Technical Sciences, 2024, 27, 19–31

UNIWERSYTET WARMIŃSKO-MAZURSKI W OLSZTYNIE

Yearbook peer-reviewed scientific journal

ISSN 1505-4675
e-ISSN 2083-4527

TECHNICAL SCIENCES

Homepage: https://czasopisma.uwm.edu.pl/index.php/ts/

DOI: https://doi.org/10.31648/ts.9732

COMPARISON OF PRESSURE-LOSS EVALUATION FIDELITY IN TURBULENT ENERGY DISSIPATION MODELS OF POPPET CHECK VALVES USING COMPUTATIONAL FLUID DYNAMICS (CFD) SOFTWARE

***Maciej Jerzy Kobielski*[1], *Wojciech Skarka*[2], *Michał Skarka*[3]**

[1]ORCID: 0009-0009-6202-4749
Sanhua-Aweco in Tychy
Silesian University of Technology in Gliwice

[2]ORCID: 0000-0003-3989-7751
Silesian University of Technology in Gliwice

[3]ORCID: 0000-0002-1202-0426
Faculty of Aerospace Engineering
Delft University of Technology, Netherlands

Received 11 December 2023, accepted 2 February 2024, available online 12 February 2024.

Keywords: CFD; check valve; computational fluid dynamics, Ansys Fluent; digital twin; systems engineering; turbulence model.

Abstract

Check valves are critical components of fluid systems and have various applications, including house appliances. This article presents a methodology for mapping geometry-specific constriction pressure loss as a function of flow and turbulence in a check valve. This study aimed to gain insight on which Ansys Fluent available turbulent energy dissipation model should be used for further design optimization. This methodology consists of a statistical comparison of computational fluid dynamics (CFD) simulation results obtained using the turbulent energy dissipation models. The key components of the simulation process are discussed. The study's main results are a comparison of empirical results among flow models' estimated pressure loss, shown as a function of flow rate in specific geometry and identification of the most suitable model for the considered application. This study concludes that the K-Epsilon (Standard) model best represents the empirically measured behavior of naturally occurring flow energy losses in the considered geometry.

Correspondence: Maciej Jerzy Kobielski, Sanhua-Aweco, ul. Turyńska 80, 43-100 Tychy, Poland, e-mail: maciej.kobielski@sanhua-aweco.com; wojciech.skarka@polsl.pl; michal.skarka@gmail.com

Nomenclature:
CFD – Computational flow dynamics
TI – Turbulence intensity
TVR – Turbulence viscosity ratio

Introduction

Check valves are used to control the hydraulic behavior of fluids in various systems. The scope of research on the applicational behavior of such valves includes nuclear power plants (HAYNES 1992, MCELHANEY 2000, TURESSON 2011), forestry equipment (NEDIĆ et al. 2017), high-pressure gas systems (YE et al. 2020), rehabilitation devices (ŻYŁKA et al. 2023), and household water safety applications (PARK 2009). Regulatory compliance for household appliance products often requires a method for preventing the backflow of fluids, not only for water taps but also for various household appliances using tap water, such as a dishwasher (EN 61770:2009/A1:2019 Electric Appliances Connected to the Water Mains – Avoidance of Backsiphonage and Failure of Hose-Sets, n.d.). Various types of valve construction have been considered for use in state-of-the-art appliances, including swing-based check valves, dual plate (wafer) check valves, and nozzle check valves (SIBILLA, GALLATI 2008). Poppet-type check valves, which can be attributed to the nozzle check-valve group, are commonly used in household appliance fluid systems to prevent the backflow of process-used water into the wholesome tap water stream.

The poppet-type check-valve type relies on the flow-pressure-driven movement of a spring-loaded piston to control the flow of fluid and ensure that only a single direction of flow can be achieved. What is disadvantageous however, is that the piston geometry in the poppet check-valve device will cause a certain hydraulic loss in the system (LANEY, FARRELL 2018), the design of which is a complex engineering task due to the complex physics behind the behavior of flow (YANG et al. 2011, LISOWSKI, FILO 2017). The goal of this research was to prepare a digital twin according to the definition of the digital twin proposed in previous publications (*Forging the Digital Twin in Discrete Manufacturing: A Vision for Unity in the Virtual and Real Worlds*, RITURAJ, SCHEIDL 2023) for computational optimization of the hydraulic efficiency of the system by following methods proposed in recent publications (LISOWSKI et al. 2015, PECIAK, SKARKA 2022).

