International journal of COMADEM
Vol # No # (January 2022) p.##-##

COMADEM

International Journal of Condition Monitoring and Diagnostic Engineering Management

A new one-dimensional control valve model for the local flow coefficient based on local analysis of CFD data

Qingyun Bu, Aliyu M. Aliyu, Rakesh Mishra*

[a] *School of Computing & Engineering, University of Huddersfield, Queensgate, Huddersfield HD1 3DH, United Kingdom*

* *Corresponding author. Tel.: +44 1484 473263; email: r.mishra@hud.ac.uk*

ABSTRACT

Control valves are essential components of the energy, nuclear, process, and the oil and gas industries. They are needed for the accurate control of flow rate, pressure, and temperature crucial to the success of reactions in chemical processes and delivering the right amount of fluid to downstream facilities for storage, transport, or sale. As a result, the optimal design of control valves for being fit for purpose and for energy efficiency is of crucial industrial and commercial importance. In order to achieve optimal design, computational fluid dynamics is used as a powerful tool for the analysis and design of many process units including control valves. The full three-dimensional CFD simulations are useful but can be computationally expensive and time consuming if a large number of designed need testing. Furthermore, rapid solutions may be needed for testing many initial design configurations or for digital twin purposes where real time or near real time solutions are needed. One way of achieving quick solutions is the use of one-dimensional approximations of the high-fidelity simulations 3D flow within the valve. In this study, CFD simulations were carried out and used to establish a 1-D numerical model to predict the flow within a control valve having a trim with a fairly complex geometry. Five valve opening positions (20, 40, 60, 80, and 100%) and two inlet flow velocities of 1 and 2 m/s were simulated. The valve flow coefficient Cv for each of the 10 cases was collected along the valve–pipe system on fifteen local planes and used to build the 1-D model in which the Cv is correlated with the VOP, Reynolds number, and length-diameter ratio. It was shown that this data driven model resulted in a coefficient of determination R^2 value of 0.82 when compared with the underlying data and can be used with confidence for initial design and digital twin applications.

Keywords: Control valves, flow coefficient, numerical simulation, valve design, pressure distribution, computational fluid dynamics, local flow analysis
Article history: *Received ; Published .*

1. Introduction

Being one of the most common and essential components of a piping system, valves are often used to adjust flow conditions, such as flow rate, temperature, pressure, and velocity. From domestic and residential piping systems to industrial water and energy systems, valves interact with other pipeline components to help achieve the desired fluid flow conditions. According to usage, valves may be divided into various types. These include the valves for starting or stopping the flow of a fluid or mixture (known as quick closing valves) and for regulating the flow rate or pressure. Control valves can also be used to direct or redirect flow. Depending on the number of its outlet ports, the valve can channel the direction of the flow to different downstream pipes. Control valves can also be used to improve the safety of a process by relieving pressure. In order to achieve any of these purposes, control valves work by changing the valve open position (VOP). The VOP is expressed in percentage terms, and it describes the distance between the valve's piston and the valve seat. The valve can be treated as fully opened when its VOP is 100% and fully closed when it is 0%. Flow regulation is achieved by varying the VOP between these two extremes. Some of the details of a typical valve are shown in figure 1.

In order to achieve optimal design of control valves and other process units, computational fluid dynamics is a powerful tool used for first modelling the flow within the system and studying the characteristics of various configurations prior to actually building the valve.

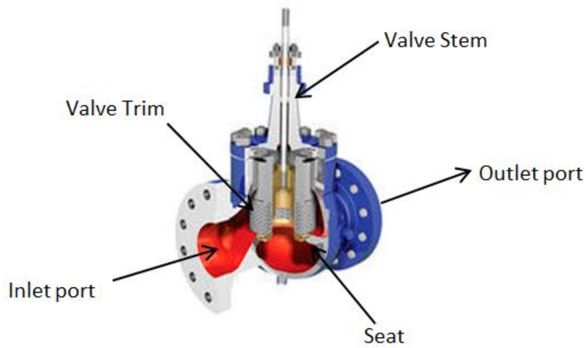


Figure 1: Components of a control valve with trim [1]

For achieving this, full three-dimensional CFD simulations are useful, but these can be computationally expensive and time consuming if many different conditions are to be tested. Furthermore, rapid fluid flow solutions are needed for testing many initial design configurations and for digital twin purposes where real time or near real time solutions are needed. One method of achieving timely solutions is the use of high-fidelity one-dimensional approximations of the 3D flow within the valve. Given that the valve especially its trim will have a complex geometry, one dimensional approximations of the valve (which could be pressure, velocity, or Cv based) present useful approximations for use as virtual twins of the real valve to estimate its flow behaviour in real time.

