International Journal of Innovative Research and Reviews, 9(2) 59-66

International Journal of Innovative Research and Reviews

ISSN: 2636-8919

Website: www.injirr.com

RESEARCH ARTICLE

Numerical Investigation of the Effects of Different Turbulators on Heat Transfer in an Internal Flow System

Şendoğan KARAGÖZ[1], Hossein ABDI[1], Orhan YILDIRIM[1,*]

[1] *Department of Mechanical Engineering, Faculty of Engineering, Atatürk University, Erzurum, Türkiye*

** Corresponding author E-mail: orhan.yildirim@atauni.edu.tr*

ARTICLE INFO

Received : 07.04.2025
Accepted : 10.10.2025
Published : 12.15.2025

Keywords:
Heat transfer
CFD
Turbulator
Friction factor
Pressure drop

ABSTRACT

In this study, the effects of different turbulator geometries—specifically spring-type and twisted-tape turbulators—on heat transfer and flow characteristics in internal pipe systems were investigated using the Computational Fluid Dynamics (CFD) method. A straight pipe with a length of 1020 mm and a diameter of 12.7 mm was used as the reference model. Three different pitch lengths were evaluated for each turbulator type: 23 mm, 43 mm, and 63 mm for the springs, and 115 mm, 215 mm, and 315 mm for the twisted-tapes. The geometry and meshing were prepared using Gambit software, while numerical simulations were conducted in ANSYS Fluent, employing the Standard k-ε turbulence model. The CFD results demonstrated that the inclusion of turbulators significantly enhanced heat transfer compared to the plain pipe. The highest Nusselt numbers were obtained with the shortest-pitch configurations, namely the 23 mm spring and the 115 mm twisted-tape. However, this enhancement was accompanied by an increase in the friction factor and pressure drop, particularly as the turbulator density increased. Temperature distribution analysis revealed that in the plain pipe, cold fluid accumulated toward the center, whereas in turbulated configurations, heat distribution became more uniform and wall temperatures were reduced. Pressure field analysis also showed that decreasing the pitch of springs and twisted-tapes led to greater resistance and increased pressure loss along the pipe. The results are in strong agreement with both experimental observations and relevant literature, underscoring the importance of turbulator geometry and arrangement in optimizing the thermo-hydraulic performance of internal pipe flows.

Contents

Cite this article Karagöz Ş, Abdi H, Yıldırım O. Numerical Investigation of the Effects of Different Turbulators on Heat Transfer in an Internal Flow System. *International Journal of Innovative Research and Reviews (INJIRR)* (2025) 9(2) 59-66

Link to this article: http://www.injirr.com/article/view/252

1. Introduction

The growing global demand for energy has made it imperative to utilize available resources more efficiently. In this context, improving the thermal performance of heat exchangers plays a vital role in enhancing energy efficiency, reducing losses, and optimizing resource utilization. Key strategies to improve heat transfer include increasing the surface area, enhancing turbulence levels by introducing surface roughness, and thereby increasing the convective heat transfer coefficient. To achieve these goals, turbulators with various geometries and materials have been developed and widely used in heat exchangers, particularly in research and development applications due to their ease of installation and minimal need for system redesign [1–3].

Numerous experimental and numerical studies in the literature have examined the effects of various turbulators on heat transfer and pressure drop in turbulent flow regimes. For instance, Kaya [4] reported only a 5% deviation between numerical and experimental results in a square cross-sectional helical channel under turbulent flow conditions. Xie et al. [5] demonstrated a 58% improvement in thermal performance using tubes with ellipsoidal dimples, while Hong et al. [6] observed up to a 14% enhancement with modified wire coil inserts. Chiu and Jang [7] highlighted that twisted-tape inserts yield higher pressure drops but also better heat transfer performance compared to straight longitudinal inserts. Karagöz [8] employed response surface methodology to examine how geometric parameters such as element height (H), transverse pitch (Sy), element radius (R), and Reynolds number (Re) influence Nusselt number(Nu) and friction factor(f) for S-shaped turbulators. The study developed correlation equations and concluded that height, transverse pitch, and Re significantly affect friction, while the radius has a negligible impact.

Kaya [4] also numerically investigated pressure drop in a square cross-sectional helical channel using Fluent software and the RNG k-ε turbulence model, reporting a maximum 5% deviation between simulations and experimental findings. Similarly, Karouei and Mousavi Ajarostaghi [9] studied flow and heat transfer in a helical double-pipe heat exchanger with a 12-blade curved conical turbulator, finding that increasing the turbulator's inner radius by 26.7% enhanced performance by 80%, while enlarging the hole radius by 133.34% led to a 50% efficiency gain.

In another approach, spherical turbulators were numerically analyzed as a novel method to enhance heat transfer by inducing vortex generation. The study tested three different ball diameters under four Re (5000–20000) and found that the proposed design generated stronger vortices at lower Re, yielding higher heat transfer efficiency [10]. Vorayos and Kiatsiriroat [11] experimentally analyzed louvered fin heat exchangers and reported that reducing the fin aspect ratio at low Re allowed more heated air to interact with the fins, boosting thermal performance. Likewise, Chai and Tassou [12] reviewed vortex generators and suggested that their application on the air side could significantly improve thermo-hydraulic performance.