Numerical methods of various types can be used for the design of check-valve systems, including computational flow dynamics (CFD) (LISOWSKI, FILO 2017). CFD is a powerful tool for simulating the behavior of fluids in such systems. Ansys Fluent is one of the most widely used CFD software packages in the engineering market. (*Computational Fluid Dynamics (CFD) Software Market Size & Share Analysis 2023-2030* n.d.). The accuracy of these simulations depends on the choice of turbulence model used to describe the turbulent flow behavior (SIBILLA, GALLATI 2008). In this article, we investigate the usability of readily available turbulence models and minimally adapted models' parameters of Ansys Fluent software for simulating the flow behavior in poppet-type check valves to achieve the closest replication of empirically measured real-life behavior.

Materials and methods

Pressure-loss behavior was investigated using a 3D CFD model incorporating a production-specific geometry of a poppet-type check valve and test-specific tubing, later used in the empirical evaluation testing of pressure loss.

CAD and numerical model

The test specimen was a poppet-type check valve with a patent-protected triple seal type, as shown in Figure 1.

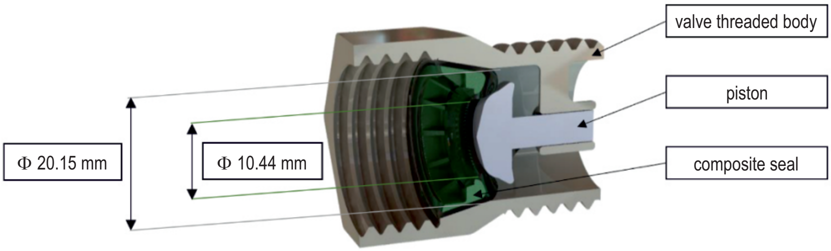

Fig. 1. CAD cross-section of the poppet check-valve specimen with marked constriction point (green lines) and large pipe diameter (grey lines)

The nominal geometry of the valve consists of an external body threaded with standardized ¾ inch NTS taper threads – female thread at the inlet and male at the outlet and an integrated bushing for linear piston movement; a 2-component injection seal fixed inside the body by the compression of an interfacing inlet pipe thread; and a concentrically positioned piston capable of movement on the central axis of the valve inside the housing-integrated bushing.

The CAD model used as the meshing basis for CFD analysis, shown in Figure 2, consisted additionally of two interfacing pipes of 100 mm length each, via which the inlet and outlet flows to the valve were realized, and for which the static pressure results were evaluated for accuracy.

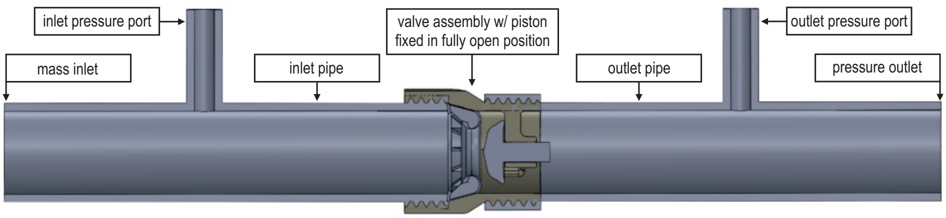

Fig. 2. CAD cross-section of test specimen arrangement

Whereas a nominal production assembly would also incorporate a spring placed between the piston and the bushing, due to the goal of evaluating the hydraulic losses in the system of fixed fully open geometry, the spring was not considered in the simulated CAD assembly, following previous studies (GOMEZ et al. 2019).

Computational mesh

The CAD geometry shown in Figure 2 was used as the basis for the generation of a fluid domain of the fully open check valve and pipe set, as shown in Figure 3.

