INTERNATIONAL JOURNAL OF AUTOMOTIVE SCIENCE AND TECHNOLOGY

2024, VOL. 8, NO:3, 381-386 www.ijastech.org e-ISSN: 2587-0963

Numerical Investigation of Flow Characteristics in Internally Flowing Pipes with Varying Roughness and Helical Angles of Vortex Generators

Fuat Tan[1*] and Alp Eren Dede[1]

[1]*Department of Mechanical Engineering, Faculty of Engineering, Balikesir University, Balıkesir, 10145, Turkey*

Research Article

Abstract

In this study, the turbulent behavior inside a pipe containing an aluminum helical vortex generator plate was investigated using computational fluid dynamics (CFD) methods, considering different plate roughness and helical angle values. Accordingly, numerical analyses of velocity and pressure distributions for a liquid flow with a constant inlet velocity of 0.5 m/s and four different helical angles were performed and compared in a computer software. To observe the effects of the roughness values of the metal plate on flow behavior, analyses were conducted for three different roughness values for each helical angle and the results were compared and interpreted. As a result of the study, the combination of a roughness value of 0.04 and a helical angle of 0° led to the highest pressure and velocity values, while the same roughness value combined with a 270° helical angle resulted in the lowest pressure and velocity values. Detailed analysis showed that helical angles (90° and 180°) presented moderate pressure and velocity values, indicating a non-linear relationship between helical angle and flow characteristics. The results demonstrate that optimizing the helical angle and roughness is crucial for enhancing the efficiency of vortex generators. This way, a design will be developed to prevent performance degradation in industrial applications that require flow efficiency and pressure management.

Keywords: CFD; Helical; Pressure; Roughness; Vortex generator

History
Received 02.07.2024
Revised 03.08.2024
Accepted 15.08.2024

Contact
* Corresponding author
Fuat TAN
fuattan@balikesir.edu.tr
Address: Mechanical Engineering Department, Faculty of Engineering, Balikesir University, Balikesir, Turkey
Tel:+902666121194

To cite this paper: Tan, F., Dede, A.E., Numerical Investigation of Flow Characteristics in Internally Flowing Pipes with Varying Roughness and Helical Angles of Vortex Generators. International Journal of Automotive Science and Technology. 2024; 8 (3): 381-386. https://doi.org/10.30939/ijastech..1509396

1. Introduction

Fluid flow inside pipes is commonly utilized for heating and distribution purposes, as well as in heat exchangers, turbines, autumotive industry and hydraulic transport systems. In these systems, the fluid entering the flow region at a specific velocity and pressure undergoes pressure drop and head loss due to pipe design and friction factor. The tangential velocity of the fluid, which transitions to a vortex state depending on the geometric design, influences the average velocity [1]. Heat exchanger components are used in the automotive industry to improve vehicle efficiency and reduce emissions. By increasing heat transfer, energy loss is minimized. This technology helps manage the thermal load of engines, batteries and other critical components.

Turbulence generators which are from heat exchangers disrupt the boundary layer, causing it to thin out, disperse and re-form. They increase the surface area available for heat transfer and amplify the turbulence level within the flow, thereby extending the flow path. As a result, turbulence is induced within the flow volume[2]. The performance of a turbulator varies depending on the geometry of the turbulator, the hydraulic diameter of the pipe with the turbulator, the thermophysical properties of the fluid and the mass flow rate of the fluid [3].

The number of vortices formed in the flow also defines the relationship between circumferential and axial momentums [4]. Initial studies on pipe flow focused on pressure drop and flow velocity, which were described mathematically by G. Hagen and J. Poiseuille [5]. The friction factor, later added to the studies by Bhatti et al., highlighted the critical behaviors induced by surface roughness in fluid flow [6]. Fang et al. examined the effects of the friction factor in adiabatic supercritical turbulent pipe flow under critical pressure [7].