Numerous investigations have been carried out to understand local flow conditions within control valves. An [2] analysed a 3D incompressible cavitation flow within a globe valve system with a trim, and a CFD simulation has been accomplished to assist in understanding the flow phenomena at inlet and outlet pipe, valve body and trim domain. Analysis was carried out using the pressure contours and the global valve flow coefficient variation. Green [3] carried out CFD studies on a control valve with a continuous resistance (otherwise known as the X-stream) trim. Experiments were carried out to globally validate the valve, but local flow analysis was not carried out. The results indicated a good match between experimental and simulation results.

Kwon [4] described the characteristics of a turbine bypass valve trim. Finite element analysis (FEM) was used to analyse the structural stability of the trim, the total pressure, stroke percentage and other performance parameters were compared with referred standards. Asim et al [5] developed a numerical method to predict the local flow condition within the valve trim and with the simulations globally validated. Local flow analysis was carried out based on different rows and disks of the trim. They built a correlation which only predicted flow parameters within the valve trim and presents a quick method to calculate the local flow coefficient within the valve trim. Asim et al. [6] used a dimensionless parameter to quantify the local pressure losses within the trim. They found that the continuous resistance trim showed maximum pressure loss at the middle cylinder radially towards the trim centre. Singh et al [7] reported a CFD study on multiphase gas/liquid through a valve using the same trim as Asim et al [6]. Singh et al. separated the valve trim into 4 quadrants and variables were monitored at the top, middle and bottom discs of the trim. Five independent variables were represented in the developed Cv equation namely, axial location of disk of trim, VOP, air volume fraction, row, and quarter of each disk. The local flow coefficient correlation was shown to very well match the CFD data.

All the above-mentioned studies have studied control valves but do not derive an equivalent 1-D model of it which can be used for digital twin applications as well as optimisation [8]. In fact, while digital twin studies of various process units have been carried out, for control valves it is scarce, especially from the standpoint of using CFD-assisted local flow analysis as the valve's virtual twin. based on these shortcomings, the current study determines the local flow condition in a globe control valve by using CFD simulation under varying VOP and inlet flow conditions from which, a Cv-based 1-D model showing the effect of VOP, Reynolds number and length-diameter ratio were incorporated. Two single-phase water inlet flow boundary conditions (velocities of 1 m/s and 2 m/s), five VOPs (20, 40, 60, 80, and 100%) and fifteen L/D locations. A nonlinear least-square regression method was then applied to correlate these with the Cv.