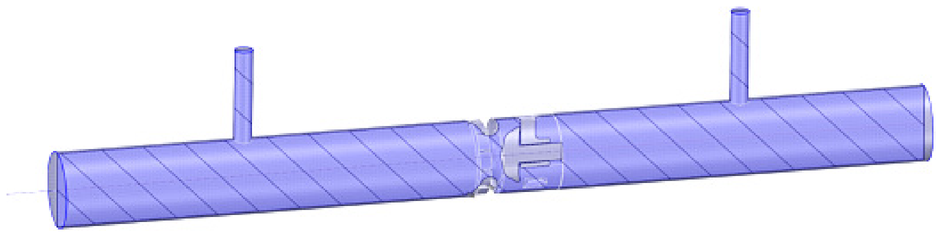

Fig. 3. Fluid domain in the Ansys Space Claim application

The experiment fixed model mesh was defined as a fluid domain, without a definition of the interfacing walls. The mesh consisted of 5,043,723 cells, in poly-hexcore configuration, with four layers for wall effects evaluation. A cross-section of the mesh was shown in the Figure 4 and a close-up of the mesh refinement in piston area was shown in Figure 5.

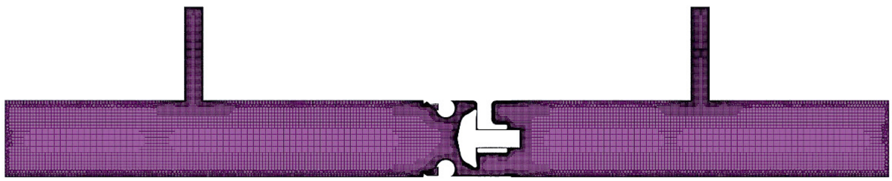

Fig. 4. A cross-section of flow domain poly-hexcore mesh

The criteria for mesh evaluation were orthogonal quality >0.71, skewness <0.5, and aspect ratio <8.7. These values were based on the existing state of publications (FILO et al. 2021).

A mesh independence study was carried out in scenarios of 4 mesh densities, with cell count at respectively 69%, 82%, 100% (model presented in Figures 4 and 5) and 114% of study-used mesh, at 2 l/min condition in 2nd order k-Omega

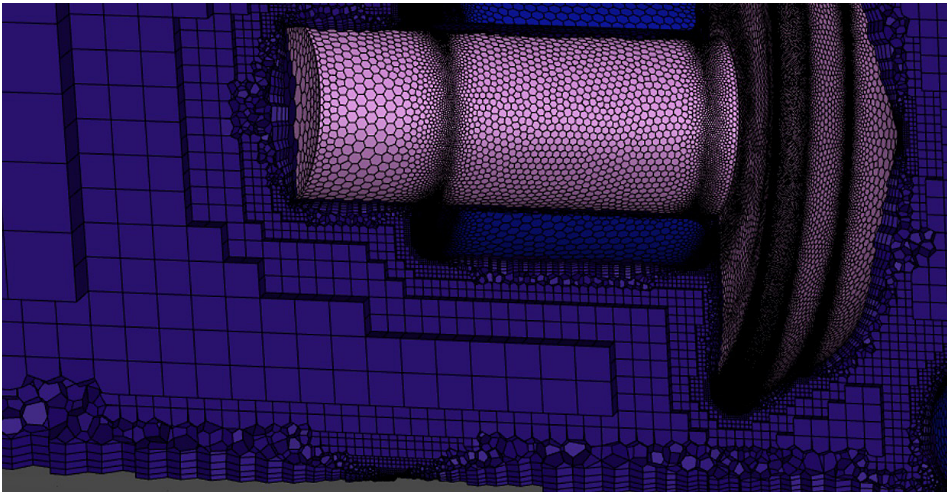

Fig. 5. A close-up of cross-section of flow domain poly-hexcore mesh in the poppet area

SST energy dissipation scenario, with the parameters evaluated being the inlet pressure build-up at 0 Pa gauge pressure outlet and the volumetric average of volumetric turbulent intensity in the model. The criteria for mesh independence evaluation were convergence of the turbulent energy volume integral of parameter (*k*) and pressure drop being below 3% with step of mesh refinement. The high independence of volumetric turbulent parameter (*k*) observed at below 1% indicated the mesh to be particularly suitable for evaluation of the internal turbulent model behavior.

The mesh independence study results were plotted in Figure 6, showing the 5,04 Mio cell model to be mesh independent for the defined criteria.