The transfer of heat and mass is influenced by the vortex flow structure. In an experimental study by Yılmaz et al., the behaviors of vortex flows generated by attaching different guide vanes were investigated. The results indicated that rotational flow exhibited higher performance compared to axial flow at high vane angles and low inlet velocities [8].

In a combined experimental and numerical study by Bilen et al., the relationships between the friction factor (f), Nusselt number (Nu) and flow Reynolds number for vortex generators with different helical angles were examined [9]. Another study by Yeşildal proposed a mathematical model for the relationship between the heat transfer coefficient, flow characteristics and geometry. Yeşildal used the Response Surface Methodology to model the effects of Reynolds number, helical angle and pitch values on the flow for an axial vortex generator placed at the pipe inlet [10].

Banerjee et al. experimentally investigated the rotational flow inside a coaxial cylinder pipe and compared the results numerically using the RSM experimental design method. The rotational flow behavior showed results closer to the turbulent kinetic energy values for vortex density [11]. A similar study was conducted by Rocha et al. [12] using a finned vortex generator. They numerically investigated the rotational flow generated for regions with Reynolds numbers less than 2000, focusing on pressure drop, velocity components and vortex density using the CFD method.

Saqr and Wahid examined the effects of turbulence, inlet vortex density and vortex heat transfer on local entropy generation in compressible, non-isothermal pipe flow [13]. According to an experimental study, Kurtbaş et al. investigated the effect of conical injector-type vortex generators in a tube with uniform heat flux under turbulent flow regime [14]. The study conducted by Baysal et al. demonstrated that friction loss in tubes with turbulators is higher compared to plain tubes [15]. In the study conducted by Solmaz et al., turbulators were used in a counterflow concentric tube heat exchanger. The different stages of circular turbulators, placed inside the inner tube, were numerically analyzed using the applicable K-ε turbulence model. As a result, it was found that the highest heat transfer enhancement of 233.08% compared to the plain tube condition was achieved with a turbulator spacing of 25 mm [16]. In their study, Karakaya and Durmuş designed conical spring turbulators and analyzed their impact on heat transfer and pressure drop in a tube for Reynolds numbers between 10,000 and 34,000. They also examined how three different turbulator angles affected heat transfer and exergy loss [17]. There are many studies observing the effects of turbulence models on flow behavior in CFD analyses [19-20].

In light of the findings and recommendations in the literature, this study investigates the turbulent behavior inside a pipe containing an aluminum helical vortex converter plate using computational fluid dynamics (CFD). The aluminum plate adapted inside the pipe was designed with four different helical angles (0°, 90°, 180°, 270°). A numerical study was conducted and interpreted to reveal the effects of different roughness values (0.04, 0.066, 0.092) on pressure and velocity in turbulent flow.

2. Materials and Methods

In this study, an aluminum plate with helical angles of 0°, 90°, 180° and 270° was used inside a horizontal straight pipe with a diameter of 15 mm and a length of 100 mm (Figure 2). The angle of 0° indicates that the plate is flat. Four different visual representations of the model based on these helical angles are provided in Figure 1.

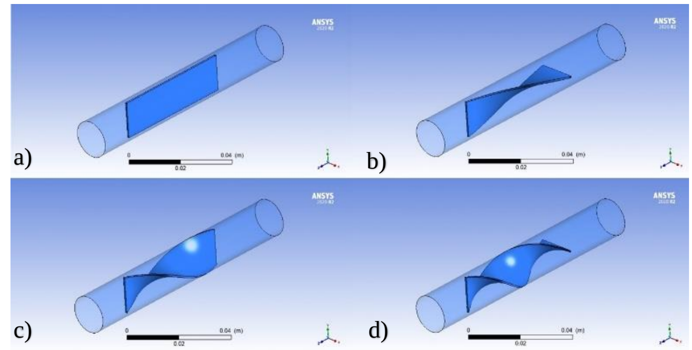

Fig. 1. Geometries with helical angles (a=0°, b=90°, c=180°, d=270°)

