/10.21608/bfemu.2020.102725. URL.

Schmidt, J., Egan, S., 2009. Case studies of sizing pressure relief valves for two-phase flow. Chem. Eng. Technol. 32, 263–272. https://doi.org/10.1002/ceat.200800572.

Schmidt, J., Peschel, W., Beune, A., 2009. Experimental and theoretical studies on high pressure safety valves: sizing and design supported by numerical calculations (CFD). Chem. Eng. Technol. 32, 252–262. https://doi.org/10.1002/ceat.200800632.

Stewart, K., Rob, I., Faik, H., 2018. CFD analysis of c-d nozzle compared with theoretical and experimental data. INCAS Bull 10, 53–64. https://doi.org/10.13111/2066-8201.2018.10.2.6.

Wallace, M., Dempster, W.M., Scanlon, T., Peters, J., Mcculloch, S., 2004. Prediction of impact erosion in valve geometries. Wear 256, 927–936. https://doi.org/10.1016/j.wear.2003.06.004.

Whitehead, N.P., Slaouti, A., Taylor, H., 2007. Optimisation of flow through a pneumatic control valve using CFD analysis and experimental validation. Int. J. Fluid Power 8, 31–41. https://doi.org/10.1080/14399776.2007.10781284.

Yang, Q., Zhang, Z., Liu, M., Hu, J., 2011. Numerical simulation of fluid flow in side the valve. Process Eng. 23, 543–550. https://doi.org/10.1016/j.proeng.2011.11.2545.

Youssry, M., Elmayyah, W., Mabrouk, M., Mahgoub, H., 2016a. Parametric study of a low cost pneumatic systemcontrolled by on/o_ solenoid valves. Int J Res Eng Technol. https://doi.org/10.15623%2Fijret.2016.0503038.

Youssry, M.A., Mahgoub, H.M., Mabrouk, M.H., Elmayyah, W.M., 2016b. Design and Simulation of Mechatronic Systems, Pneumatic Positioning Application, Master's Thesis. In: The Military Technical College (ed.), Mechanical Engineeiring.

Youssry, M., Elmayyah, W., Mabrouk, M., 2020. Position control of a pneumatic cylinder actuator using modi_ed PWM algorithm. J Eng Sci Mil Technol 4, 121–126. https://doi.org/10.21608/ejmtc.2020.31861.1145.

المخلص

تستخدم صمامات التحكم في الاتجاه الهوائية منخفضة التكلفة بتوسع مع دوائر التحكم الكهربية الرقمية للتحكم في سرعة أو وضع الأسطوانات الهوائية. تتميز هذه الصمامات بمعدلات تدفق محدودة مما يؤثر علي سرعة استجابة الأسطوانات. ولكي نستطيع أن نحسن من أدائها فلابد من فهم أعمق لتدفق الهواء داخل الصمام وتأثير الشكل الهندسي عليه.

في هذه المقالة تم عمل نموذج حسابي باستخدام ديناميكا الموائع الحسابية لصمام تحكم في الاتجاه 2/3 منخفض التكلفة يعمل بإشارة داخلية ويتحكم به عن طريق فتح وغلق تيار الكهرباء وذلك لدراسة سلوك تدفق الهواء للتمهيد لتحسين أداء الصمام. تم دراسة سلوك الهواء المضغوط كما تم مناقشة مساحات التدفق الهندسية الحرجة للصمام. تم التحقق من فعالية النموذج الحسابي عن طريق مقارنة نتائجه بنتائج عملية منشورة مسبقا.

أوضحت النتائج أن تحسين الشكل الهندسي الداخلي للصمام عن طريق التخلص من الزوايا الحادة عند المساحات الحرجة أدي إلي تحسين أداء الصمام عن طريق زيادة معدل تدفق الهواء وبالتالي تم تحسين زمن رد فعل الصمام. فعند نفس ضغط دخول الهواء للصمام من 2-12 بار زاد معدل تدفق كتلة الهواء بنسبة 79.1 % عند 2 بار وبنسبة 103.6 % عند 12 بار. ويمكن أن تعمل هذه الطريقة علي تحسين دقة التحكم في وضع الأسطوانات الهوائية ودوائر الهوائية بصفة عامة باستخدام صمامات تحكم في الاتجاه منخفضة التكلفة المعدلة لهذا الغرض.