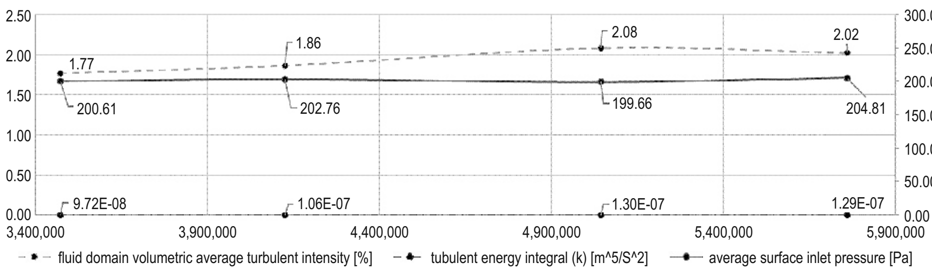

Fig. 6. Mesh Independence Study results for evaluation of mesh suitability for behavioural evaluation of valve system by CFD means

Turbulent models

A series of six simulations of at least 30 iterations per simulation was performed for each of the following turbulent energy dissipation models:

– K-Omega SST,
– K-Omega Standard,
– K-Epsilon Realizab.le,
– K-Epsilon Standard

The residuals of energy were expected in the frame <10e-06, with most results obtained at <10e-10.

CFD border conditions

The CFD model was evaluated as a series of evaluation points for calculated pressure loss between the inlet-side pressure connector and outlet-side atmospheric pressure.

The border conditions were defined as specifically expected in the domain of white goods' check valves, being 1-6 l/min flow with 1 l/min increments, and with no external pressure at the wide outlet point for simplification of empirical measurement. The temperature conditions reflected the average test water of 17°C. The geometry point of the evaluation was fixed at a fully opened valve based on previously run, non-published studies showing a complete opening of the valve for flows <<1 l/min using the product-specific spring parameters. The fluid domain was defined as Ansys Fluent pre-configured "liquid-water", with no dissolved fractions. The turbulent parameters at the inlet were set with turbulence specified as turbulence intensity (TI) and viscosity ratio, with TI set at 1% and turbulent viscosity ratio (TVR) set at 0.7 [-], which was a deviation from the application pre-configured TI=5% and TVR=10 [-].

Physical empirical testing model

The empirical evaluation of real-life behavior was tested on mixed manufacturing technology prototypes:

The housing of the valve, piston, seal core, and measurement setup tubing, due to expected water tightness and surface finish impact (Ramanath, Chua 2006) were MSLA-printed using the Elegoo Saturn 2 printer.

The seal was formed using custom tooling for 2k prototype 2-component silicone RTV-2 compression molding, with the tool halves and seal core, manufactured in MSLA technology, as shown in Figure 7. The complete prototype valve assembly was shown in Figure 8.

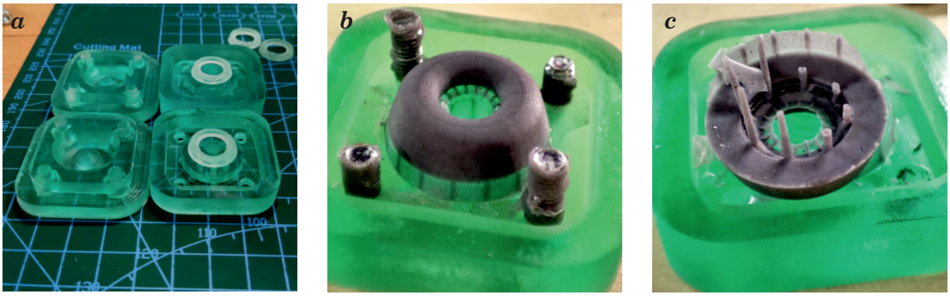

Fig. 7. Set of compression molds and cores (*a*), demolded part (*b*), demolded and removed part before post-processin (*c*)

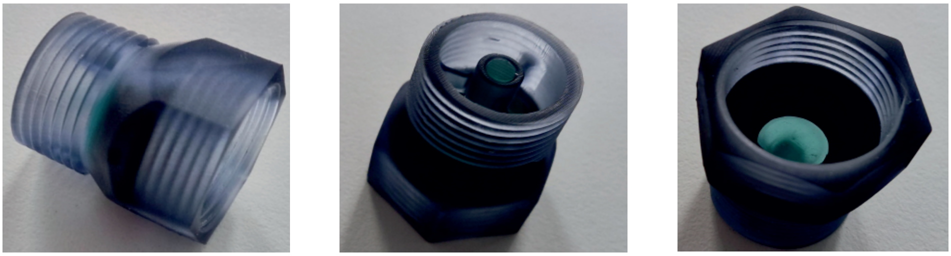

Fig. 8. Three views of the prototype check-valve complete assembly

Physical test bench arrangement

The pressure loss was measured using a pressure sensor device in the hydraulic configuration shown in Figure 9.