Recent advancements include the use of aerodynamic turbulators with novel configurations. For instance, systems employing perforated spherical turbulators with oval holes achieved up to a 2.81-fold increase in Nu compared to smooth pipes [13]. Ring-shaped turbulators have also demonstrated 34–54% higher Nu values while maintaining relatively low pressure drop [14]. Vibrational ball-type turbulators have proven effective in enhancing heat transfer in low Re flows by promoting strong vortex formation [15]. Twisted-tape and spiral turbulators further boost heat transfer through intensified turbulence and fluid mixing, though they often increase pressure losses. Spiral-ribbed and perforated designs have shown 17–34% higher Nu compared to smooth tubes [3, 7]. The improved contact surface area in spring and twisted-tape turbulators also contributes significantly to heat transfer efficiency [4, 5, 7].

This study numerically investigates how spring and twisted-tape turbulators with different pitch values (23, 43, 63 mm for springs; 115, 215, 315 mm for tapes) affect heat transfer and f in a horizontally pipe. The primary goal was to compare these configurations under identical boundary conditions and determine which design offers the most favorable thermal performance despite associated frictional losses. The outcomes aim to inform the selection of optimal turbulator designs for enhanced energy efficiency in internal flow systems.

2. Material and Method

2.1. Data Description

In this study, the effects of spring and twisted-tape turbulators with various pitch lengths on heat transfer and pressure drop in internal pipe flows were investigated experimentally. Water was used as the working fluid, and the performance of the turbulator-integrated systems was also assessed through comparative theoretical analysis [1, 2]. The test section consisted of a straight pipe fabricated from 316L stainless steel, with a total length of 1020 mm and an inner diameter of 12.7 mm (Figure 1). To minimize ambient heat losses, the pipe was thoroughly insulated along its outer surface. Experimental tests were conducted using three different pitch lengths for each turbulator type: 63 mm, 43 mm, and 23 mm for the springs, and 315 mm, 215 mm, and 115 mm for the twisted tapes (Figure 2). The experiments were performed under five distinct flow rates: 615, 511, 409, 307, and 205 L/h [1].

Figure 1 General view of the experimental apparatus

Thermal energy was supplied using a power unit with a maximum capacity of 1192 W, and the system operated at a constant pressure of 1 bar. Flow rates were measured via a turbine-type flowmeter integrated into the setup. Pressure drop across the test section was monitored using two pressure sensors placed at the inlet and outlet. For temperature measurement, 24 thermocouples were strategically positioned along the length of the pipe, and data were collected using an Advantech PC-LabCard for computer-based recording and processing.

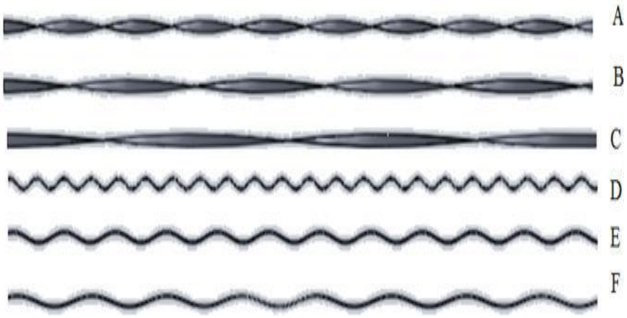

Figure 2 Types of turbulators used in the analysis

To ensure continuous fluid circulation, a circulation pump was included, while temperature stabilization was managed via a heat exchanger. In addition, an expansion tank was used to eliminate trapped air and maintain pressure stability throughout the system.

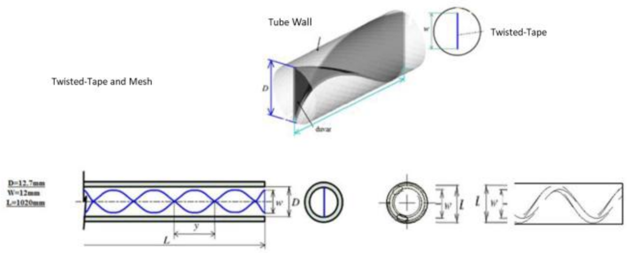

Figure 3 SolidWorks models of the twisted-tape and spring elements used in the analysis

This experimental setup was carefully designed to enable accurate and consistent analysis of how different turbulator geometries and placement densities influence heat transfer efficiency and pressure loss. The experimental results were also utilized to support and validate the numerical analysis conducted in the study.

1.1. Computational Fluid Dynamics (CFD) solution

In this study, a three-dimensional (3D) numerical analysis was conducted for internal pipe systems without turbulators and with spring and twisted-tape turbulators at various pitch lengths (Figure 2 and Figure 3). The geometric modeling of the configurations was carried out using SolidWorks, while the mesh structures were generated using Gambit. All flow and heat transfer simulations were performed in ANSYS Fluent.

For the smooth pipe case, a mesh consisting of triangular surface elements was created, comprising a total of 47,034 cells and 11,232 nodes (Figure 4 and Figure 5). In turbulator-equipped configurations, mesh independence was tested by comparing models with 50,000, 100,000, 500,000, and 1,000,000 cells. Since the simulation results between 500,000 and 1,000,000 cells showed negligible differences, the mesh structure with 500,000 cells was selected for all subsequent analyses to maintain a balance between computational efficiency and solution accuracy [1, 16].