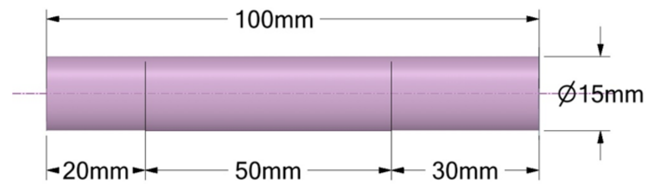

Fig. 2. Dimensions of the model

For each model, the fluid inlet velocity was assumed to be constant at 0.5 m/s and the fluid used was water at a temperature of 20°C. To examine the fully developed turbulent behavior, flow analysis was conducted over a length of 30 mm following the aluminum plate. This ensured the establishment of a stable regime with fully developed turbulent behavior. All flow volumes in the model were considered as liquid flow volumes, excluding the metal. The ANSYS-Fluent software package was used for the computational fluid dynamics (CFD) analysis. The assumptions made in all analyses and physical equations are as follows [18];

- Incompressible water in the liquid phase was used as the working fluid.
- The physical properties of water were assumed to be a density of 998.2 kg/m³ and a viscosity of 0.001003 kg/m·s.
- The liquid flow inside the pipe is steady and stable.
- The effects of gravity have not been considered.

The following conservation equations were used to examine the flow behavior in the analyses conducted;

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho u}{\partial x}+\frac{\partial \rho v}{\partial y}+\frac{\partial \rho w}{\partial z} = 0 \qquad (1)$$

The momentum conservation equation:

$$\frac{\partial \rho}{\partial t}+\vec{\nabla}.\left(\rho\vec{V}\right)=\frac{\partial \rho}{\partial t}+\vec{V}.\vec{\nabla}\rho+\rho\nabla.\vec{V}=0 \qquad (2)$$

The energy conservation equation;

$$P c_v \frac{dt}{dt} = k\nabla^2 T + \emptyset \tag{3}$$

The turbulence kinetic energy equation;

$$\frac{\partial}{\partial x_i}(pku_i) = \frac{\partial}{\partial x_j} + \left[a_k \mu_{eff} \frac{\partial k}{\partial xj}\right] + G_k - \rho\varepsilon \tag{4}$$

The standard K-ε turbulence model with Standard Wall Functions conditions was selected for this study due to its well-established reliability and effectiveness in simulating turbulent flow in internal pipe flows, as extensively documented in the literature [8-9]. For four different helical angles (0°, 90°, 180°, 270°), a tetrahedron mesh type with an element size of 0.5 mm was used, resulting in mesh element counts of 330609, 349723, 351403 and 348513, respectively. The mesh structure of the model is shown in Figure 3.

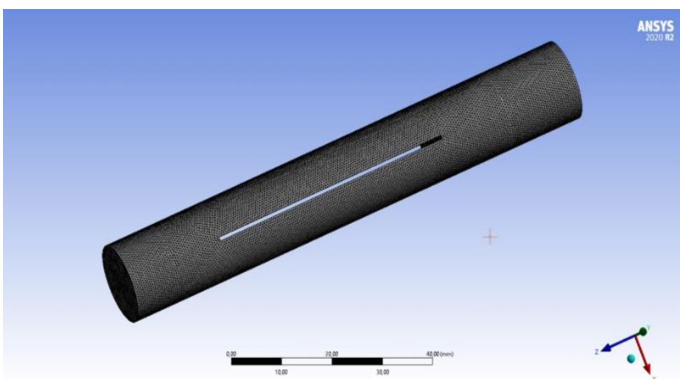

Fig. 3. Mesh visualization for 0° helical geometry

3.Results

Below are the pressure and velocity results within a vortex generator containing plates with four different helical angles, corresponding to a constant inlet speed. Numerical analysis simulation images are also provided in the same order of the helical angles.