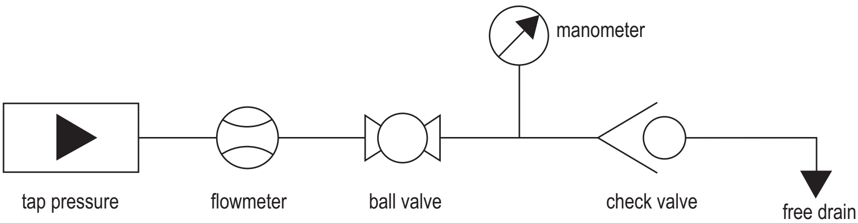

Fig. 9. Hydraulic arrangement of the measurement setup

The water flow measurement was performed using an SM6120 Magnetic Inductive Flowmeter (resolution ±0.02 l/min), and pressure measurement was performed on a calibrated pressure transmitter XMLGB05L73SBM(resolution ±0.3%), with connection via a 200 ml buffer container for evaluation of possible pressure leaks. The complete setup was shown in Figure 10.

The measurements were performed in six steps by varying the pre-set water flow from 1 l/min to 6 l/min in 1 l/min steps and taking the average result of four measurements per step.

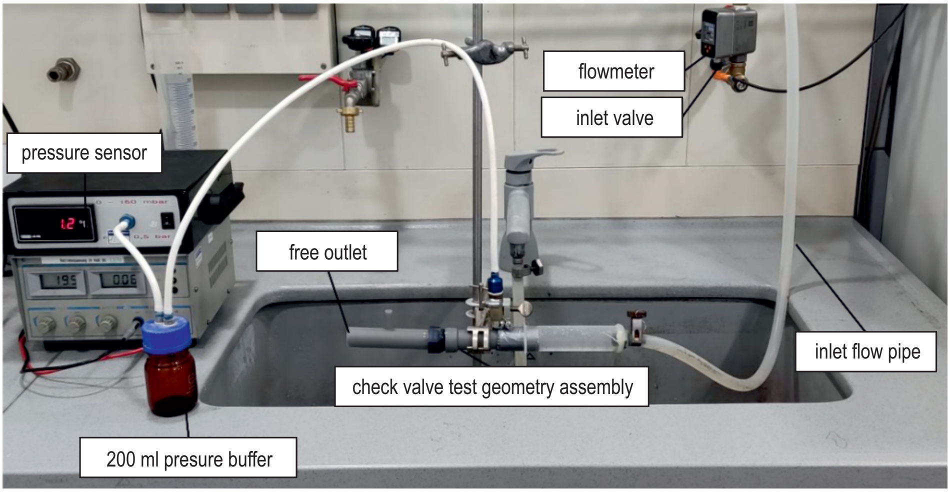

Fig. 10. Test station with the test device

Data analysis

The obtained test results were subject to analysis of the mean square deviation between the simulated results using each turbulence model and the empirically obtained results of pressure loss in the laboratory setup. Each stock turbulent energy dissipation loss model was evaluated based on a comparison of these mean square deviations. The cross section visualization of the turbulent intensity results for 6 l/min flow were also evaluated respective to the calculated Reynold's number flow regime prediction.

Results

The obtained results of pressure loss for all simulation variants and for the averaged empirical measurements in Pa were shown in Table 1 and Figure 11.

Table 1

Pressure loss data points for all data sources

Data source / flow conditions	1 l/min	2 l/min	3 l/min	4 l/min	5 l/min	6 l/min
Simulation – K-Omega SST	23.13	88.47	199.62	350.71	533.75	795.43
Simulation – K-Omega Standard	23.12	88.94	198.81	361.60	569.29	825.39
Simulation – K-Epsilon Standard	32.01	88.70	175.02	276.79	425.68	581.25
Simulation – K-Epsilon Realizable	22.19	85.51	187.52	331.55	511.56	733.58
Empirical measurement (mean of 4, rounded to 0.5)	34	76.5	173	308	439	608.5