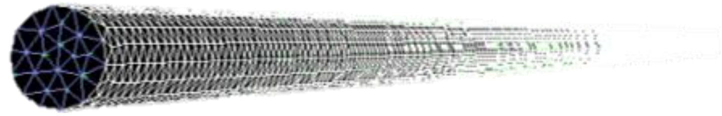

Figure 4 Mesh structure generated for the plain pipe

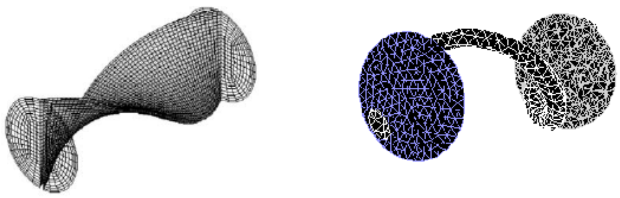

Figure 5 Mesh structure generated for spring and twisted-tape element

In the numerical simulations, the fluid was assumed to be turbulent and incompressible. The Standard k–ε turbulence model, which is widely used in engineering applications, was selected to model turbulence effects. Near-wall behavior was treated using standard wall functions. The governing equations included in the model comprise the continuity equation, the momentum equation, the turbulent kinetic energy equation (k), and the turbulent dissipation rate equation (ε).

Boundary Conditions:

1. Inlet fluid temperature: 293 K
2. Outlet: Atmospheric pressure (p = 1 atm)
3. Constant heat flux at pipe wall: 29,290 W/m²
4. Turbulence intensity at inlet: 4.73%
5. Hydraulic diameter: 0.0127 m
6. Re range: 6000 <Re <18000

Under these boundary conditions, the simulations aimed to assess the heat transfer characteristics—specifically the Nu, pressure losses (f), and temperature distributions along the pipe. Numerical results were validated by comparing them with experimental data, and a mesh independence study was conducted to ensure the accuracy of the results [1, 16].

Equation (1) represents the *continuity equation* for incompressible flows, expressing the principle of mass conservation at any point within the domain. Written using Einstein notation, it states that the sum of the velocity gradients in all spatial directions must be zero.

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

Equation (2) represents the Reynolds-Averaged Navier–Stokes (RANS) *momentum equation* for incompressible turbulent flows. On the right-hand side of the equation, the first term accounts for the pressure gradient, while the second term represents the combined viscous and turbulent stress tensors. The effective viscosity (μ_e) used in this formulation includes both molecular and turbulent viscosity contributions.

$$\rho \frac{u_i u_j}{\partial x_i} = -\frac{\partial p}{\partial x_j} + \frac{\partial[\mu_e(\frac{u_i}{\partial x_j} + \frac{u_j}{\partial x_i})]}{\partial x_i} \tag{2}$$

Equation (3) describes the *energy transport equation* under turbulent flow conditions, incorporating heat transfer due to turbulence. The turbulent heat flux term v′T′ indicates that energy transfer occurs not only via molecular conduction but also through fluctuating turbulent motion, which enhances thermal mixing.

$$\rho C_p(\overline{u}\frac{\partial \overline{T}}{\partial x} + \overline{v}\frac{\partial \overline{T}}{\partial y}) = \frac{\partial(k\frac{\partial \overline{T}}{\partial y} - \rho C_p \overline{v'T'})}{\partial y} \tag{3}$$

Equation (4) governs the transport of *turbulent kinetic energy (k),* accounting for convection, diffusion, production, and dissipation mechanisms. The production term G_k represents the generation of turbulent kinetic energy, while ε denotes its dissipation rate.

$$\rho \frac{\partial u_i k}{\partial x_i} = \frac{\partial[\frac{\mu_e}{\sigma_k}\frac{\partial}{k}]}{\partial x_i} + G_k - \rho\varepsilon \tag{4}$$

Equation (5) models the transport of the *turbulent dissipation rate (ε),* which controls how rapidly turbulent kinetic energy is dissipated within the flow. It includes terms for convection, diffusion, and source/sink dynamics related to turbulence intensity.

$$\rho \frac{\partial u_i \varepsilon}{\partial x_i} = \frac{\partial[\frac{\mu_e}{\sigma_\varepsilon}\frac{\partial}{\varepsilon}]}{\partial x_i} + C_{\varepsilon 1}\frac{\varepsilon}{k}G_k - C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} \tag{5}$$

Equation (6) defines the *total effective viscosity* (μ_e) as a combination of molecular viscosity and turbulent viscosity. This formulation ensures an accurate representation of viscous effects in turbulent regimes.

$$\mu_e = \mu + \mu_t \tag{6}$$

In turbulent flows, *turbulent viscosity (μ_t)* is calculated based on the turbulent kinetic energy (k) and its dissipation rate (ε). The empirical constant C_μ is a critical calibration factor for ensuring the accuracy of the turbulence model, as shown in Equation (7):

$$\mu_t = C_\mu \frac{\rho k^2}{\varepsilon} \tag{7}$$

Equation (8) represents the production of turbulent kinetic energy due to shear stress. It is one of the primary source terms in turbulence modeling, as it characterizes the energy transferred from mean flow to turbulence.

$$G_k = \mu_t \frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{8}$$

3. Results and Discussion

In this study, the effects of turbulators with different geometries and pitch lengths—specifically spring-type and twisted-tape configurations—on heat transfer and flow characteristics were investigated numerically. The CFD simulations provided data on the Nu, f, as well as temperature and pressure distributions, which were then evaluated in comparison with both experimental results and findings reported in the literature.