3.1 Pressure Behavior

Upon analyzing the results, the maximum pressure values are observed at the point where the roughness reaches its minimum, specifically at a value of 0.04 and where the helix angle is 0° (Figure 5). Another noteworthy point is that for a helix angle of 90°, the maximum pressure value occurs at a medium roughness value of 0.066. For a helix angle of 180°, the minimum pressure value also occurs at the medium roughness value of 0.066 and this value is equal to the pressure value at the same roughness level for a helix angle of 0°. While no significant pressure variation is observed for the roughness value of 0.066 except for the 90° helix angle, a sudden increase in pressure is noted for the 180° helix angle (Figure 6).

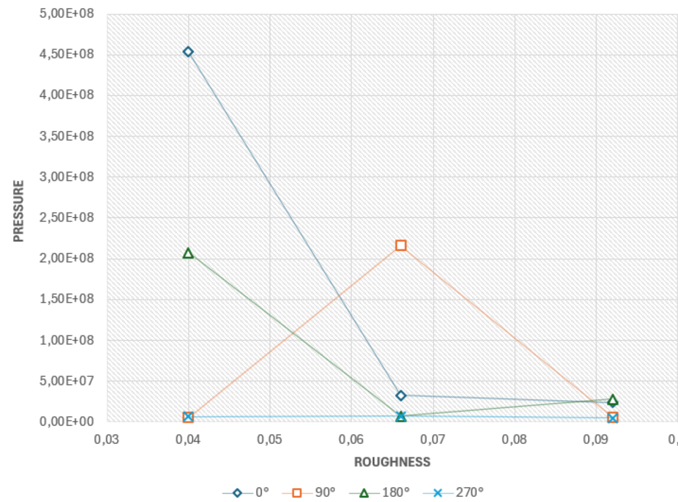

Fig. 4. Pressure vs. roughness and angle

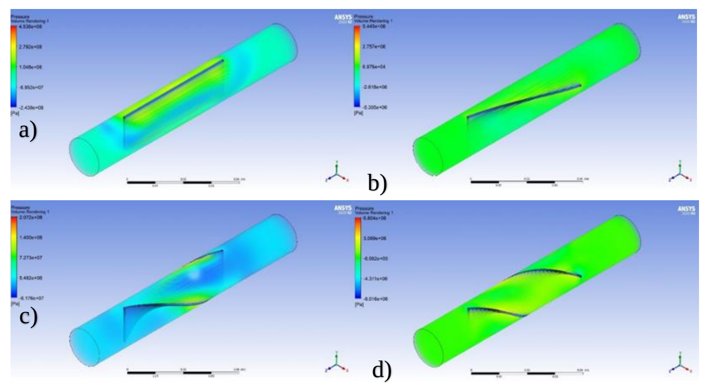

Fig. 5. Pressure vs roughness of 0.04 (a=0°, b=90°, c=180°, d=270°)

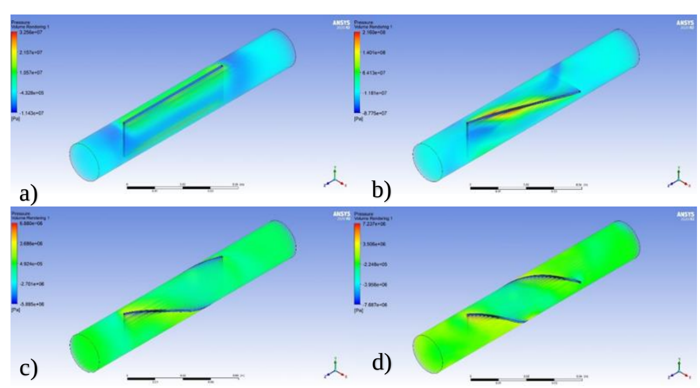

Fig. 6. Pressure vs roughness of 0.066 (a=0°, b=90°, c=180°, d=270°)

In general, across all results in Figure 4, an inverse relationship was observed for a helix angle of 0°, where an increase in the roughness value corresponded to a decrease in the pressure value. Additionally, at roughness values of 0.04 and 0.066, a sudden decrease in pressure was noted when the helix angle increased from 90° to 180°.

