SEMARAK ILMU
PUBLISHING

Journal of Advanced Research in Fluid
Mechanics and Thermal Sciences

Journal homepage:
https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/index
ISSN: 2289-7879

Analysis of Fluid Pressure Drop through a Globe Valve using Computational Fluid Dynamics and Statistical Techniques

Mauricio Rhenals[1], Armando Robledo[1], Jonathan Fábregas[1,*], Javier Carpintero[2]

[1] Facultad de Ingeniería, Maestría en Ingeniería Mecánica, Universidad Autónoma del Caribe, Barranquilla, Colombia
[2] Department of Civil and Environmental, Universidad De La Costa, Calle 58 #55-66, 080002 Barranquilla, Atlántico, Colombia

ARTICLE INFO

Article history:
Received 26 October 2023
Received in revised form 24 February 2024
Accepted 6 March 2024
Available online 30 March 2024

Keywords:
Computational Fluid Dynamics (CFD); DOE statistical techniques; globe valve; response surface; roughness

ABSTRACT

Fluid mechanics plays a crucial role in everyday life, enabling the selection of accessories, materials, and various components essential for a system through which fluid flows. Pressure drop stands out as one of the most relevant factors in the design of fluid flow systems. However, analytical and experimental physical methods can increase these analyses' costs and time. Hence, in this study, statistical tools are employed to carry out specific experiments supported by numerical fluid simulation, aiming to comprehend the pressure drop behavior in a fluid as it passes through a globe valve. This valve, in turn, possesses distinct operating and manufacturing characteristics. The methods employed encompass a complete factorial system of response surface as support to construct the experimental design path through computational fluid dynamics. Among the key findings, it is demonstrated that, for systems with relatively low flow rates, the valve opening percentage does not exhibit a significant relationship with fluid pressure drop. Conversely, significant effects are observed for systems with relatively high flow rates regarding the valve opening percentage and pressure drop, reaching values of up to 73% pressure drop in this study. It can be inferred that the integration of statistical experimental design techniques and computational fluid dynamics constitutes a valuable resource for studying the pressure drop of a fluid passing through a system.

1. Introduction

The application of Computational Fluid Dynamics (CFD) in systems involving hydraulic valves is essential for enhancing performance, efficiency, and safety while concurrently reducing costs and development time. This approach enables a more precise and efficient focus on the design and operation of valves across various industrial and engineering applications.

Design and optimization through CFD allow the simulation of fluid flow through valves, analyzing how their design influences efficiency, pressure loss, control capacity, and lifespan. This facilitates the optimization of the design to meet the specific requirements of a given system, as presented in previous studies [1-3]. Moreover, by identifying areas where energy losses occur in the simulated

---

[*] *Corresponding author.*
E-mail address: jonathanfabregasv@gmail.com

https://doi.org/10.37934/arfmts.115.2.7082

system, these losses can be minimized, significantly impacting energy efficiency and operational costs.

CFD tools also contribute to cost reduction in development and testing, as adjustments can be made in a computational environment to achieve the expected results, thereby improving the performance and safety required by the design. Additionally, the application of Design of Experiments (DOE) in fluid mechanics aids in the analysis of developed tests, providing a valuable assessment of fluid behavior in simulated systems and optimizing performance efficiently and effectively, as indicated in previous studies [4-8].

Furthermore, CFD analysis allows for the study of the effects of hydraulic jumps caused by obstacles in the path of a fluid, significantly affecting its behavior. This leads to an increase in the Reynolds number and, consequently, an increase in fluid energy, as demonstrated in the work of Jasim *et al.,* [9]. CFD is widely employed in various studies to examine the thermal and pressure drop effects of fluid flow through corrugated or obstructed surfaces, as shown in previous studies [10-14].

Measuring the pressure drop of a fluid passing through pipes and fittings is crucial, as it impacts the power and energy consumption of devices driving the fluids, thereby influencing investment, operational costs, and maintenance, as indicated in previous studies [15-20].

In works such as Bu *et al.,* [21], the behavior of fluid flow through a control valve is simulated to obtain a model for predicting the flow coefficient through the valve, varying opening positions and flow velocities. In other studies, such as Garg *et al.,* [22] the effect of air pressure flowing through a pipe is determined by analyzing leakage behavior in the pipe, all through the application of CFD techniques. Reich's [23] work evaluates the behavior of steam flow through a valve by adjusting control input conditions, such as flow velocities. Serani *et al.,* [24] conducted a study using adaptive sampling methods linked to the CFD simulation process to optimize the performance of the dynamic fluid system they studied.

In studies such as those presented by Al-Obaidi *et al.,* [25-27] and Al-Obaidi [28], they develop simulated and experimental models of the behavior of fluid mechanics in axial pump systems, where the influence of the pressure difference behavior stands out in the process stages, reflecting a good approximation of the experimental and simulated study highlighting the effectiveness of the use of the CFD technique. These studies examined the influence of geometric parameters of the studied system and turbulence models on the behavior of fluids using CFDs, which are very close to experimental data.

Consequently, in other fields of fluid mechanics studied by Al-Obaidi *et al.,* [29], they analyze cases where cavitation occurs when using CFD fluid techniques for centrifugal pump systems, obtaining findings that correlate the impact of pressure, speed, and geometric characteristics of the system to detect the phenomenon of cavitation.

For the reasons mentioned earlier, this study examines the pressure drop behavior of a fluid flowing through a globe valve, applying computational fluid mechanics techniques such as CFD, and supported by statistical tools for experimentation like DOE. These contribute to an efficient interaction for resource savings.

2. Materials and Methods

It is essential to model the dynamic fluid behavior of a fluid passing through a globe valve system to evaluate the system's pressure drop under specific initial process conditions. To achieve this, the analysis is implemented through a Design of Experiments (DOE) that allows for correlating the tests applied at each stage of the process, simulated through CFD.

This is where an analysis of variance (ANOVA), applying response surfaces in statistical tools used for a DOE, enables an understanding of the relationship between independent and dependent variables, as explained in Chen *et al.,* [30].

In a design of experiments, experiments are planned and executed to explore how independent variables (factors) affecting a dependent variable (response) are systematically designed to obtain data that helps understand the influence of factors on the response variable and, simultaneously, find optimal conditions in a simulated design space.

In the current study, a response surface-based design of experiments will be applied for a complete 3^3 factorial design, with three factors and, for each of these, three levels of quantitative variation, along with a quantitative response variable. This results in 27 treatments for a good approximation of the result in a regression equation.