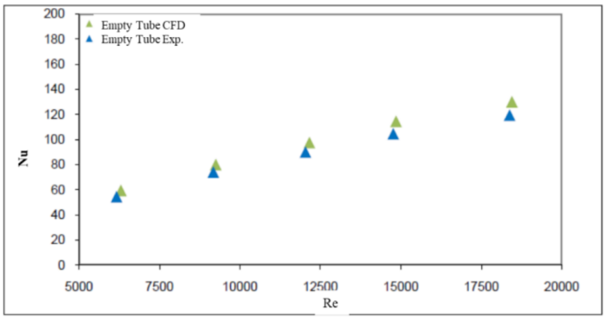

Figure 6 Comparison of Re–Nu variation in the plain pipe

Figure 6 presents the variation of the Nu with respect to the Re in a smooth pipe, based on both experimental and numerical data. As observed, the numerical and experimental results are in close agreement, demonstrating the accuracy and reliability of the computational modeling approach. The CFD predictions deviate by approximately 3% from the experimental results and by around 2% when compared with the Dittus–Boelter correlation. These minor discrepancies confirm that the model effectively represents the physical reality.

The experimental data were sourced from the study conducted by Abdi (2014), which also found a positive correlation between Re and Nu. Similar trends and consistency between experimental and CFD results have also been reported in the literature. For example, Vahidifar and Banihashemi [17] showed that CFD simulations for pipes with ring-type turbulators deviated from experimental values by only 2–4%. Additionally, Eiamsa-ard et al. [18, 19] demonstrated that the Dittus–Boelter correlation is a reliable benchmark for plain pipe flows, with CFD results typically falling within a 1–3% error range. Altun et al. [20] similarly reported that an increase in Re leads to a significant rise in Nu, attributing this to enhanced turbulence intensity.

Overall, the theoretical and experimental findings are in strong agreement with each other and with prior research, reinforcing the validity of CFD simulations as a dependable tool for future turbulator designs and flow system analyses.

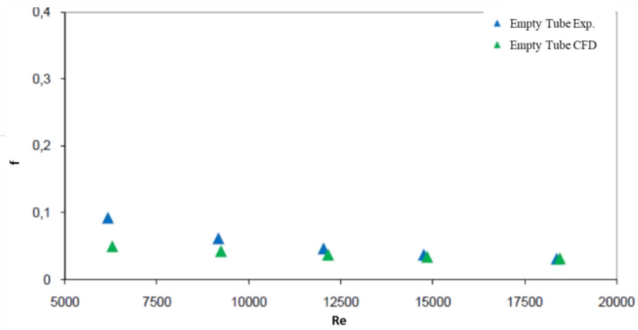

Figure 7 Comparison of Re–f variation in the plain pipe

Figure 7 illustrates the change in f with respect to the Re for a smooth pipe, using both experimental and numerical results. The data clearly indicate that as the Re increases, the f decreases. This trend is consistent with classical fluid mechanics theory, where thinner boundary layers at higher flow rates lead to reduced viscous effects and hence lower frictional losses.

In particular, CFD results tend to underestimate the f compared to experimental data, especially at lower Re. This is likely due to the numerical model's closer alignment with the Blasius correlation, which is more accurate in transitional regimes. The Blasius equation ($f = 0.3164/Re^{0.25}$), commonly used for turbulent flow in smooth pipes, was found to deviate from both experimental and CFD data by approximately 5% [18, 19]. The difference between CFD and experimental values was around 7%, which could be attributed to measurement uncertainty, surface roughness, or temperature fluctuations in the experimental setup.

Nonetheless, the consistency of the overall trend confirms the reliability of the CFD model. Supporting this, Vahidifar and Banihashemi [17] found that deviations in the f between CFD and experimental studies for pipes with ring turbulators ranged between 5–8%, which they deemed acceptable.

The findings presented in Figure 7 show that the CFD model aligns well with both experimental results and empirical correlations in terms of f. This supports the use of CFD as a powerful engineering tool for analyzing internal pipe flow systems.

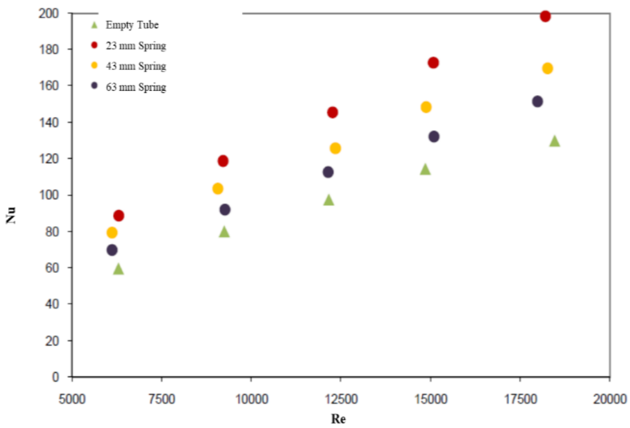

Figure 8 CFD-based comparison of Re–Nu for spring-type turbulators

Figure 8 illustrates the variation of the Nu with Re for spring-type turbulators with different pitch lengths (23 mm, 43 mm, and 63 mm) placed inside a smooth pipe, based on numerical simulations (CFD). The figure clearly shows that as the pitch decreases, meaning the turbulator elements are positioned more closely together, the heat transfer performance improves significantly. Among all configurations, the 23 mm spring pitch yielded the highest Nu values across the entire Re range. Accordingly, the following order was observed: $Nu_{(23)} > Nu_{(43)} > Nu_{(63)}$. This is attributed to the increased turbulence generated by denser turbulator arrangements, which enhance boundary layer disruption and promote more effective convective heat transfer.