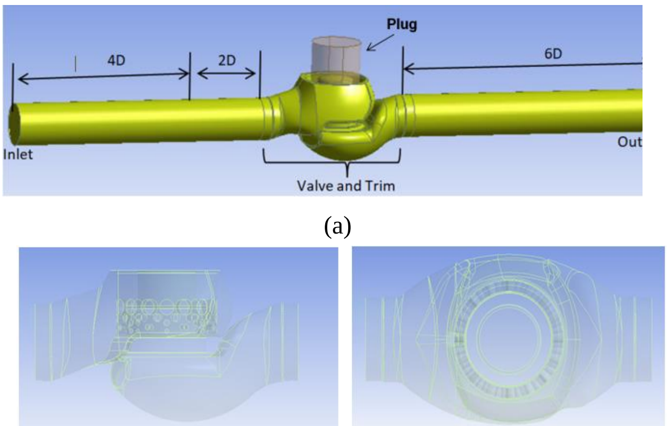
2. CFD modelling of the control valve

2.1. CFD modelling

CFD modelling of a control valve was carried out using the ANSYS Fluent 20.2 software. The control valve's local flow characteristic and global performance were calculated and analysed. The workflow followed the traditional CFD-based numerical investigation which involve pre-processing (geometry creation and meshing), solver setup, running the simulation, and post-processing as used by many previous similar works [9], [10]. In the pre-processing stage, the exact geometry of the laboratory valve is defined, and a suitable mesh size is required as it gives an optimum simulation quality and accuracy. Usually, small mesh element size gives better results. However, small mesh sizes can cause longer process time, and the simulation results may not change when the grid size reaches a certain level. Thus, ensuring the balance between the accuracy of results and the processing time important. For this reason, a mesh independence study was caried out to define an optimal mesh size. After that, the next step was to specify the boundary conditions, turbulence model, material settings and solution method.

2.2. Control valve and trim geometry

The geometry of the control valve with trim inside the valve body is shown in Fig. 2. Fig 2 (a) shows a 4-inch (100 mm internal diameter) globe valve with an inlet and outlet pipe connected to it. In accordance with the BS EN 60534-2-1 standard for control valve flow capacity and sizing, the length of inlet and outlet pipes should be set at as at least 2D and 6D respectively [11]. Figure 2 (a) shows the valve and pipe geometry with an additional 4D pipe section added to the inlet pipe for flow development. Figs. 2 (b) and (c) show the geometry of the side and top views of the valve trim. The trim has three rows of holes in a staggered formation (Fig. 2 b), with the flow moving inwards from outwards of the trim. The presence of the trim gives rise to extra pressure drop than without a trim, and this is useful for flow control with the plug moving upwards or downwards above the valve seat.



(a)

(b) (c)

Figure 2: (a) Valve and pipeline flow domain geometry showing the pipe lengths according to the relevant BS standard (b) side view of the valve section (c) top view of the valve section

2.3. Meshing and mesh independence test

As the geometries of the pipe domains are simple, so the hexahedral mesh method is used on the inlet and outlet pipe sections while tetrahedral mesh used for the valve part (shown in Fig. 3). In order to obtain reliable simulation results, an appropriate grid size is of vital importance. Three different mesh element sizes were applied to carry out the mesh independence study, From Table 1, it can be seen that there is a difference in pressure drop of about 8.7% between the pressure drop obtained for the 5 mm & 2 mm valve mesh size. Based on this, the 5-mm pipe region mesh and the 2-mm valve body mesh were decided as the best to be adopted.

Table 1: Mesh independence study of the valve at 100% VOP.

Pipe mesh size (mm)	Valve mesh size (mm)	Number of mesh Elements	Pressure at Inlet (Pa)	Pressure at Outlet (Pa)	Outlet pressure difference (%)
5	3	2232055	4544.886	527.444	
5	2	6752356	4918.658	518.738	8.7%
4	2	7721905	5046.233	521.854	2.76%

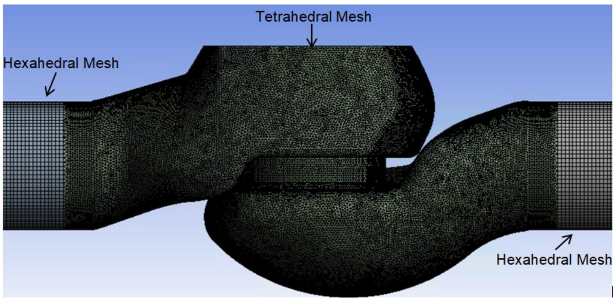


Figure 3: Mesh of the valve flow domain

2.4. Solver settings

After an optimal mesh has been decided, the solver settings were selected in Fluent solver. These include the boundary conditions, turbulence model selection, and number of iterations. For the turbulence model, the K-omega SST was selected which has been shown to perform best for problems with complex geometry as well as high velocity and pressure gradients. Velocity inlet boundary conditions of 1 and 2 m/s were set while the output boundary condition was set as pressure outlet with a gauge pressure of 0 Pa. The turbulence specification method selected was intensity and hydraulic diameter with 5% turbulent intensity and 0.1-m hydraulic diameter. For each case and convergence was monitored using the normalised residuals for the continuity, momentum, and turbulence equations up to 10^{-5}.