The factors chosen for the DOE are the surface roughness of the globe valve material (*Ra*), the valve opening percentage (*%A*), and the flow rate (*Q*). The response variable is the pressure drop (*ΔP*) expressed in *kPa*. The choice of the application range will be based on the operating values in this type of system. The Ra of materials used for hydraulic valves may vary depending on the type of metal and the manufacturing process and is generally measured in micrometers *µm*. For three common metals used in hydraulic valves, such as Stainless Steel, Brass, and Aluminum, Ra values typically fall within the following operating range: Stainless Steel from 0.2 to 0.6 *µm*, Brass from 0.8 to 1.6 *µm*, Aluminum from 1.6 to 3.2 *µm*. Therefore, the factors and treatment levels will be expressed as shown in Table 1.

Table 1
Factors implemented in the DOE

Factors	Minimum level: -1	Medium level: 0	Maximum level: +1
Ra [µm]	0.2	1.7	3.2
%A [%]	20	60	100
Q [m^3/s]	0.001	0.0155	0.03

To initiate the CFD simulation, we begin by obtaining the Computer-Aided Design (CAD) of the target globe valve, as illustrated in Figure 1 below.

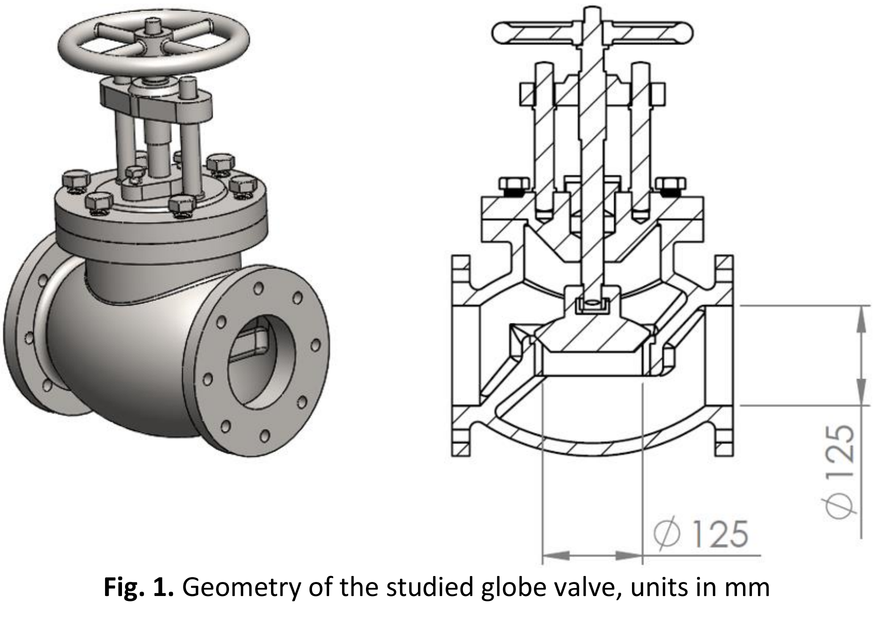

Fig. 1. Geometry of the studied globe valve, units in mm

The volumetric mesh is generated, and boundary conditions, including entry, exit, and wall effects, are assigned. Mesh inflation conditions were applied during volumetric meshing to model the boundary layer effects of the fluid as it passes through the valve passage. Figure 2 below shows the setup of the system.

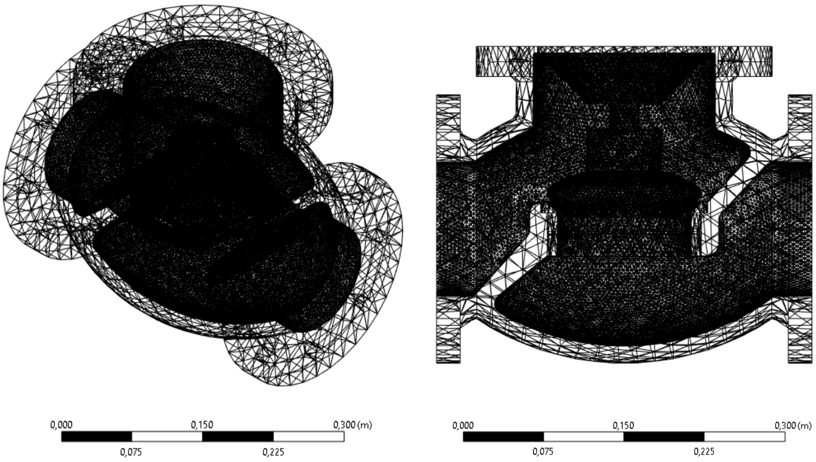

Fig. 2. Generated volumetric meshing of the system

When conducting numerical analysis and modeling of the behavior of fluid flowing through a globe valve, it is essential to activate the relevant equations for the effective implementation of the process. These equations include the energy equation, momentum equation, continuity equation, viscosity equation, among others.

The development of these equations is achieved through CFD analysis, utilizing tools for fluid flow behavior analysis. This involves working with a set of equations specifically tailored for the *k-ε* turbulence model, represented by Eq. (1) and Eq. (2), as indicated in Al-Obaidi *et al.,* [25], Alvarez *et al.,* [31], and Li *et al.,* [32].

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}\left(\rho k u_j\right)=\frac{\partial}{\partial x_j}\left[\Gamma_k \frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \tag{1}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_j}\left(\rho\varepsilon u_j\right)=\frac{\partial}{\partial x_j}\left[\Gamma_\varepsilon \frac{\partial \varepsilon}{\partial x_j}\right]-\rho C_{2\varepsilon}\frac{\varepsilon^2}{k}+C_{1\varepsilon}\frac{\varepsilon}{k}\left(G_k+C_{3\varepsilon}G_b\right)+S_\varepsilon \tag{2}$$

In these equations, ρ is the fluid density, u is the fluid velocity, G_k represents the generation of turbulent kinetic energy due to mean velocity gradients, while G_ε represents the generation of ε. Γ_k and $\Gamma\varepsilon$ denote the effective diffusivity of k and ε, respectively. G_b is the generation of turbulence kinetic energy due to buoyancy, Y_M represents the contribution of fluctuating expansion in compressible turbulence to the overall dissipation rate. S_k and S_ε are user-defined source terms, and the C_ε values are design constants.

To carry out the study in a controlled manner, an initial pressure value was set as an input border doncicion with a value of 300 kPa, characteristic value in aqueduct tubers as observed in a study by Carpintero *et al.,* [7], this for all treatments, since in CFD simulations for fluid flows, the pressure should be indicated as the input boundary, and the flow rate should be indicated as the output boundary for this type of study.