This finding is consistent with several studies in the literature. For instance, Budak et al. [21] demonstrated in their comparative analysis of different inlet turbulators that increasing turbulator density (i.e., decreasing pitch) significantly boosts Nu values. Similarly, studies on ring-type turbulators have shown that tightly spaced elements can enhance heat transfer by up to 70% [14]. Additionally, all spring-type turbulator configurations outperformed the smooth pipe baseline in terms of Nu across the entire Re range. This improvement is primarily due to the vortex structures generated by the turbulators, which weaken the thermal boundary layer and thus accelerate heat transfer. These CFD-predicted trends align closely with experimental results. Notably, Vahidifar and Banihashemi [17] reported 40–80% increases in heat transfer for turbulator-enhanced systems compared to smooth pipes within similar Re ranges. In conclusion, the data presented in Figure 8 demonstrate that turbulator geometry and pitch spacing play a decisive role in enhancing heat transfer. Shorter spring pitches yield stronger turbulence effects, leading to higher Nu values—an outcome strongly supported by both theoretical and experimental literature.

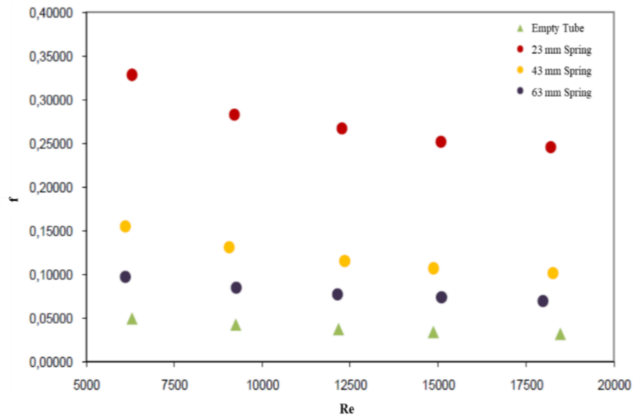

Figure 9 CFD-based comparison of Re–f for spring-type turbulators

Figure 9 presents the variation of the f with Re for spring-type turbulators with different pitch lengths (23 mm, 43 mm, and 63 mm) placed inside a smooth pipe. The results clearly show that a shorter spring pitch, which increases the density of coil windings, leads to higher f values. For all Re, the ordering was consistent: $f_{(23)} > f_{(43)} > f_{(63)}$. This indicates that denser turbulator configurations result in greater flow resistance and, consequently, higher internal pipe friction. For all configurations, the f decreases as the Re increases. This trend is attributed to the relative reduction in viscous effects at higher flow velocities, despite increased turbulence. These results are in agreement with findings reported in various studies. For example, Altun et al. [20] observed that placing sinusoidal turbulators more closely within the pipe not only increased Nu but also significantly elevated the f.

Budak et al. [21], in their comparative analysis using spring-like turbulators, found that the shortest pitch configuration resulted in the highest pressure drop and f, confirming the strong relationship between turbulator density and flow resistance. These findings support the CFD results and demonstrate a high level of correlation with experimental observations. The Re–f relationship shown in Figure 9 clearly reflects how decreasing spring pitch increases fin density and leads to a corresponding rise in the f. The CFD outcomes are well aligned with documented trends in the literature, reinforcing the validity of the numerical model and its applicability in engineering design and optimization.

Figure 10 presents a comparative analysis of temperature distributions within pipes fitted with spring-type turbulators of different pitch lengths (23 mm, 43 mm, and 63 mm) and a reference smooth pipe. The images display both longitudinal and cross-sectional temperature profiles.

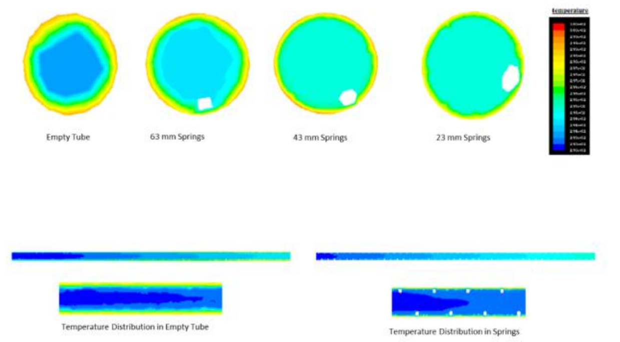

Figure 10 Comparison of temperature distributions from CFD results for spring-type turbulators

In the smooth pipe configuration, clear thermal stratification is observed, with low-temperature fluid concentrated near the centerline, indicating that the boundary layer remains largely undisturbed and that the primary flow path remains centralized. In contrast, the presence of spring-shaped turbulators leads to a noticeably more uniform temperature distribution. Particularly with the shortest pitch (23 mm), heat is more effectively transferred toward the pipe walls, and cold fluid appears to be pushed outward from the center. This demonstrates that the vortices generated by the turbulators enhance mixing, disrupt the thermal boundary layer, and significantly improve convective heat transfer. Supporting this, Assari et al. [13] reported that oval-perforated turbulators notably reduce wall temperatures, resulting in superior heat transfer performance.