3. Results and discussion

There were five VOPs to be used for each of the two velocities, which gave a total of 10 simulations carried out. The pressure and velocity contours of different VOPs at 1 m/s and 2 m/s inlet velocities will be examined in the following sections.

3.1 Experimental validation

Figure 4 shows the comparison between experimental flow coefficient (Cv) data with CFD simulation results. A ±20% error band was set, and it can be seen that most of the experimental and CFD Cv values are within the ±20% of each other.

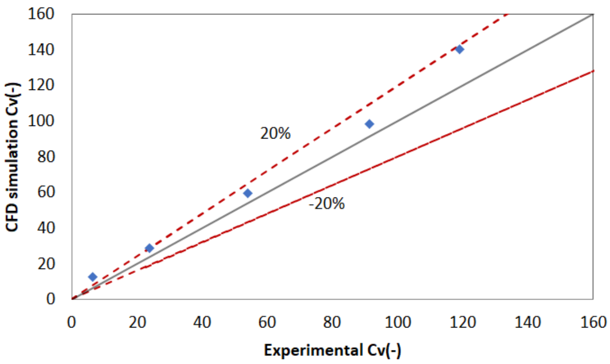
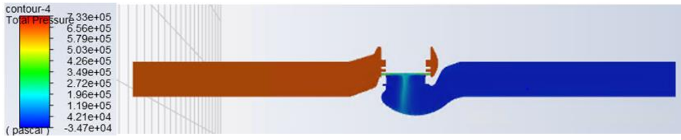


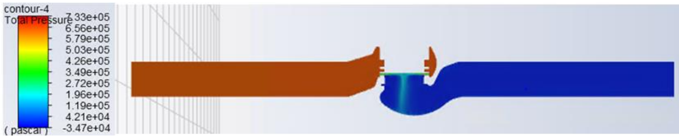
Figure 4. validation of numerical results of the valve's Cv with experiments.

3.2 Local pressure distributions

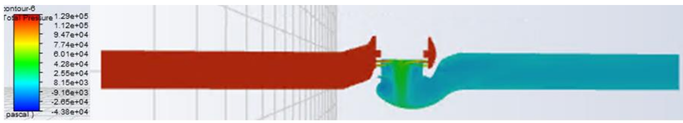
After the simulations of all five-valve openings with two velocities were carried out, the global flow condition and local flow characteristics can be analysed. Figure 5 shows the middle plane of the whole valve system with the total pressure distribution contours corresponding to each simulation. It can be seen that the control valve has a big influence on global pressure drop. When VOP is 20%, the pressure inlet is about 652000 Pa; then, because of the trim inside the valve body, there is significant pressured drop and the outlet pressure is 1700 Pa, resulting in the global pressure drop of 650300 Pa when inlet velocity is 1m/s. Similarly in Fig 5 (b), The pressure values at inlet and outlet is 2604000 Pa and 2400 Pa (pressure drop of 2601600 Pa), for the flow condition with same valve opening but higher flow velocity of 2 m/s. Similar trends are seen in figure 5 (c, d) at 60% valve opening. Additionally, when the valve is fully opened, as the VOP is 100%,the pressure drop is about 4400Pa when velocity is 1m/s and 17000 Pa for 2m/s.



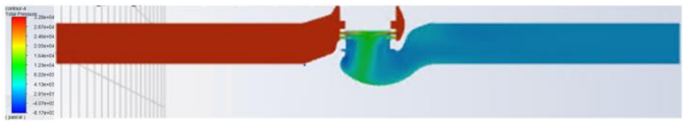
(a)



(b)

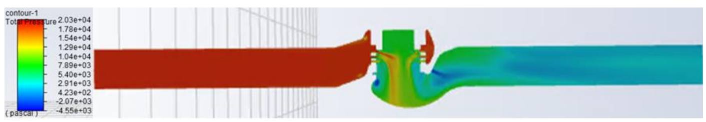


(c)