3. Results

Once the experiments supported by the CFD tool have been conducted, the treatment table is obtained based on the levels of the independent variables used, along with its corresponding result in the response variable, which, in this research, is the pressure drop. Table 2 below presents the developed Design of Experiments (DOE), and after this, the behaviors of these treatments in the CFD program will be illustrated.

Table 2
DOE results performed for globe valve system

Q	*%A*	*Ra*	*ΔP*
0.001	20	0.2	0.225
0.0155	20	0.2	55.108
0.03	20	0.2	213.88
0.001	60	0.2	0.067
0.0155	60	0.2	14.223
0.03	60	0.2	59.133
0.001	100	0.2	0.049
0.0155	100	0.2	11.63
0.03	100	0.2	43.813
0.001	20	1.7	0.249
0.0155	20	1.7	55.95
0.03	20	1.7	216.155
0.001	60	1.7	0.065
0.0155	60	1.7	13.809
0.03	60	1.7	58.255
0.001	100	1.7	0.049
0.0155	100	1.7	11.474
0.03	100	1.7	43.343
0.001	20	3.2	0.247
0.0155	20	3.2	56.731
0.03	20	3.2	220.079
0.001	60	3.2	0.065
0.0155	60	3.2	13.831
0.03	60	3.2	58.04
0.001	100	3.2	0.051
0.0155	100	3.2	11.553
0.03	100	3.2	41.678

The results are depicted through pressure profiles for each treatment configuration conducted in the 27 DOE experiments. Figure 3 below illustrates the pressure drop behavior as the fluid flows through the globe valve, considering a minimum valve opening level and varying the flow rate and surface roughness within their respective ranges of values.

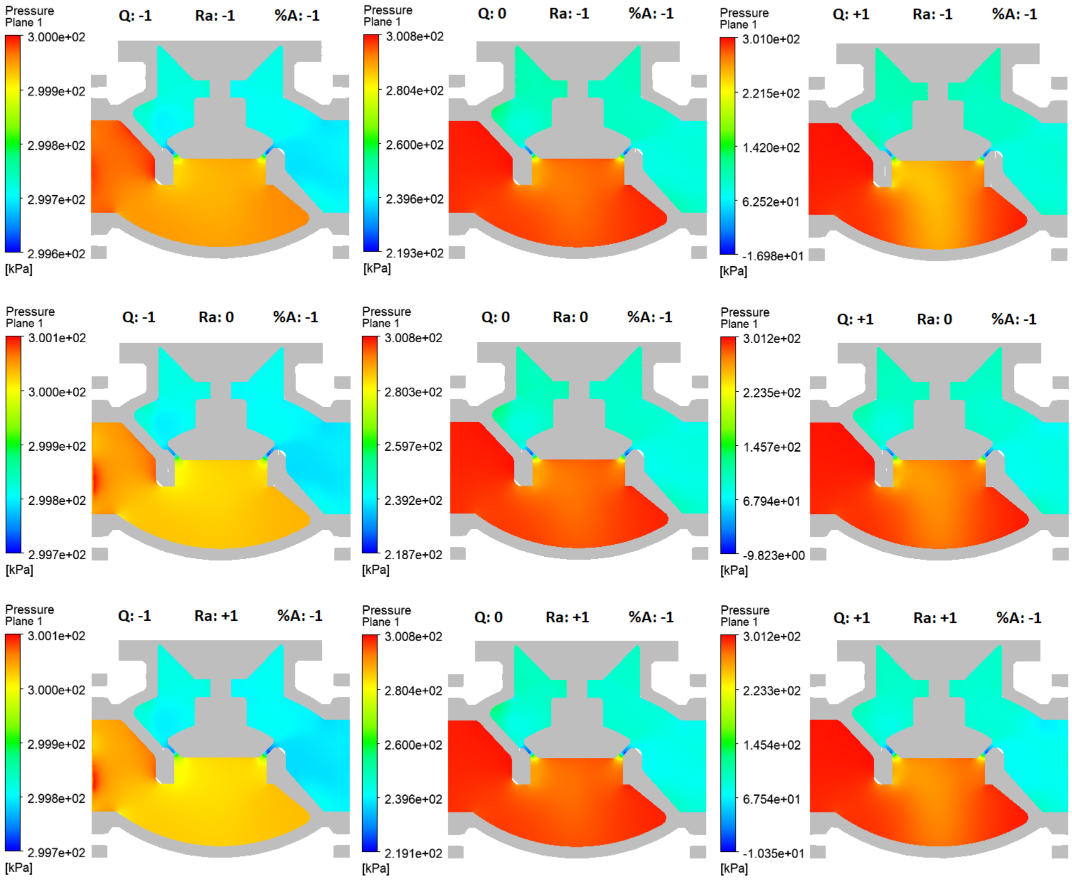

Fig. 3. Tests for pressure drop for a minimum valve opening level

Based on the visual results presented in the previous Figure 3, it is noteworthy that the variation in pressure drop becomes evident as the fluid flow rate increases. In contrast, the impact is less pronounced when maintaining a constant flow rate while varying the surface roughness.

Where it is highlighted that for the maximum flow value, the highest-pressure drops are obtained in the System by maintaining an opening percentage of 20% of the valve, for the minimum, medium, and high roughness values the drop values. pressure was 213.88, 216.155 and 220.079 *kPa* respectively. The Figure 4 below shows the maximum velocity value in the study, which was 21.46 *m/s* for a valve opening percentage of 20%.

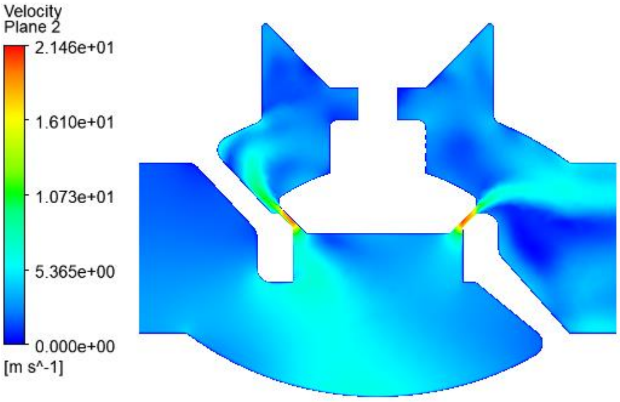

Fig. 4. Velocity profile for a minimum valve opening level, maximum flow rate and maximum roughness

In Figure 5 below, the schematic sequence is replicated for a medium valve opening level.