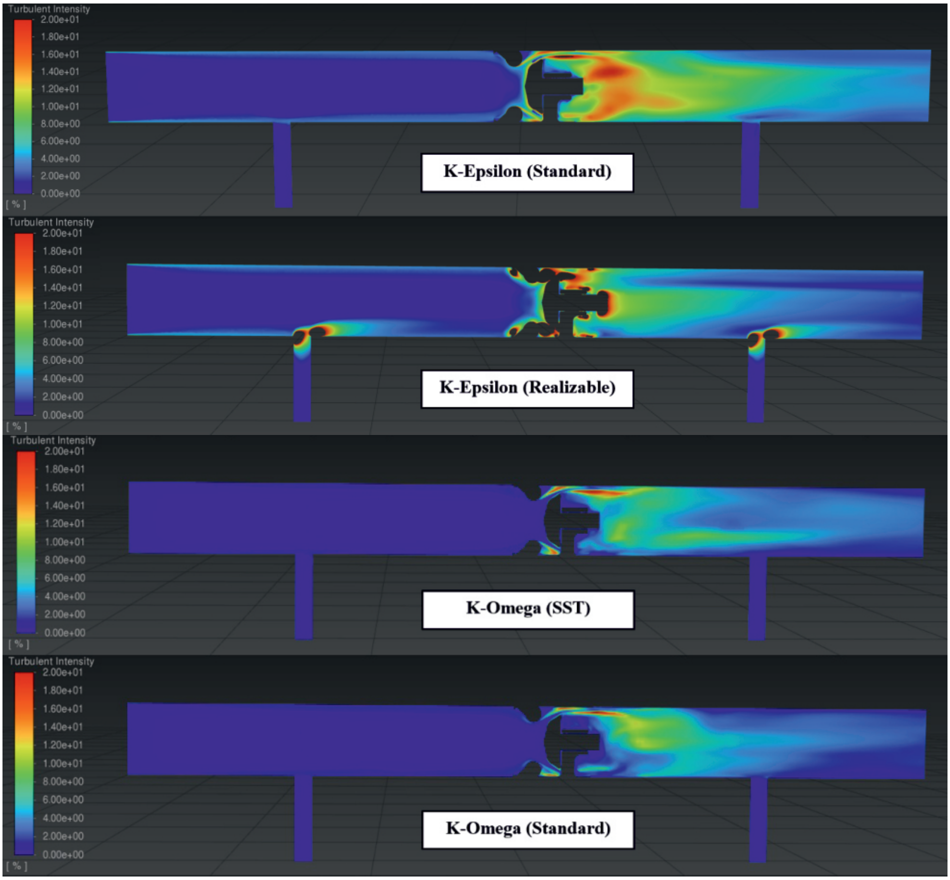

Fig. 11. Turbulent intensity contours for 6 L/min for all evaluated models

The square deviations relative to the empirically obtained mean results were shown in Table 2.

Table 2
Square deviations of CFD results' data points relative to empirical measurements

Simulation type / flow conditions	1 l/min	2 l/min	3 l/min	4 l/min	5 l/min	6 l/min
Simulation – K-Omega SST	0.10566	0.02439	0.02199	0.01841	0.06575	0.09478
Simulation – K-Omega Standard	0.10022	0.03200	0.02876	0.03753	0.07386	0.10896
Simulation – K-Epsilon Standard	0.00344	0.02545	0.00014	0.01027	0.00095	0.00199
Simulation – K-Epsilon Realizable	0.12065	0.01386	0.00705	0.00585	0.02714	0.04233

The averaged deviations relative to the empirically obtained averaged results were shown in Table 3. For evaluation purposes, the simulation results were shown in Figure 11 as a uniform-legend TI plot in axial cross-section for the condition of 6 L/min flow.

Table 3
Averaged deviations of CFD models relative to empirical measurements

Turbulent model	Averaged square deviation [%]
Simulation – K-Omega SST	5.52
Simulation – K-Omega Standard	6.36
Simulation – K-Epsilon Standard	0.70
Simulation – K-Epsilon Realizable	3.61

Discussion

Pressure-loss comparison is a commonly used method for evaluating the performance of turbulent models (GOMEZ et al. 2019). The comparison between the predicted pressure loss and experimental data can provide insights into the accuracy of the model in predicting real-life behavior. In this study, we found that the K-Epsilon model performed better than the K-Omega model in predicting pressure loss, contrary to some publications (HUOVINEN et al. 2015). This could be because the K-Epsilon model is more suited to flows with high TI, whereas the K-Omega model is better suited to low TI flows. It is also notable, that for the lowest flow value of 1 [l/min] and the lowest expected turbulence, the K-Epsilon Standard model behaved the best. However, there is no clear position on this matter: the K-Epsilon model is often used in valve modeling because it considers the effects of rotation and curvature of the wall. RANS models are the most widely used approach for industrial flows.