(d)

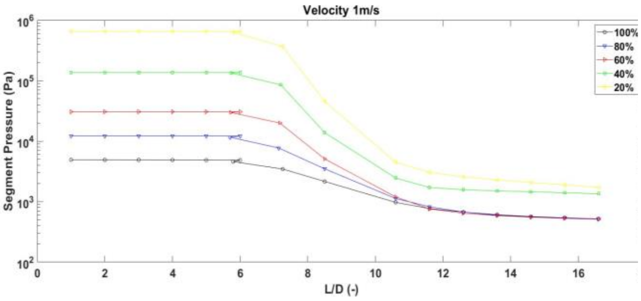
(e)



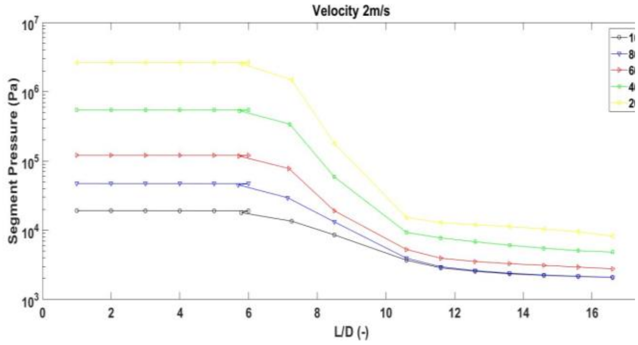
(f)

Figure 5: (a) (b) Contours of total pressure at 20% VOP with 1 m/s and 2 m/s velocity inlet, (c)(d) total pressure contours at 60% VOP with 1 m/s and 2 m/s, (e)(f) total pressure contours 100% open with 1 m/s and 2 m/s velocities at the inlet.

Thus, from the pressure contour results, it is apparent that nearer the trim most pressure drop takes place. To further quantify the areas which influence pressure field drastically further analysis has been carried out.



(a)



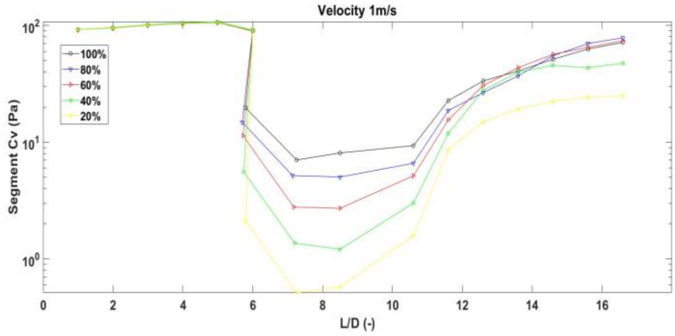
(b)

Figure 6: Local segment pressure at different locations within the valve at 20%,40%, 60% ,80% and 100% VOP with a) 1 m/s inlet velocity (b) 2 m/s inlet velocity

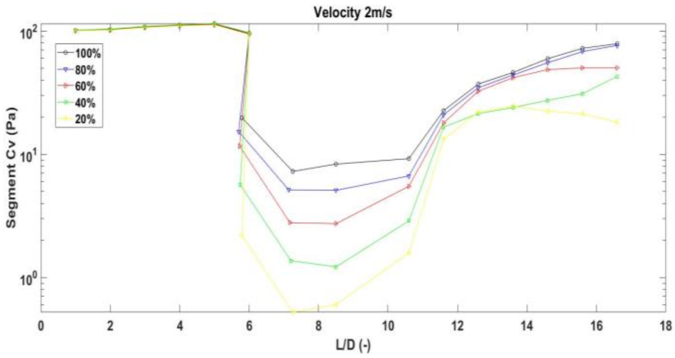
Fig. 6 (a) shows the pressure variation of the valve along the flow direction when the inlet velocity is 1 m/s. It can be seen that pressure continuously decreases in the direction of the flow. The pressure drop is higher for the low VOP and less for higher VOP. A similar trend is observed for the pressure at 2 m/s inlet boundary condition as shown in Fig. 6 (b) but with a significantly higher pressure drop. From the above figure it is clear that at higher velocity of 2 m/s the overall pressured drop is approximately four times the value observed at 1 m/s flow velocity.