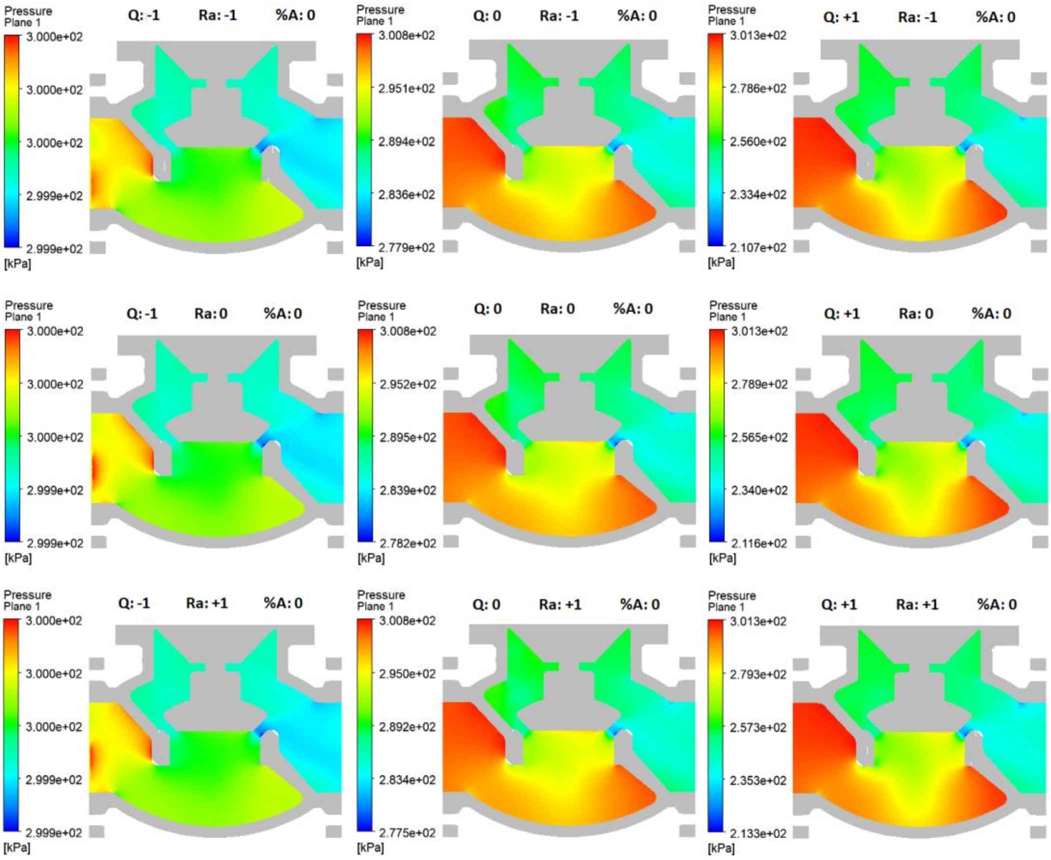

Fig. 5. Tests for pressure drop for a medium valve opening level

Similar to Figure 3, in the Figure 5 the pressure drop behavior is primarily influenced by increasing flow values and is less significantly affected by variations in the surface roughness of the valve.

It is highlighted that for the average flow value, the following pressure drop values are obtained from the system by maintaining an opening percentage of 60% of the valve, for the minimum, medium, and high roughness values. The pressure drop values were 59.133, 58.255 and 58.04 *kPa* respectively. Observing the results, we can see the little variation in the pressure drop value when changing the surface roughness values for a 60% valve opening. Figure 6 shows the velocity profile with a maximum value of 9.71 m/s for the valve opening percentage of 60%.

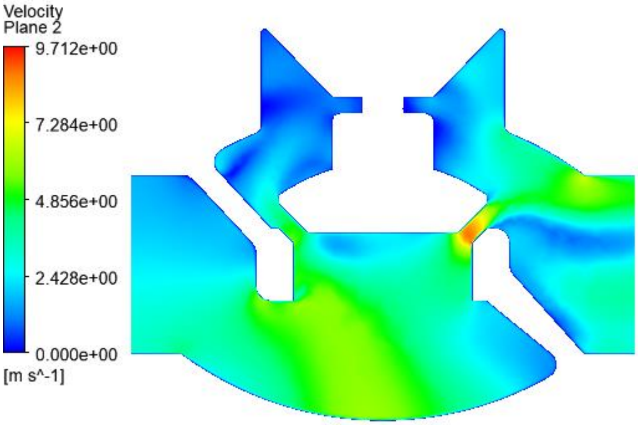

Fig. 6. Velocity profile for a medium valve opening level, maximum flow rate and maximum roughness

As the final diagram completes the 27 treatments, Figure 7 illustrates the pressure drop of the fluid when the globe valve is at its maximum opening percentage.