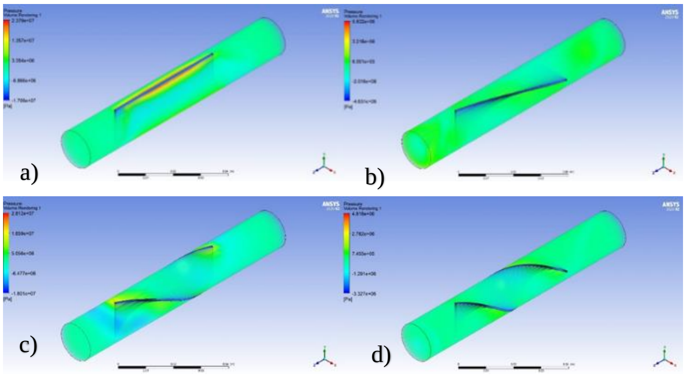

Fig. 7. Pressure vs roughness of 0.092 (a=0°, b=90°, c=180°, d=270°)

This decrease was more pronounced at the roughness value of 0.04 and the roughness value of 0.092 did not have a significant effect on the pressure (Figure 7). For a helix angle of 270°, roughness did not have a notable impact on pressure. However, a sudden drop in pressure was observed with the helix angle change from 180° to 270° (Figure 8).

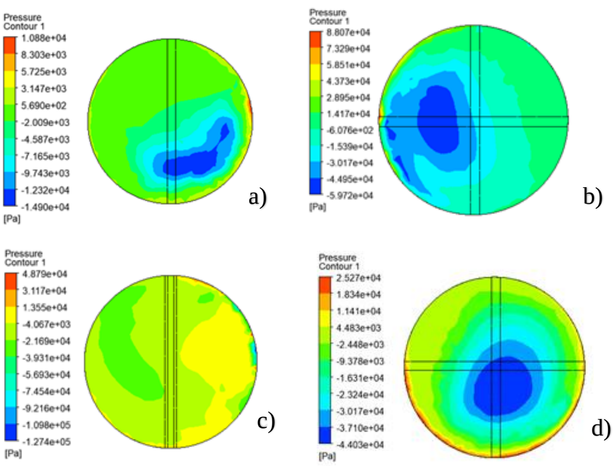

Fig. 8. Pressure vs roughness of 0.092 on outlet (a=0°, b=90°, c=180°, d=270°)

Velocity Behaviors

Upon examining Figure 9, it is observed that the maximum velocity point occurs at the minimum roughness value for a helix angle of 0°. The minimum point, on the other hand, is at a helix angle of 90°. When the roughness increases from 0.04 to 0.066, there is an increase in velocity for helix angles of 90° and 270°, while for 0° and 180°, a decreasing trend is observed.

Another important point is that when the roughness is increased to its maximum value, the velocity decreases for helix angles of 0°, 90° and 270°, whereas for 180°, an increasing behavior is observed. The maximum velocity at a helix angle of 270° is at a medium roughness value of 0.066 (Figure 11). This behavior is expected and consistent, as seen from Eq. (4).