3.1. Relationship between valve local pressure drop, flow area and VOP

Fig 7 (a) and (b) show the relationship between local segment Cv with the length of the valve system at 20%, 40%, 60%, 80% and 100% VOPs. This has been attempted as C_V is an important design parameters for the valves. By evaluating Cv locally it may be possible to modify the valve sections locally to get the desired global C_v. The plots indicate that when L/D is less than 6, the Cv can be treated as uniform (because uniform flow characteristics) as can be seen in the inlet pipe domain. At the valve and trim, the local Cv dramatically drops corresponding to L/D values of between 6 to 9. When flow passes to outlet pipe section there is a recovery in local Cv and it is increased. This Cv characteristic seen is mainly because of the complex geometrical shape within the valve resulting in higher a pressure drop.



(a)



(b)

Figure 7: (a) Local flow coefficient C_v at 1 m/s inlet velocity (b) Local flow coefficient Cv at 2 m/s inlet velocity

As explained in the earlier section, the factors that affect pressure drop in valve system are valve open percentage, The flow characteristics, and the valve geometry. Thus, to build the 1-D correlation equation to predict the valve Cv along the entire length of the system, the input parameter used would be length-diameter ratio, valve open percentage and the Reynolds number which represents the flow characteristics.

4. Prediction of local flow parameters incorporating the local of the geometry

4.1. One-dimensional local Cv representation

In the simulations carried out, the independent variables are the two fluid inlet velocities 1 m/s, 2 m/s and five VOPs. Fifteen local planes were created to extract the mass flow rate, pressure value, velocity, and these were used to calculate the local Cv at the respective L/D locations. In order to establish a reliable 1-D model, the least squares method was used to calculate an optimal regression curve, which predicts the local flow coefficient Cv as a function of the Reynolds number, VOP, and length–to–diameter ratio. Thus, through a power-law relationship, the form of the equation used to fit the data is as follows:

$$\mathrm{K} = A\left(\frac{L}{D}\right)^{a} \times Re^{c} \times VOP^{d} \tag{1}$$

$$Cv = K \times Ae \tag{2}$$

where Ae is the available flow area at the L/D locations. Five VOPs 20%, 40%, 60%, 80%, and 100% were represented in non-dimensional form in the correlation as 0.2, 0.4, 0.6, 0.8 and 1 respectively. The coefficient A and indices a, b, c are regression constants. In order to find the values of the constraints that best fit the data, the problem (Eqn. 1) is formulated as a least squares problem where $\sum[\mathrm{K}_{\mathrm{i,exp}} - \{A\,(L/D)_i{}^{a} \times Re_i^{c} \times VOP_i^{d}\}]^2$ is minimised. Initial guesses of the constants were provided before using the Excel solver to find the values that produce the best goodness of fit with the data. After the regression procedure, the following equation was derived:

$$K = 402.43 \times (L/D)^{2.61} \times Re^{-0.20} \times VOP^{0.0016} \quad (3)$$

As $Cv = K \times Ae$ with Ae being the equivalent area of the local planes, the 1-D correlation equation used to predict the local flow coefficient Cv_{local} along the pipe–valve system is:

$$Cv_{local} = 402.43 \times (L/D)^{2.61} \times Re^{-0.20} \times VOP^{0.0016} \times Ae \quad (4)$$

Fig. 8 shows the Cv prediction by this equation when compared to the CFD values. As can be seen, the predicted value by using this equation gives a high R^2 value of 0.8273, and most of the samples are within a ±30 band range. Thus, the new correlation can calculate the local flow coefficient along the whole valve geometry for this valve and the particular trim.