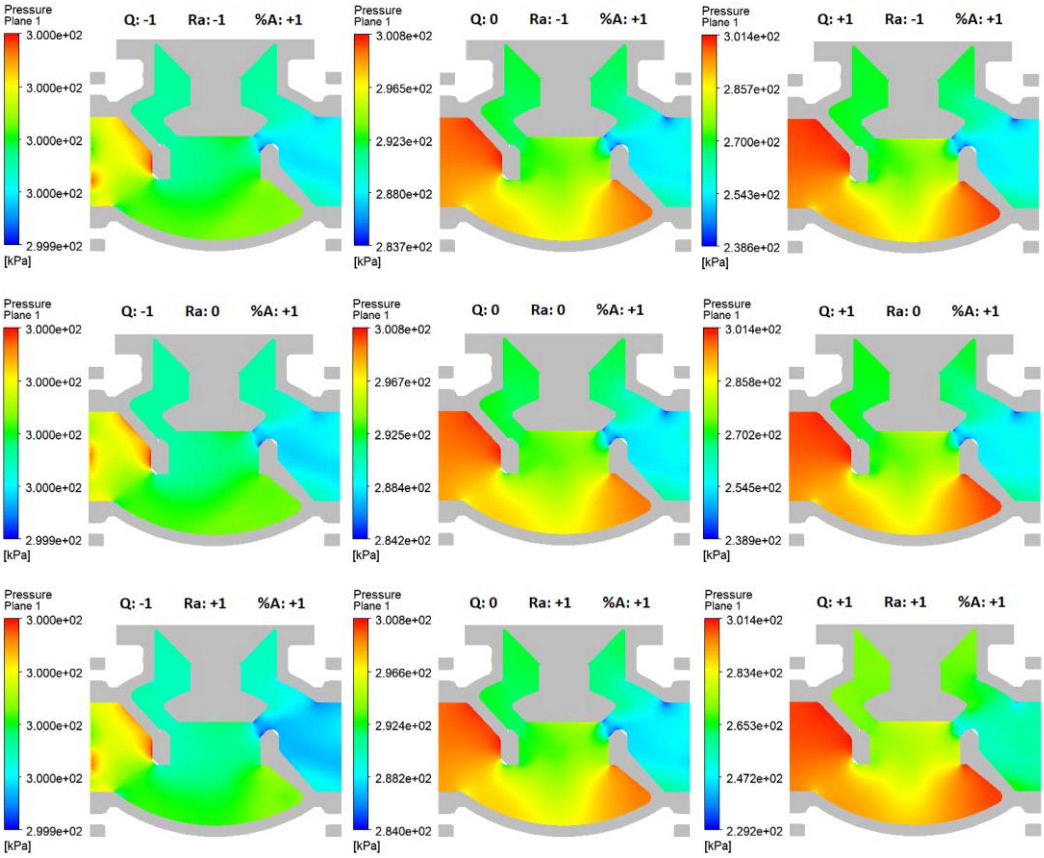

Fig. 7. Tests for pressure drop for a maximum valve opening level

For the maximum valve opening percentage, the pressure drop values are displayed as the minimum from the conducted experiments, varying both the flow rate and the roughness. The pressure drop values for the 100% opening percentage, in addition to being the lowest, present little variation between experiments, a case similar to that of the 60% opening percentage. Figure 8 below shows the case of 100% valve opening percentage with maximum flow and maximum roughness, obtaining values around 7.98 *m/s* of fluid velocity, highlighting that the greater the valve opening, the less increase in fluid velocity there will be through the system.

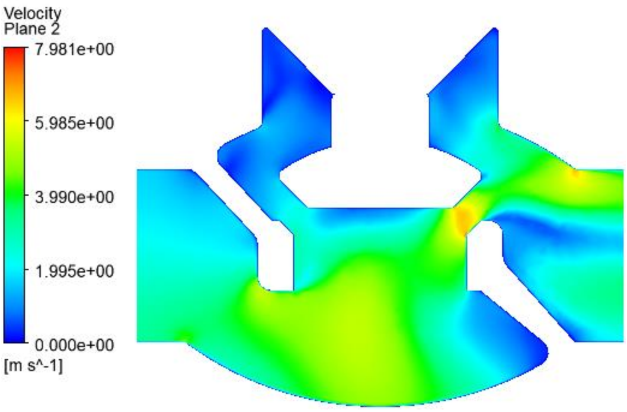

Fig. 8. Velocity profile for a maximum valve opening level, maximum flow rate and maximum roughness

Due to complications in the visual analysis of the results, the statistical tool of variance analysis (ANOVA) is employed. This tool enables the correlation of all results and helps identify which factors are genuinely significant for the investigated system. as shown in Table 3 below.

Table 3
ANOVA of the simulated system

Sources of variation	Sum of squares	DF	Mean sum of square	F-ratio	P-value
A: Q	36548.9	1	36548.9	85.74	0.0000
B: %A	16936.7	1	16936.7	39.73	0.0000
C: Ra	0.0828245	1	0.0828245	0.00	0.9890
AA	4035.66	1	4035.66	9.47	0.0068
AB	22594.7	1	22594.7	53.01	0.0000
AC	0.724717	1	0.724717	0.00	0.9676
BB	5546.74	1	5546.74	13.01	0.0022
BC	8.42358	1	8.42358	0.02	0.8899
CC	0.0538338	1	0.0538338	0.00	0.9912
Total error	7246.28	17	426.252		
Total (corr.)	113756,	26			

The ANOVA Table 3 partitions the variability of ΔP into separate components for each effect, then assesses the statistical significance of each effect by comparing its mean square against an estimate of the experimental error. In this case, five effects have a P-value less than 0.05, indicating their significant difference from zero at a 95.0% confidence level.

By excluding the effects of non-significant factors and their interactions, the R-Square statistic suggests that the adjusted model explains 93.621% of the variability in PD. The adjusted R-squared statistic, more suitable for comparing models with different numbers of independent variables, is 92.1022%. The standard error of the estimate reveals that the standard deviation of the residuals is 18.5888. The mean absolute error (MAE) of 14.5004 represents the average value of the residuals. The Durbin-Watson (DW) statistic tests the residuals for significant correlations based on the data order. Since the P-value is greater than 5.0%, there is no indication of serial autocorrelation in the residuals at the 5.0% significance level.

The Pareto diagram depicted in Figure 9 visually displays the significant effects of the conducted experiments concerning the response variable. It indicates that, to a greater extent, the maximum value of the flow rate is the effect most closely related to the pressure drop in the system. Simultaneously, the minimum value of the opening percentage significantly influences the pressure drop. However, it is noteworthy that the surface roughness does not significantly affect the pressure drop.

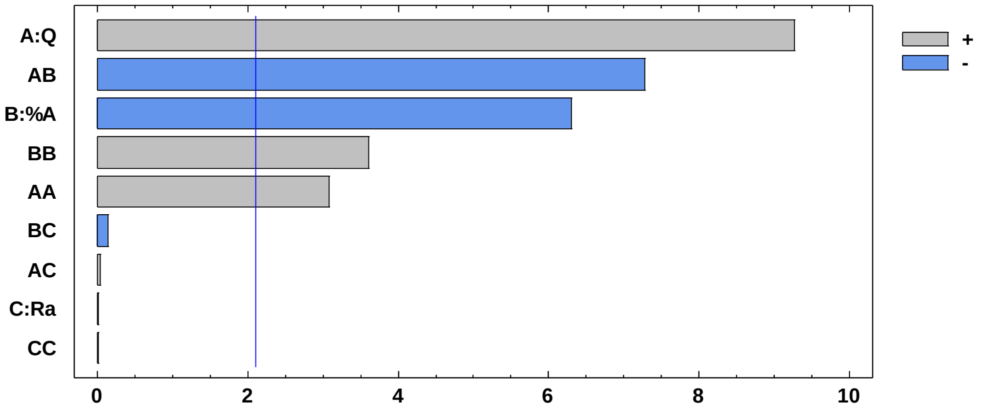

Fig. 9. Pareto diagram of the standardized effect for ΔP of the system

From the preceding Figure 10, we can emphasize the non-linearity of the factors Q and %A in the valve system concerning the pressure drop ΔP. This highlights the impact of these effects on the behavior of the response variable.