The calculated Reynolds' number in terms of flow regime in the connection pipe areas for 6 L/min, calculated at Re=580 [-], aligned with all model predictions shown by turbulent intensity contours with largely laminar behavior and a noticeable increase in wall-adjacent turbulent intensity. The only deviation from the Reynolds-based regime prediction was observed in the k-epsilon (realizable) contour, in which the model visibly induced eddies of turbulent intensity >20% in areas of sharp edges.

It is important to note that the choice of turbulent model depends on the specific problem being solved, and other factors such as computational cost and accuracy requirements should be considered.

Conclusions

In this study, we compared the performance of the K-Epsilon and K-Omega turbulent models in two variants each to predict the pressure loss of fluid flow through a fully open poppet-type check-valve assembly. The pressure-loss comparison method proved to be effective in evaluating the performance of the models (LISOWSKI et al. 2015). Our results suggest that the K-Epsilon Standard model performs best, with a mean square deviation from real-life behavior of 0.70%. In order of further suitability was the K-Epsilon Realizable model with an averaged square deviation of 3.61%; K-Omega SST with an averaged square deviation of 5.52%, and K-Omega Standard with an averaged square deviation of 6.36%.

The behaviour of K-Omega SST, K-Omega Standard and and K-Epsilon Standard numerical model in the described geometry can be trusted to match the flow regime expectation derived by Reynolds' number calculation. It is worth noting however that an empirical visual evaluation of the real-life behavior with use of strobe-light based video registration and a color stream marker could be a more suitable way to evaluate the mimicry of real fluid micro-vortices than a pressure-loss-based method.

What is also worth noting, is that evaluation of meshing suitability has shown the meshing density suitable for evaluation of volumetric pressure behavior (variants reduced by 18% and 31%) would not have been suitable for evaluation of turbulent energy dissipation, as a smaller size mesh would not result in result independence for these parameters. It leads to conclusion, that the any mesh independence study should be suited for the modelled parameters in mind, not only to model tuning calibration parameter.

Nevertheless, for purposes of hydraulic system engineering in conditions of flow preventing creation of cavitation phenomena, this model could be considered a digital twin of the real-life model (WRIGHT, DAVIDSON 2020).

Funding sources

This research was co-financed by the Ministry of Science and Higher Education, Republic of Poland, within the "Doktorat wdrożeniowy" program, 4th edition. The research was supported by the Polish National Centre of Research and Development from the project POIR.01.01.01-00-1408/20.

References

Computational Fluid Dynamics (CFD) Software Market Size & Share Analysis 2023-2030. n.d. Retrieved from https://www.marketwatch.com/press-release/computational-fluid-dynamics-cfd-software-market-size-share-analysis-2023-2030-2023-04-19 (02.05.2023).

Forging the Digital Twin in Discrete Manufacturing: A Vision for Unity in the Virtual and Real Worlds. n.d. Retrieved from https://www.lnsresearch.com/research-library/research-articles/ebook-forging-the-digital-twin-in-discrete-manufacturing-a-vision-for-unity-in-the-virtual-and-real-worlds (05.06.2023).

EN 61770:2009/A1:2019. Electric Appliances Connected to the Water Mains – Avoidance of Back-siphonage and Failure of Hose-Sets. n.d.

Filo G., Lisowski E., Rajda J. 2021. *Design and Flow Analysis of an Adjustable Check Valve by Means of CFD Method*. Energies, 14(8): 2237. https://doi.org/10.3390/en14082237

Gomez I., Gonzalez-Mancera A., Newell B., Garcia-Bravo J. 2019. *Analysis of the Design of a Poppet Valve by Transitory Simulation*. Energies, 12(5): 889. https://doi.org/10.3390/en12050889

Haynes H.D. 1992. *Evaluation of Check Valve Monitoring Methods*. Nuclear Engineering and Design, 134(2–3): 283–294. https://doi.org/10.1016/0029-5493(92)90146-M