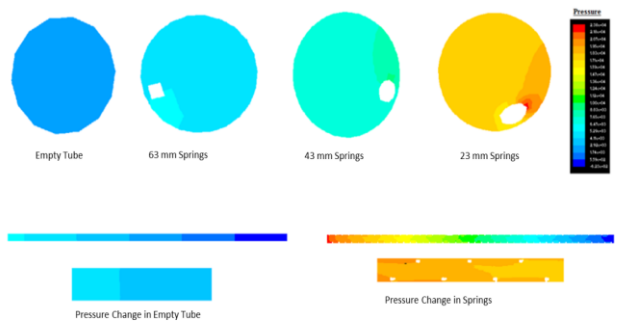

Figure 11 Comparison of pressure distributions from CFD results for spring-type turbulators

Figure 11 illustrates the pressure distribution along the pipe for each configuration. While the smooth pipe maintains a low and uniform pressure profile throughout the flow domain, the presence of spring-type turbulators increases overall pressure levels. This pressure rise becomes more pronounced as the pitch decreases, indicating that closely spaced coils impose greater flow resistance and thereby contribute to higher pressure losses. This finding is consistent with the results of Budak et al. [21], who also reported increased pressure drop for turbulators with smaller pitch spacing. Furthermore, the flow in the smooth pipe proceeds more uniformly and with minimal resistance, whereas the vortex structures induced by spring turbulators generate stronger pressure gradients, which may affect energy consumption in practical applications. This observation highlights the classic engineering trade-off between improved heat transfer performance and increased pressure loss in thermal-fluid system design.

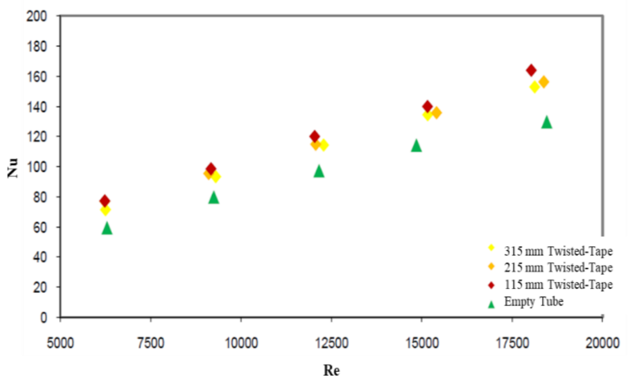

Figure 12 CFD-based comparison of Re–Nu for twisted-tape turbulators

Figure 12 illustrates the influence of twisted-tape turbulators with three different pitch lengths (115 mm, 215 mm, and 315 mm) on the Nu as a function of Re in a smooth pipe. The CFD results show a clear trend: as the pitch length decreases, and thus the turbulator density increases, the heat transfer performance improves significantly. For all Re, the following relationship holds: $Nu_{115} > Nu_{215} > Nu_{315}$. This enhancement is attributed to the intensified turbulence and mixing caused by the more closely spaced tape elements, which effectively disturb the boundary layer and promote convective heat transfer. These findings are consistent with the studies of Chiu and Jang [7] and Budak et al. [22]. Chiu and Jang reported that twisted-tape inserts result in higher heat transfer rates compared to straight strips, while Budak et al. confirmed that increasing the turbulator density leads to substantial increases in Nu values.

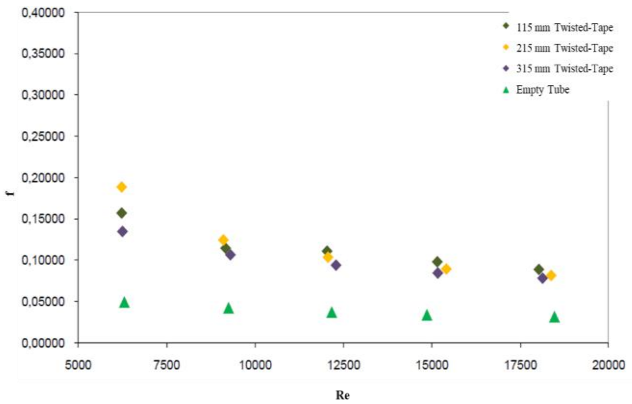

Figure 13 CFD-based comparison of Re–f for twisted-tape turbulators

Figure 13 shows the corresponding effects of the same configurations on the f. The CFD results indicate that decreasing the pitch length also leads to an increase in friction losses. For all Re, the trend follows: $f_{115} > f_{215} > f_{315}$. This is due to the greater flow obstruction caused by a higher number of twisted-tape elements, which increases turbulence and disrupts the flow, leading to elevated pressure drops. Additionally, the f tends to decrease with increasing Re for all configurations. This behavior is explained by the relatively reduced viscous effects at higher flow rates, despite the stronger turbulence. Similar observations have been reported in the literature by Eiamsa-ard et al. [18, 19] and Altun et al. [20]. In conclusion, shorter-pitch twisted-tape turbulators are shown to increase both the Nu and the f. This illustrates the classic engineering trade-off between improved heat transfer and increased pressure loss. The results of this analysis demonstrate strong alignment between the CFD findings and existing literature, supporting the reliability of the numerical approach.