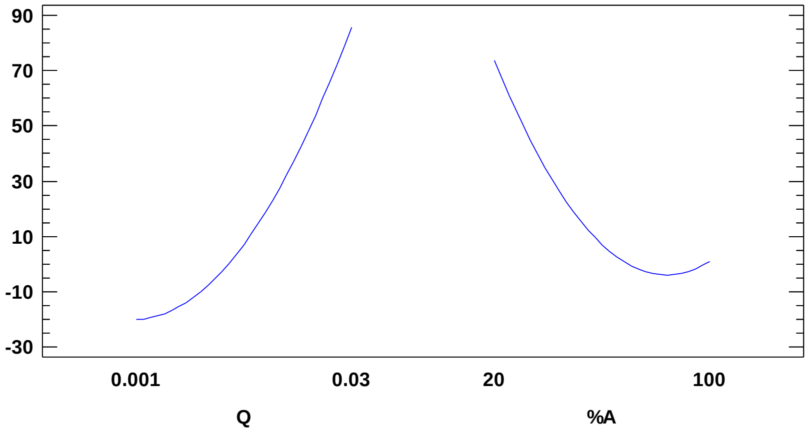

Fig. 10. Plot of main effects with respect to ΔP of the system

The interaction of effects illustrated in Figure 11 reveals that for the minimum level of Q, the variation in %A is not significant for the ΔP of the system. However, for the medium and maximum levels of Q, the variation in %A becomes indeed significant for the ΔP of the system. When relating to various studies such as those developed by Joshi *et al.,* [33] where they applied CFD computational tools to model a particulate fluid, they stand out as relevant factors that influence the pressure drop at high flow rates. Through the pipe, and the increase in roughness in sections of appreciable lengths. The short measurement path has led us to conclude that roughness was not of great importance for the system studied in the present study.

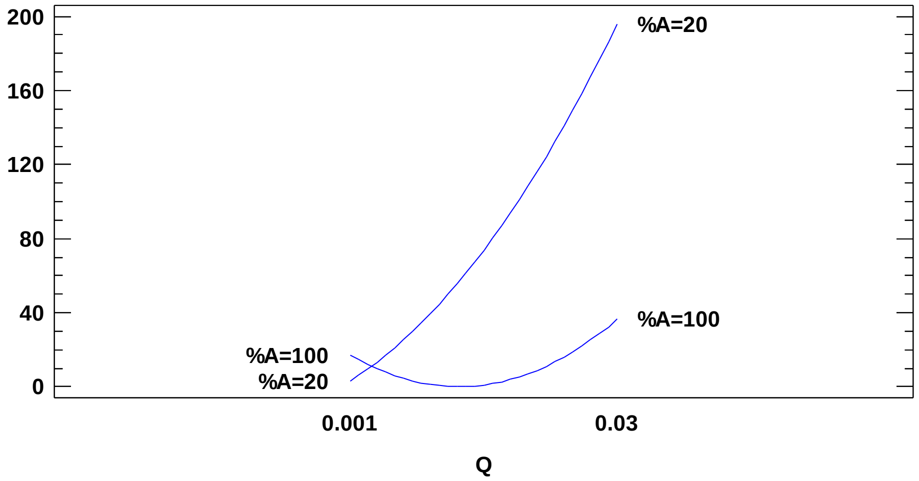

Fig. 11. Effects interaction plot for the ΔP of the system

In the preceding Figure 12, the behavior of the pressure drop is visually depicted through color profiles of the fluid passing through a globe valve. It emphasizes that the lower the opening percentage and the higher the flow, the greater the value of the pressure drop. Subsequently, the regression model for the evaluated system is obtained and represented as follows

$$\Delta P = 33.3123 + 4317.48\mathrm{Q} - 2.03044\%\mathrm{A} + 123352\mathrm{Q}^2 - 74.8144\mathrm{Q}\%\mathrm{A} + (1.24171\mathrm{E} - 10)\mathrm{QRa} + 0.0190031\%\mathrm{A}^2 \quad (3)$$

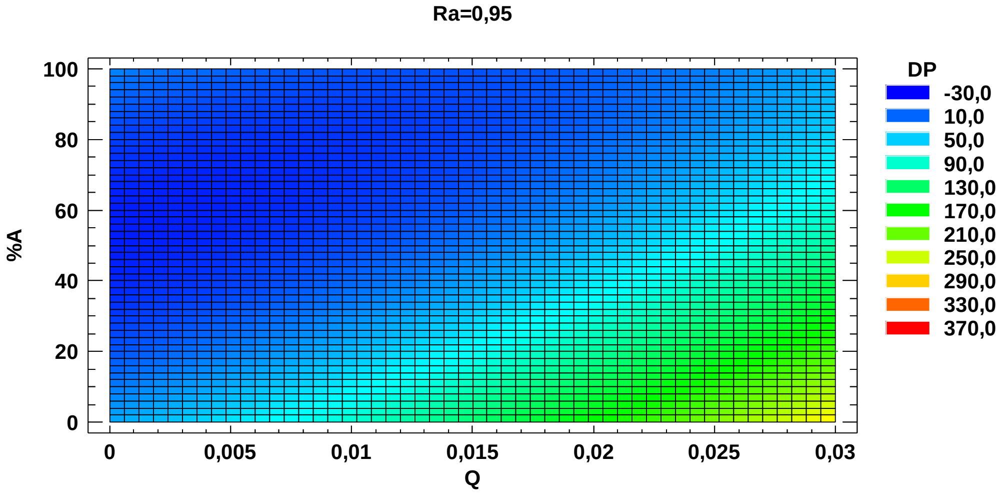

Fig. 12. Contour map of the pressure drop of a fluid across the valve

Reviewing the literature highlights the influence presented by using CFD tools for the study of mechanics of fluids in motion flows, and the importance of interacting with DOE statistic techniques which allow optimizing and obtaining these regression models by having greater reading accuracy in vital variables such as pressure drop, dimensions, physical characteristics of the tubers, concentrations of particulate matter, among other factors, as indicated by Ahmadi *et al.,* [34] in their study.

4. Conclusions

An analysis utilizing Design of Experiments (DOE) statistical techniques for a response surface model of the experimental factors, along with the application of Computational Fluid Dynamics (CFD), enabled the numerical prediction of the pressure drop behavior of a fluid as it passes through a globe valve.

A notable finding is that the roughness of materials commonly used for valve manufacturing does not significantly impact the pressure drop of a fluid flowing through these systems. However, the opening percentage of the globe valve proves to be significant, especially in scenarios where the flow is relatively high.

The methodology employed to derive equations that incorporate various influencing factors on pressure drop can be applied to a range of accessories within the field of fluid mechanics. This approach yields an academic product derived from computational experimentation.