Huovinen M., Kolehmainen J., Koponen P., Nissilä T., Saarenrinne P. 2015. *Experimental and Numerical Study of a Choke Valve in a Turbulent Flow*. Flow Measurement and Instrumentation, 45: 151–161. https://doi.org/10.1016/j.flowmeasinst.2015.06.005

Laney M., Farrell R. 2018. *Piston-Lift Check Valve Flow Verification Using CFD*. In: *ASME 2018 Pressure Vessels and Piping Conference*. Volume 7: *Operations, Applications, and Components*. Prague. https://doi.org/10.1115/PVP2018-84672

Lisowski E., Filo G. 2017. *Analysis of a Proportional Control Valve Flow Coefficient with the Usage of a CFD Method*. Flow Measurement and Instrumentation, 53: 269–278. https://doi.org/10.1016/j.flowmeasinst.2016.12.009

Lisowski E., Filo G., Rajda J. 2015. *Pressure Compensation Using Flow Forces in a Multi-Section Proportional Directional Control Valve*. Energy Conversion and Management, 103: 1052–1064. https://doi.org/10.1016/j.enconman.2015.07.038

Mcelhaney K.L. 2000. *An Analysis of Check Valve Performance Characteristics Based on Valve Design*. Nuclear Engineering and Design, 197(1–2): 169-182. https://doi.org/10.1016/S0029-5493(99)00264-2

Novak N., Pršić D., Fragassa Ch., Stojanović V., Pavlovic A. 2017. *Simulation of Hydraulic Check Valve for Forestry Equipment*. International Journal of Heavy Vehicle Systems, 24(3): 260. https://doi.org/10.1504/IJHVS.2017.084875

Park S-H. 2009. *Development of a proportional poppet-type water hydraulic valve*. Proceedings of the Institution of Mechanical Engineers. Part C: Journal of Mechanical Engineering Science, 223(9): 2099–2107. https://doi.org/10.1243/09544062JMES1380

Peciak M., Skarka W. 2022. *Assessment of the Potential of Electric Propulsion for General Aviation Using Model-Based System Engineering (MBSE) Methodology*. Aerospace, 9(2): 74. https://doi.org/10.3390/aerospace9020074

Ramanath H.S., Chua C.K. 2006. *Application of Rapid Prototyping and Computational Fluid Dynamics in the Development of Water Flow Regulating Valves*. The International

Journal of Advanced Manufacturing Technology, 30(9–10): 828–35. https://doi.org/10.1007/s00170-005-0119-5

RITURAJ R., SCHEIDL R. 2023. *Towards Digital Twin Development of Counterbalance Valves: Modelling and Experimental Investigation*. Mechanical Systems and Signal Processing, 188: 110049. https://doi.org/10.1016/j.ymssp.2022.110049

SIBILLA S., GALLATI M. 2008. *Hydrodynamic Characterization of a Nozzle Check Valve by Numerical Simulation*. Journal of Fluids Engineering, 130(12): 121101. https://doi.org/10.1115/1.3001065

TURESSON M. 2011. *Dynamic Simulation of Check Valve Using CFD and Evaluation of Check Valve Model in RELAP5*. Master of Science Thesis (Nuclear Engineering). Department of Chemistry and Bioscience Division of Chemical Reaction Engineering. Chalmers University of Technology, Göteborg. Retrieved from https://publications.lib.chalmers.se/records/fulltext/142017.pdf

WRIGHT L., DAVIDSON S. 2020. *How to Tell the Difference between a Model and a Digital Twin*. Advanced Modeling and Simulation in Engineering Sciences, 7(1): 13. https://doi.org/10.1186/s40323-020-00147-4

YANG Q., ZHANG Z., LIU M., HU J. 2011. *Numerical Simulation of Fluid Flow inside the Valve*. Procedia Engineering, 23: 543–50. https://doi.org/10.1016/j.proeng.2011.11.2545

YE J., ZHAO Z., ZHENG J., SALEM S., YU J., CUI J., JIAO X. 2020. *Transient Flow Characteristic of High-Pressure Hydrogen Gas in Check Valve during the Opening Process*. Energies, 13(16): 4222. https://doi.org/10.3390/en13164222

ŻYŁKA M., MARSZAŁEK N., ŻYŁKA W. 2023. *Numerical Simulation of Pneumatic Throttle Check Valve Using Computational Fluid Dynamics (CFD)*. Scientific Reports, 13(1): 2475. https://doi.org/10.1038/s41598-023-29457-4