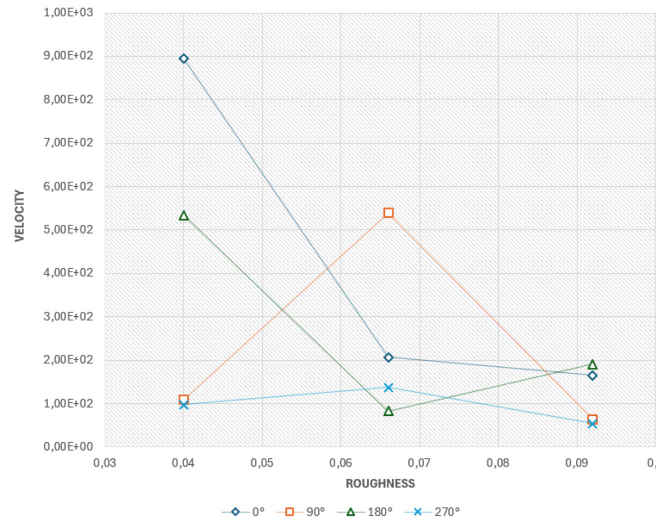

Fig. 9. Velocity vs. roughness and angle

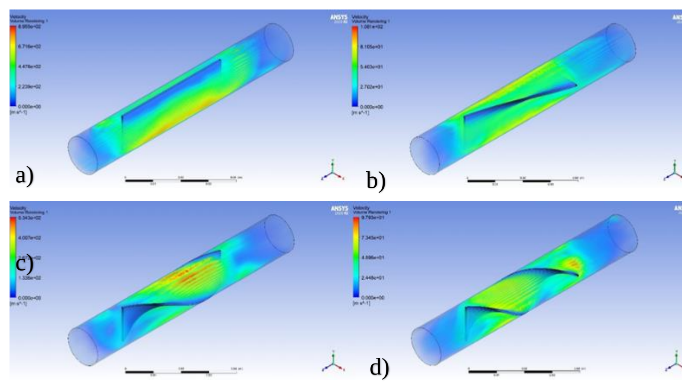

Fig. 10. Velocity vs roughness of 0.04 (a=0°, b=90°, c=180°, d=270°)

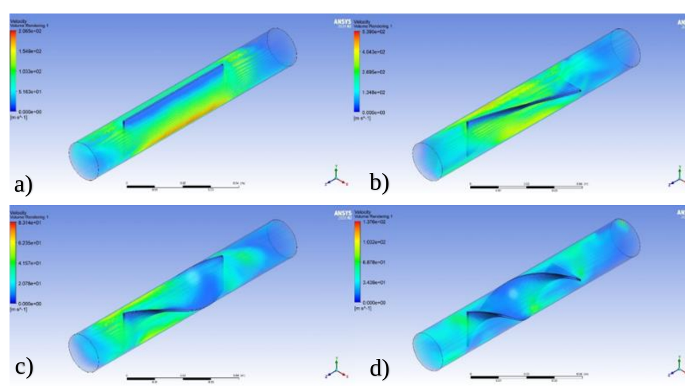

Fig. 11. Velocity vs roughness of 0.066 (a=0°, b=90°, c=180°, d=270°)

The highest speed values were obtained at a 0° helix angle and as the helix angle increased, the speed values decreased. The flow path lengthens, the flow resistance increases and the energy is distributed over a wider area. As the roughness increased, a decrease in speed values occurred. This indicates that the flow encounters the least resistance on the plate, resulting in the highest speed. Velocity vb. roughness figures depending on different helix angles are given in Figures 10, Figure 11 and Figure 12 and Figure 13.

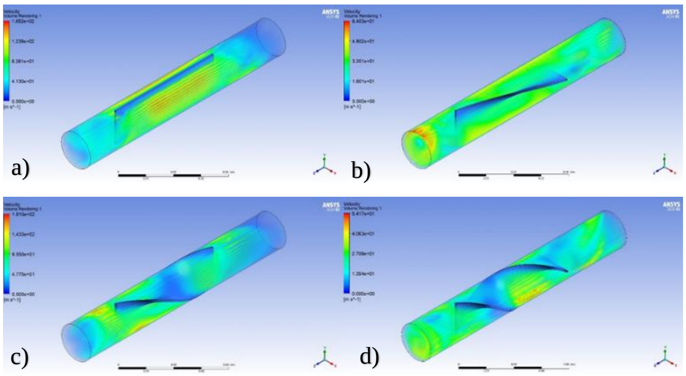

Fig. 12. Velocity vs roughness of 0.092(a=0°, b=90°, c=180°, d=270°)