It is recommended to pursue similar studies involving various types of accessories within fluid mechanics. This approach serves as a valuable method for teaching and learning in the realms of computational fluid mechanics and statistical analysis through the application of the Design of Experiments (DOE).

A pressure drop of 73% was presented when an opening percentage of 20% was used, maximum flow rate and the maximum roughness of the study developed. and a minimum pressure drop value of 0,049 kPa for minimum flow conditions, maximum opening, and surface roughness of the valve.

It is recommended to use the combination of CFD techniques and DOE strategies, to develop further research around the pressure drop, when conditions such as pipe diameter, shape and configuration of a valve are varied.

Acknowledgement
This research was not funded by any grant.

References

[1] Rezaey, Saber. "Numerical investigation of a globe control valve and estimating its loss coefficient at different opening states." *European Journal of Computational Mechanics* 29, no. 4-6 (2021): 549-576. https://doi.org/10.13052/ejcm1779-7179.294610

[2] Rani, Ahmad Majdi Abdul, Mohd Norhisyam Awang, M. Nasir A. Rani, Nur Athirah Ishar, Harvey M. Thompson, M. Hafizun Hassan, Bushra Mohamad Zakir, and Ahmad Tarmizi Hassan. "Simulation-enabled design and analysis of globe valve." In *Journal of Physics: Conference Series*, vol. 2051, no. 1, p. 012025. IOP Publishing, 2021. https://doi.org/10.1088/1742-6596/2051/1/012025

[3] Nguyen, Quang Khai, Kwang Hyo Jung, Gang Nam Lee, Sung Bu Suh, and Peter To. "Experimental study on pressure distribution and flow coefficient of globe valve." Processes 8, no. 7 (2020): 875. https://doi.org/10.3390/pr8070875

[4] Lira, Jéssica O. B., Humberto G. Riella, Natan Padoin, and Cíntia Soares. "CFD+ DoE optimization of a flat plate photocatalytic reactor applied to NOx abatement." *Chemical Engineering and Processing-Process Intensification* 154 (2020): 107998. https://doi.org/10.1016/j.cep.2020.107998

[5] Izadi, Ahad, Elham Kashani, and Ali Mohebbi. "Combining 10 meta-heuristic algorithms, CFD, DOE, MGGP and PROMETHEE II for optimizing Stairmand cyclone separator." *Powder Technology* 382 (2021): 70-84. https://doi.org/10.1016/j.powtec.2020.12.056

[6] Tambolkar, Pooja, Aishwarya Ponkshe, Vrushali Mulay, and Anand Bewoor. "Use of taguchi DOE for CFD simulation to maximize the reusability of working fluids of centrifugal filter." *Procedia Manufacturing* 46 (2020): 608-614. https://doi.org/10.1016/j.promfg.2020.03.087

[7] Carpintero, Javier, Fausto A. Canales, Jonathan Fábregas, and José Ávila. "Factors and interactions that influence the pressure drop across an air volume reducing device on low-pressure water distribution networks." *Iranian Journal of Science and Technology, Transactions of Civil Engineering* 46, no. 2 (2022): 1433-1443. https://doi.org/10.1007/s40996-021-00682-z

[8] Carpintero, Javier, Jennifer Villa-Dominguez, María José Tavera-Quiroz, Humberto Carlos Tavera-Quiroz, Bartosz Kaźmierczak, Jonathan Fábregas-Villegas, and Fausto A. Canales. "Application of a 2k-p fractional experimental design in coagulation-flocculation processes in the treatment of wastewater from a slaughterhouse." *Sustainability* 14, no. 16 (2022): 10402. https://doi.org/10.3390/su141610402

[9] Jasim, Rasha Abdulrazzak, Wajdi Qasim Hussen, Mahir Faris Abdullah, and Rozli Zulkifli. "Numerical Simulation of Characterization of Hydraulic Jump Over an Obstacle in an Open Channel Flow." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 106, no. 1 (2023): 1-15. https://doi.org/10.37934/arfmts.106.1.115

[10] Amina, Mostefaoui, Saim Rachid, and Abboudi Saïd. "Numerical Analysis of The Thermo-Convective Behaviour of an Al_2O_3-H_2O Nanofluid Flow Inside a Channel with Trapezoidal Corrugations." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 89, no. 2 (2022): 114-127. https://doi.org/10.37934/arfmts.89.2.114127

[11] Benkara-Mostefa, Karima Heguehoug, and Rahima Benchabi-Lanani. "Heat Transfer and Entropy Generation of Turbulent Flow in Corrugated Channel using Nanofluid." Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 109, no. 2 (2023): 136-150. https://doi.org/10.37934/arfmts.109.2.136150

[12] Abugnah, Elhadi Kh, Wan Saiful-Islam Wan Salim, and Abdulhafid M. Elfaghi. "Impact of Geometrical Parameters on Heat Transfer Characteristics of SiO_2-Water Nanofluid Flow through Rectangular Corrugated Channels." Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 104, no. 2 (2023): 86-102. https://doi.org/10.37934/arfmts.104.2.86102

[13] Elfaghi, Abdulhafid M. A., Alhadi A. Abosbaia, Munir F. A. Alkbir, and Abdoulhdi A. B. Omran. "CFD Simulation of Forced Convection Heat Transfer Enhancement in Pipe Using Al_2O_3/Water Nanofluid." Journal of Advanced Research in Numerical Heat Transfer 8, no. 1 (2022): 44-49.

[14] Idris, Muhammad Syafiq, Irnie Azlin Zakaria, and Wan Azmi Wan Hamzah. "Heat transfer and pressure drop of water based hybrid Al_2O_3: SiO_2 nanofluids in cooling plate of PEMFC." *Journal of Advanced Research in Numerical Heat Transfer* 4, no. 1 (2021): 1-13.

[15] Sidik, Nor Azwadi Che, Saidu Bello Abubakar, and Siti Nurul Akmal Yusof. "Measurement of Fluid Flow and Heat Transfer Performance in Rectangular Microchannel using Pure Water and Fe_3O_4-H_2O Nanofluid." *Journal of Advanced Research in Applied Mechanics* 68, no. 1 (2020): 9-21. https://doi.org/10.37934/aram.68.1.921

[16] Sidik, Nor Azwadi Che, Abdolbaqi Mohammed Khdher, Siti Nurul Akmal Yusof, and M'hamed Beriache. "Heat Transfer Enhancement in Straight Channel with Nanofluid in Fully Developed Turbulent Flow." *Journal of Advanced Research in Applied Mechanics* 63, no. 1 (2019): 1-15.