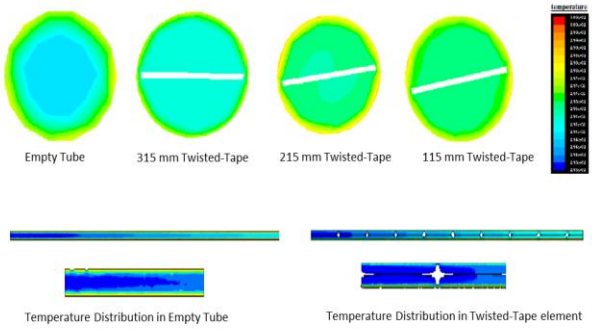

Figure 14 CFD-based comparison of temperature distributions for twisted-tape turbulators

Figure 14 presents a comparative analysis of temperature distributions in a smooth pipe and in pipes equipped with twisted-tape turbulators of varying pitch lengths (315 mm, 215 mm, and 115 mm). Visual inspection reveals that, in the smooth pipe, the temperature field is highly stratified, with cold fluid predominantly concentrated in the central region, indicating weak convective mixing and limited boundary layer disruption. In contrast, the twisted-tape turbulated pipes exhibit a more uniform temperature distribution, with significantly improved heat transfer toward the pipe wall. This effect becomes more pronounced as the pitch is reduced (e.g., 115 mm), resulting in lower wall temperatures and enhanced heat transfer. The stronger vortical motions generated by the denser turbulator arrangement effectively disturb the boundary layer and promote better fluid mixing across the pipe cross-section. Similar findings were reported by Saadat et al. [16], who observed increased thermal uniformity with shorter spacing in vibrational turbulator configurations.

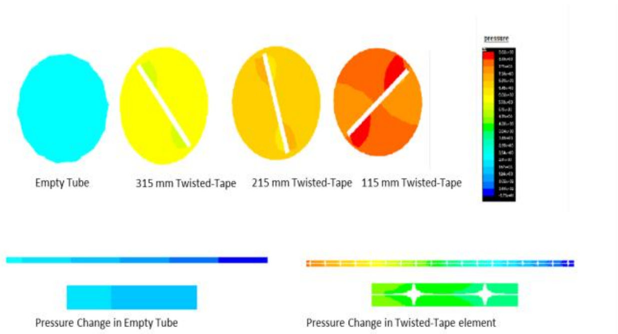

Figure 15 CFD-based comparison of pressure variation for twisted-tape turbulators

Figure 15 illustrates the effect of twisted-tape turbulators on pressure variation. While the smooth pipe shows a low and uniform pressure distribution, the presence of twisted-tape inserts leads to a marked increase in pressure levels. The highest pressure rise was observed for the shortest pitch (115 mm), which can be attributed to the increased number of turbulators, resulting in greater flow resistance and intensified turbulence. Consistent with this, Budak et al. [21] highlighted that turbulator geometry and density significantly impact pressure losses. Moreover, shorter pitch configurations exhibited steeper pressure gradients along the pipe, indicating higher energy losses. This underlines the importance of balancing heat transfer enhancement and pressure drop in thermal-fluid system design.

When comparing Nu, all turbulated configurations yielded substantially higher heat transfer performance than the smooth pipe. The highest Nu values were obtained with the densest arrangements—23 mm spring pitch and 115 mm twisted-tape pitch—demonstrating that closely spaced turbulators enhance fluid mixing and heat conduction to the pipe wall by disturbing the thermal boundary layer.

In terms of f, turbulated pipes exhibited significantly higher values compared to the smooth pipe. A reduction in pitch length led to more turbulators, which increased flow resistance and pressure drop. This effect was especially noticeable at lower Re. These results are in strong agreement with the Blasius correlation and with similar CFD and experimental studies reported in the literature [17].

Regarding temperature field behavior, the smooth pipe showed central cold zones with limited boundary layer disruption, while the turbulated cases revealed more uniform temperature distributions and lower wall temperatures. This indicates that heat transfer efficiency is supported not only by high Nu values but also by favorable thermal field development.

Lastly, pressure distribution results demonstrated that while the smooth pipe maintained low and stable pressure, the addition of turbulators led to increased pressure levels and drops. This implies that while heat transfer is significantly improved, the pumping power requirement of the system also increases—a classic trade-off in engineering applications.

4. Conclusion

According to the CFD analyses, both spring-type and twisted-tape turbulators significantly enhance heat transfer performance. The highest heat transfer rates were achieved with the shortest pitch configurations (23 mm for spring and 115 mm for twisted-tape turbulators). The use of turbulators also resulted in an increase in the f, leading to higher pressure losses. Temperature distribution became more uniform in turbulated systems, indicating more efficient heat transfer along the pipe surface. As the number of turbulators increased, the pressure drop across the system also rose, which may contribute to higher energy consumption.

Future Research

Turbulator design should consider not only heat transfer performance but also pressure losses, aiming for an optimized thermo-hydraulic balance. The 23 mm spring or 115 mm twisted-tape turbulators may be recommended for systems requiring enhanced heat transfer, but the increased pumping power demand must be accounted for. Future studies could explore the use of nanofluids, hybrid turbulator geometries, or passive/active system combinations to further improve thermal efficiency. To ensure industrial applicability, experimental validation should be extended to better support and confirm CFD predictions.

Author Contributions

SK: Conceptualization, Methodology, Validation, Resources, Writing – review & editing, Supervision, Project administration.

HA: Conceptualization, Methodology, Software, Validation, Formal analysis, Investigation, Data curation, Writing – original draft, Visualization.

OY: Methodology, Validation, Visualization, Writing – original draft, Writing – review & editing,

Acknowledgements

This article is derived from a portion of the master's thesis prepared by Hossein Abdi, titled "Experimental and Theoretical Investigations of the Impact of Turbulators in Horizontal Pipes on Heat Transfer" (Abdi, 2014), at Atatürk University, Erzurum, Turkey.