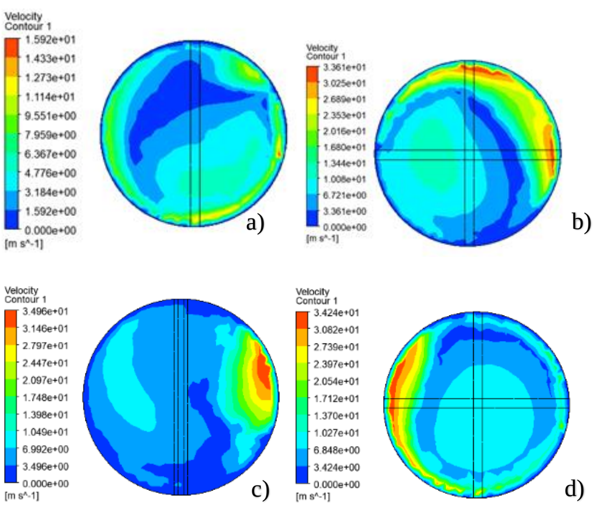

Fig. 13. Velocity vs roughness of 0.092 on outlet (a=0°, b=90°, c=180°, d=270°)

4. Conclusions

The study shows that changes in the roughness and helical angles of vortex generators significantly affect both pressure and velocity in the pipes. The following conclusions have been drawn from the examination of numerical analysis and simulation results:

- While the flat plate (0°) provides the most homogeneous flow distribution, the 270° helical angle shows the most turbulent flow distribution.

- As the roughness value increased (from 0.04 to 0.092), significant changes in pressure values were observed for each helical angle. Notably, the highest pressure values were observed for the roughness value of 0.04.

- The increase in surface roughness has also led to decreases in flow velocity; this finding confirms that roughness increases flow resistance, thereby reducing the velocity.

- The combination of helix angle and surface roughness significantly affects flow performance; it has been concluded that helix angle and roughness need to be carefully optimized for optimal flow performance. These findings are crucial for vortex generator designs to minimize pressure losses and control flow speed.

The results show that the optimal use of helical angle and roughness can contribute to thermal management in the automotive industry. This would lead to more efficient cooling systems and improved performance of heat recovery units. It would be beneficial to explore different fluid types, flow rates and temperature ranges for automotive applications.

Conflict of Interest Statement

The authors declare that there is no conflict of interest in the study.

CRediT Author Statement

Fuat Tan: Writing-original draft, Supervision, Analysis, Validation,
Alp Eren Dede: Numerical Analysis, Data curation

References

[1] Siddique H, et al. Effect of swirl flow on heat transfer characteristics in a circular pipe. Paper presented at the AIP Conference Proceedings. 2016. http://dx.doi.org/10.1063/1.4958419

[2] Behçet R, Yakut AK, Argunhan Z. The effect of rotary, type placed in entrance of heat exchanger on heat transfer and frictional loss. Energy education science and technology part A: Energy science and research. 2021;28:239-248. http://dx.doi.org/10.31202/ecjse.871896

[3] Acır A, Canlı ME, Ata İ, Tanürün HE. Effects of a circular-shaped turbulator having varying hole numbers on energy and exergy efficiencies of a solar air heater. International Journal of Ambient Energy.2018;1-10. http://dx.doi.org/10.1080/01430750.2017.1423385

[4] Chen B, et al. Fluid dynamics and heat transfer investigations of swirling decaying flow in an annular pipe Part 2: Fluid flow. International Journal of Heat and Mass Transfer. 2016;97:1012-1028. http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.07.129

[5] Zhang XJ, Hoshino K. Microfluidics and Micro Total Analytical Systems. Molecular Sensors and Nanodevices. 2014. http://dx.doi.org/10.1016/B978-0-12-814862-4.00003-X

[6] Bhatti MS and Shah RK. Turbulent and Transition Flow Convective Heat Transfer in Ducts. In Handbook of Single-Phase Convective Heat Transfer. ed. S. Kakaç. R. K. Shah. and W. Aung. New York: Wiley Interscience. 1987.