[17] Sukamta, Sukamta, Rama Putrantyo Anwar, Sudarja Sudarja, Sri Sundari, and Mochamad Syamsiro. "Computational Fluid Dynamics of Two-Phase Flow Pressure Gradient of Air-Sodium Chloride and Glucose in Horizontal Capillary Channels." *CFD Letters* 14, no. 1 (2022): 52-65. https://doi.org/10.37934/cfdl.14.1.5265

[18] Enam, Samin, Muhammad Noor Afiq Witri Muhammad Yazid, and Nor Azwadi Che Sidik. "Numerical Investigation on The Effect of Grille Blockage Ratio on Air Flow Characteristics of Air Vents." *CFD Letters* 14, no. 10 (2022): 32-41. https://doi.org/10.37934/cfdl.14.10.3241

[19] Khan, Umair, William Pao, Nabihah Sallih, and Farruk Hassan. "Identification of horizontal gas-liquid two-phase flow regime using deep learning." *CFD Letters* 14, no. 10 (2022): 68-78. https://doi.org/10.37934/cfdl.14.10.6878

[20] Fábregas, Jonathan, Henry Santamaria, Edgardo Buelvas, Saul Perez, Carlos Díaz, Javier Andrés Carpintero Durango, Ricardo Mendoza, and Jennifer Villa. "Computational fluid dynamics modeling of microchannels cooling for electronic microdevices." *IIUM Engineering Journal* 23, no. 1 (2022): 384-396. https://doi.org/10.31436/iiumej.v23i1.2113

[21] Bu, Qingyun, Aliyu Aliyu, and Rakesh Mishra. "A new one-dimensional control valve model for the local flow coefficient based on local analysis of CFD data." *International Journal of COMADEM* 25, no. 1 (2022): 5-8.

[22] Garg, Kapil, Suvir Singh, Mahadev Rokade, and Shubham Singh. "Experimental and computational fluid dynamic (CFD) simulation of leak shapes and sizes for gas pipeline." *Journal of Loss Prevention in the Process Industries* 84 (2023): 105112. https://doi.org/10.1016/j.jlp.2023.105112

[23] Reich, Alton. "Use of CFD to Improve Control Valve Effectiveness." In *Pressure Vessels and Piping Conference*, vol. 58943, p. V003T03A059. American Society of Mechanical Engineers, 2019.

[24] Serani, Andrea, Riccardo Pellegrini, Jeroen Wackers, C-E. Jeanson, Patrick Queutey, Michel Visonneau, and Matteo Diez. "Adaptive multi-fidelity sampling for CFD-based optimisation via radial basis function metamodels." *International Journal of Computational Fluid Dynamics* 33, no. 6-7 (2019): 237-255. https://doi.org/10.1080/10618562.2019.1683164

[25] Al-Obaidi, Ahmed Ramadhan, and Jassim Alhamid. "Investigation of the main flow characteristics mechanism and flow dynamics within an axial flow pump based on different transient load conditions." *Iranian Journal of Science and Technology, Transactions of Mechanical Engineering* 47, no. 4 (2023): 1397-1415. https://doi.org/10.1007/s40997-022-00586-x

[26] Al-Obaidi, Ahmed, Hussam Khalaf, and Jassim Alhamid. "Investigation of the influence of varying operation configurations on flow behaviors characteristics and hydraulic axial-flow pump performance." In *Proceedings of the 4th International Conference on Science Education in The Industrial Revolution 4.0, ICONSEIR 2022, November 24th, 2022, Medan, Indonesia*. 2023. https://doi.org/10.4108/eai.24-11-2022.2332719

[27] Al-Obaidi, Ahmed, Hussam Khalaf, and Jassim Alhamid. "Investigation on the characteristics of internal flow within three-dimensional axial pump based on different flow conditions." In *Proceedings of the 4th International Conference on Science Education in The Industrial Revolution 4.0, ICONSEIR 2022, November 24th, 2022, Medan, Indonesia*. 2023. https://doi.org/10.4108/eai.24-11-2022.2332720

[28] Al-Obaidi, A. R. "Effect of different guide vane configurations on flow field investigation and performances of an axial pump based on CFD analysis and vibration investigation." *Experimental Techniques* 48, no. 1 (2024): 69-88. https://doi.org/10.1007/s40799-023-00641-5

[29] Al-Obaidi, Ahmed Ramadhan, and Ali Qubian. "Effect of outlet impeller diameter on performance prediction of centrifugal pump under single-phase and cavitation flow conditions." *International Journal of Nonlinear Sciences and Numerical Simulation* 23, no. 7-8 (2022): 1203-1229. https://doi.org/10.1515/ijnsns-2020-0119

[30] Chen, Wei-Hsin, Manuel Carrera Uribe, Eilhann E. Kwon, Kun-Yi Andrew Lin, Young-Kwon Park, Lu Ding, and Lip Huat Saw. "A comprehensive review of thermoelectric generation optimization by statistical approach: Taguchi method, analysis of variance (ANOVA), and response surface methodology (RSM)." *Renewable and Sustainable Energy Reviews* 169 (2022): 112917. https://doi.org/10.1016/j.rser.2022.112917

[31] Alvarez, Jorge, Jonathan Fabregas Villegas, Mauricio Márquez, and Javier Carpintero. "Energy Evaluation of Synthesis Gas in a Turbocharger System Employing CFD Tools." *CFD Letters* 16, no. 6 (2024): 109-119. https://doi.org/10.37934/cfdl.16.6.109119

[32] Li, Ning, Yongqian Liu, Li Li, Siyu Chang, Shuang Han, Hao Zhao, and Hang Meng. "Numerical simulation of wind turbine wake based on extended k-epsilon turbulence model coupling with actuator disc considering nacelle and tower." *IET Renewable Power Generation* 14, no. 18 (2020): 3834-3842. https://doi.org/10.1049/iet-rpg.2020.0416

[33] Joshi, Tanuj, Om Parkash, and Gopal Krishan. "Numerical investigation of slurry pressure drop at different pipe roughness in a straight pipe using CFD." *Arabian Journal for Science and Engineering* 47, no. 12 (2022): 15391-15414. https://doi.org/10.1007/s13369-022-06583-1

[34] Ahmadi, Ali Akbar, Masoud Arabbeiki, Hafiz Muhammad Ali, Marjan Goodarzi, and Mohammad Reza Safaei. "Configuration and optimization of a minichannel using water-alumina nanofluid by non-dominated sorting genetic algorithm and response surface method." *Nanomaterials* 10, no. 5 (2020): 901. https://doi.org/10.3390/nano10050901