The authors would like to express their sincere gratitude to Hossein Abdi for his valuable support and contributions throughout the studies.

Conflict of Interest

The authors have no conflicts of interest to declare.

References

[1] Abdi H. *Yatay borularda türbülatörlerin ısı geçişine olan etkisinin deneysel araştırılması [Experimental and Theoretical Investigations of the Impact of Turbulators in Horizontal Pipes on Heat Transfer]*. Master Thesis. Atatürk University. Erzurum, Türkiye (2014).

[2] Çiltaş S. *Yatay borularda türbülatörlerin ısı geçişine olan etkisinin deneysel araştırılması [Experimental Investigations of the Impact of Turbulators in Horizontal Pipes on Heat Transfer]*. Master Thesis. Atatürk University. Erzurum, Türkiye (2012).

[3] Karagöz Ş, Çiltaş S, Yıldırım O, Erdoğan S. Yatay borularda türbülatörlerin ısı transferine olan etkisinin deneysel araştırılması [Experimental Investigation of the Effect of Turbulators on Heat Transfer in Horizontal Pipes]. *Erzincan University Journal of Science and Technology* (2019) **12**(1):306–316. doi:10.18185/erzifbed.459957.

[4] Kaya O. Kare kesitli helisel kanalda türbülanslı akışta basınç düşüşünün deneysel ve numerik analizi [[The Experimental and Numerical Analysis of Turbulent Flow Pressure Drop of Helical Square Duct]]. *Pamukkale Üniversitesi Mühendislik Bilimleri Dergisi* (2004) **10**(2):283–289.

[5] Xie S, Liang Z, Zhang L, Wang Y. A numerical study on heat transfer enhancement and flow structure in enhanced tube with cross ellipsoidal dimples. *International Journal of Heat and Mass Transfer* (2018) **125**:434–444.

[6] Hong Y, Du J, Wang S, Huang SM, Ye WB. Heat transfer and fluid flow behaviors in a tube with modified wire coils. *International Journal of Heat and Mass Transfer* (2018) **124**:1347–1360.

[7] Chiu YW, Jang JY. 3D numerical and experimental analysis for thermal–hydraulic characteristics of air flow inside a circular tube with different tube inserts. *Applied Thermal Engineering* (2009) **29**:250–258.

[8] Karagöz S. Investigation of thermal performances of "S-shaped" enhancement elements by response surface methodology. *Heat and Mass Transfer* (2015) **51**(2):251–263.

[9] Karouei SH, Mousavi Ajarostaghi SS. Influence of a curved conical turbulator on heat transfer augmentation in a helical double-pipe heat exchanger. *Heat Transfer* (2021) **50**(2):1872–1894.

[10] Murata A, Sadanari M. Comparison between laminar and turbulent heat transfer in a stationary square duct with transverse or angled rib turbulators. *International Journal of Heat and Mass Transfer* (2001) **44**(6):1127–1141.

[11] Vorayos N, Kiatsiriroat T. Thermal characteristics of louvered fins with a low-Reynolds number flow. *Journal of Mechanical Science and Technology* (2010) **24**:845–850.

[12] Chai L, Tassou S. A review of airside heat transfer augmentation with vortex generators on heat transfer surface. *Energies* (2018) **11**(10). doi:10.3390/en11102700.

[13] Assari M, Banihashemi S, Setareh M, others. Effects of aerodynamic turbulator on efficiency improvement and heat transfer enhancement of the internal flow in a pipe. *Sādhanā* (2024) **49**:196. doi:10.1007/s12046-024-02519-7.

[14] Dirker J, Meyer JP, Steyn RM. Influence of ring type flow turbulators on the local heat transfer coefficients in an annular passage–An experimental and numerical investigation. *International Journal of Thermal Sciences* (2021) **168**:107052.

[15] Saadat M, Aghlichanche A, Ataelahi A, Mohammadi O, Shafii MB. Thermal study of the internal flow in a circular tube with vibrational ball turbulators. *International Journal of Heat and Mass Transfer* (2022) **196**:123276.

[16] ANSYS Inc. *Ansys Fluent user's guide (Release 15.0)* (2014).

[17] Vahidifar S, Banihashemi S. Experimental and numerical evaluation of heat transfer enhancement by internal flow excitation. *International Journal of Thermal Sciences* (2023) **192**(A):108395.

[18] Eiamsa-ard S, Thianpong C, Eiamsa-ard P. Turbulent heat transfer enhancement by counter/co-swirling flow in a tube fitted with twin twisted tapes. *Experimental Thermal and Fluid Science* (2010) **34**:53–62.

[19] Eiamsa-ard S, Thianpong C, Promvonge P. Thermal characteristics in a heat exchanger tube fitted with dual twisted tape elements in tandem. *International Communications in Heat and Mass Transfer* (2010) **37**:39–46.

[20] Altun AH, Gurdal M, Berber A. Effects of sinusoidal turbulator in cylindrical channel on heat transfer and flow characteristics. *Maejo International Journal of Science & Technology* (2020) **14**(1):27–42.

[21] Budak N, Yucel HL, Argunhan Z. Experimental and numerical investigation of the effect of turbulator on heat transfer in a concentric-type heat exchanger. *Experimental Heat Transfer* (2016) **29**(3):322–336.