[7] Fang X, Xu L, Chen Y, Chen W. Correlations for friction factor of turbulent pipe flow under supercritical pressure: Review and a new correlation. Progress in Nuclear Energy. 2019. http://dx.doi.org/10.1016/j.pnucene.2019.103085

[8] Yilmaz M, et al. Enhancement of heat transfer by turbulent decaying swirl flow. Energy conversion and management. 1999;40(13):1365-1376. http://dx.doi.org/10.1016/S0196-8904(99)00030-8

[9] Bilen K, et al. Thermo-hydraulic performance of tube with decaying swirl flow generators. Applied thermal engineering. 2022; 200:117643.http://dx.doi.org/10.1016/j.applthermaleng.2021.117643

[10] Yeşildal F. Numerical Optimization of Heat Transfer Parameters in a Pipe with Decaying Swirl Flow Generators Using Response Surface Methodology. International Journal of Innovative Research and Reviews. 2021;5(2):9-14

[11] Banerjee C, Urankar S, Raikar S, Suresh D. Numerical Study of Decaying Swirling Flow in an Annulus. Engineering Applications of Computational Fluid Mechanics. 2020;14(1):1198-1214.

[12] Rocha AD, Bannwart AC, Ganzarolli MM. Numerical and experimental study of an axially induced swirling pipe flow. International journal of heat and fluid flow. 2015;53:81–90. https://doi.org/10.1016/j.ijheatfluidflow.2015.02.003

[13] Saqr KM, et al. Effects of swirl intensity on heat transfer and entropy generation in turbulent decaying swirl flow. Applied thermal engineering. 2014;70(1):486-493. https://doi.org/10.1016/j.applthermaleng.2014.05.059

[14] Kurtbaş İ, Gülçimen F, Kılıçarslan A, Kaya M. Effect of swirl generator inserted into a tube on exergy transfer: decaying flow. Experimental Heat Transfer. 2014;27(5):472–487. http://dx.doi.org/10.1080/08916152.2013.803175

[15] Baysal E, Solmaz Ö, Ökten M, Başeski Y. İç İçe Borulu Isı Değiştiricide Zıt Akışta Türbülatör Kullanımının Basınç Değişimine Etkisinin Sayısal Olarak İncelenmesi. El-Cezeri Fen ve Mühendislik Dergisi. 2021;8(2):817-826. https://doi.org/10.31202/ecjse.871896

[16] Solmaz Ö, Baysal E, Ökten M. Numerical Investigation and Machine Learning-Based Prediction of the Effect of Using Ring Turbulators on Heat Transfer Characteristics in a Counterflow Heat Exchanger. Recent Innovations in Chemical Engineering. 2023;16(5):362-378. http://dx.doi.org/10.2174/0124055204273603231004071130

[17] Karakaya H, Durmuş A. Heat Transfer and Exergy Loss in Conical Spring Turbulators. International Journal of Heat and Mass Transfer. 2013;60:756-762. https://doi.org/10.1016/j.ijheatmasstransfer.2013.01.054

[18] Özakın AN, Öner İV. Yatay Bir Boruda Dışarıdan Oluşturulan Farklı Geometrilerdeki Dönel Akışın Isı ve Akış Karakteristiklerinin Sayısal Analizi. The Black Sea Journal of Sciences.2022;12(1):414-424. https://doi.org/10.31466/kfbd.1085259

[19] Thabet S, Thabit T.H, Jasim Y.A. CFD Analysis of a Backward Facing Step Flows. International Journal of Automotive Science and Technology 2018; 2(3): 10-16. https://doi.org/10.30939/ijastech..447973

[20] Alp Arslan T, Bayrakçeken H, Yavuz H. CFD Analysis of sloshing in the fuel tank of a heavy vehicle. International Journal of Automotive Science and Technology 2023; 7(4): 340-348. https://doi.org/10.30939/ijastech..1360466