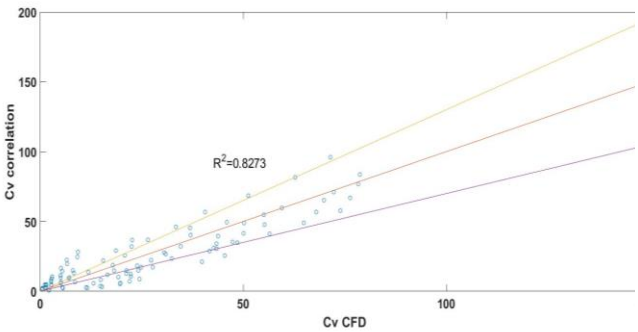


Figure 8: Comparison of 1-D Cv model prediction with with Cv computed from CFD

5. Conclusions

This study evaluated the variation of valve flow coefficient (Cv) along a valve and pipe system with a specific trim geometry. A 1-D numerical model has been developed that can be used to predict the local flow coefficient Cv. Two different velocities and five different valve opening positions were used as independent variables for carrying out CFD simulations using the Ansys Fluent simulation software. Values of water pressure, velocity, and mass flow rate were extracted from 15 different segments of the whole valve. These were used to calculate the local Cv along the valve and pipeline system from the inlet through the valve and trim to the outlet. The results show that there is a dramatic drop in pressure at the trim for all the VOPs and fluid velocities studied. This is because of the flow restriction caused by the trim geometry. However, this property allows flow control to be achieved by the valve. Correspondingly, the local Cv within the valve was observed to change, and not constant as was the case in the pipe sections. Based on these observations, a one-dimensional relationship of the local Cv was generated with the CFD data, which was shown to match the CFD data closely. It is envisaged that similar 1-D model could be developed for other valve/trim geometries, and these could be used by designers to rapidly assess various valve configurations for their usefulness.

References

[1] W. Valves, "Blakeborough Cage & Top Guided Valves," *Direct Industry*, p. 17, 2017.

[2] Y. J. An, B. J. Kim, and B. R. Shin, "Numerical analysis of 3-D flow through LNG marine control valves for their advanced design," *J. Mech. Sci. Technol.*, vol. 22, no. 10, pp. 1998–2005, Oct. 2008.

[3] J. Green, R. Mishra, M. Charlton, and R. Owen, "Validation of CFD predictions using process data obtained from flow through an industrial control valve," *J. Phys. Conf. Ser.*, vol. 364, no. 1, pp. 0–6, 2012.

[4] W. C. Kwon, G. R. Kim, S. C. Park, and J. Y. Yoon, "Design of a tortuous path trim for a high-pressure turbine bypass valve," *Proc. Inst. Mech. Eng. Part E J. Process Mech. Eng.*, vol. 224, no. 2, pp. 149–153, 2010.

[5] T. Asim, M. Charlton, and R. Mishra, "CFD based investigations for the design of severe service control valves used in energy systems," *Energy Convers. Manag.*, vol. 153, pp. 288–303, Dec. 2017.

[6] T. Asim, A. Oliveira, M. Charlton, and R. Mishra, "Improved design of a multi-stage continuous-resistance trim for minimum energy loss in control valves," *Energy*, vol. 174, pp. 954–971, May 2019.

[7] D. Singh, A. M. Aliyu, M. Charlton, R. Mishra, T. Asim, and A. C. Oliveira, "Local multiphase flow characteristics of a severe-service control valve," *J. Pet. Sci. Eng.*, vol. 195, Dec. 2020.

[8] V.C. Agarwal and R Mishra, "Optimal design of a multi-stage capsuile handling multi-phase pipeline" *International Journal of Pressure Vessel and Piping*, vol. 74, Jnauary 1998.

[9] M. Altwieb, K. J. Kubiak, A. M. Aliyu, and R. Mishra, "A new three-dimensional CFD model for efficiency optimisation of fluid-to-air multi-fin heat exchanger," *Therm. Sci. Eng. Prog.*, vol. 19, Oct. 2020.

[10] A. M. Aliyu, D. Singh, C. Uzoka, and R. Mishra, "Dispersion of virus-laden droplets in ventilated rooms: Effect of homemade facemasks," *J. Build. Eng.*, vol. 44, p. 102933, Dec. 2021.

[11] BS EN 60534-2-1, "Industrial-process control valves. Part 2–1: Flow capacity – Sizing equations for fluid flow under installed conditions." 2011.