

Article

Numerical Investigation of Heat Transfer Enhancement Using Novel “Ram-Horn” Internal Fins in a Tube

Abraham Adu-Mills, Kofi Yeboah Asante, Yashitha Thuraka and John P. Kizito *

Department of Mechanical Engineering, North Carolina Agricultural and Technical State University,
Greensboro, NC 27411, USA

* Correspondence: jpkizito@ncat.edu

Abstract

Most passive heat exchanger tube inserts only raise the Nusselt number by dramatically increasing friction, and studies on fin inserts have generally focused on low turbulent Reynolds numbers used in plant equipment. In this study, we numerically investigate a ram-horn internal fin whose branch radii follow a Fibonacci spiral, implemented in a 25 mm inner-diameter copper tube and solved using computational fluid dynamics software (ANSYS FLUENT 2026) with conjugate heat transfer for $Re = 10,000$ – $100,000$. The fin number ($N = 4, 6, 8$) and stem height–diameter ratio ($s/D = 0.06, 0.08, 0.1$) were individually varied to examine their effects on the Nu , f , and heat retention values. Water was used as the working fluid, entering the tube at 350 K and 4 bar, and a $50,000 \text{ W/m}^2$ outer-wall flux was imposed. Mesh independence and turbulent viscosity ratio sensitivity analyses were performed, with the results favoring the SST k – ω turbulence model. The plain tube sat close to the Petukhov friction factor correlation and agreed well with the Dittus–Boelter correlation, validating the plain pipe case. The results indicate that the proposed horn-like fins pin coherent vortices to the wall, increasing the near-fin turbulent kinetic energy. The thermal performance factor peaked at 1.46 ($N = 6, s/D = 0.06, Re = 80,000$), where the Nusselt number was 110% above that of the plain tube; heat retained by the fluid also rose by 14%. Constant pumping power analysis showed a 28% maximum heat transfer improvement when all finned pipe cases were examined. A multiple regression analysis yielded a correlation equation between Nu , Re , Pr , s/D , and N , which achieved $R^2 = 0.992$.

Keywords: heat transfer enhancement; conjugate heat transfer; heat exchangers; thermohydraulic performance; computational fluid dynamics

Academic Editor: Artur Blaszczyk

Received: 22 September 2026

Revised: 19 October 2026

Accepted: 23 October 2026

Published: 26 October 2026

Copyright: © 2026 by the authors.
Licensee MDPI, Basel, Switzerland.
This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

1. Introduction

In the pursuit of energy-efficient engineering systems, minimizing irreversible losses and recovering otherwise wasted thermal energy have become essential design priorities. Heat exchangers stand at the center of this effort—by enabling controlled heat transfer between two or more fluids that must remain unmixed, they convert what would otherwise be discarded enthalpy [1] into useful work or process heat. Their ubiquity across aircraft propulsion systems [2], regenerative power cycles [3], and solar thermal collectors [4] underscores both their technological maturity and their continuing importance with respect to sustainable energy utilization. Modern systems use several heat exchanger types, including shell-and-tube, plate, double-pipe, regenerative, and direct-contact exchangers. A key component in certain exchangers is a conduit through which fluid flows;

for example, in a shell-and-tube heat exchanger, this conduit is usually a cylindrical pipe. The walls of the conduit are made of a material that efficiently transfers heat from its inner wall, where one fluid flows, to the fluid flowing over it (or vice versa, depending on the direction of heat flow). The heat transfer efficiency depends on a key factor—the heat transfer coefficient h , which largely depends on the flow regime, the fluid's properties, and the surface area of the walls over which the fluid flows. Enhancing heat transfer increases the amount of heat energy transferred between fluids, thereby moving toward the goal of energy recovery and reuse. Contemporary research focuses on several methods intended to enhance heat transfer in heat exchanger tubes through passive methods, which have been found to cost less, are easier to implement, and are more reliable when compared to active and hybrid methods [5].

Marzouk et al. [6] reported on heat transfer enhancement techniques used in shell-and-tube heat exchangers, highlighting a major focus on passive strategies, followed by the moderate investigation of hybrid and active methods. Al-Chlahawi et al. [7] conducted an extensive review on techniques that use combinations of passive methods such as vortex generators, twisted tapes, surface roughness, and other turbulators simultaneously, and found that such combinations improve the Nusselt number (Nu); however, they also incur additional friction factor penalties. Ali Khan et al. [8] and Diwaker & Kumar [9] also conducted broad surveys and further classified passive techniques into surface modification, extended surfaces, inserts, and bulk fluid additives such as nanofluids. Both groups concluded that these passive methods enhance heat transfer by disrupting the boundary layer and promoting mixing; however, this comes with a corresponding increase in the pressure drop. These reviews raise an underlying design question: how is an increase in the Nusselt number coupled with a proportionate (if any) increase in the friction factor?

The literature on tube inserts clarifies this problem even further. Yildirim et al. [10] have used both experimental and numerical methods to investigate how spring-type and twisted-tape turbulators can enhance heat transfer in cylindrical pipes. After studying several pitch lengths (23 mm, 43 mm, and 63 mm for spring-type; 115, 215, and 315 mm for twisted tape), they obtained an optimum Nusselt number of 200 for the spring-type turbulator with a pitch length of 23 mm. They concluded that these turbulators enhance heat transfer by mixing the flow to ensure a more uniform temperature distribution within the bulk fluid. However, this heat transfer enhancement comes with pressure drop penalties: as the Nusselt number increased, so did the friction factor. In the optimal case, a 160% increase in the Nusselt number corresponded to a 700% increase in the friction factor. Lotfi et al. [11] performed a computational analysis of how four different vortex generators can enhance heat transfer in smooth wavy-fin and elliptical heat exchangers, concluding that the curved-angle rectangular winglets performed best at smaller angles of attack. Zhi-Jiang et al. [12] designed and studied the performance of a spirally corrugated heat exchanger tube. They concluded that increasing the pitch and corrugation depth decreased Nu , while increasing the corrugation depth increased heat transfer. Abotsi and Kizito [13] used a Kenics mixer with several twist ratios to improve heat transfer in an axially rotating pipe, achieving an optimal heat transfer enhancement of 47% at a rotation rate of 3. Sam-utpraphut et al. [14] found that twisted tapes with sawtooth edges doubled the Nusselt number in heat exchanger tubes when compared to plain twisted tapes; however, on average, these designs also tripled the friction factor when compared to the base design (plain twisted tapes). They also used the aerothermal performance index (API) to evaluate and determine an optimum value of 1.33 for their design. The author calculated this index as follows:

$$API = \frac{\left(\frac{Nu}{Nu_0}\right)}{\left(\frac{f}{f_0}\right)^{\frac{1}{3}}} \quad (1)$$

where Nu is the Nusselt number of the sawtooth-edge twisted tape, Nu_0 is the Nusselt number for a plain pipe without any inserts, and f and f_0 are the friction factors of their design and that of a plain pipe, respectively. Abdulhamed et al. [15] found that the use of relatively large holes in twisted tapes (for $4800 \leq Re \leq 9000$) can increase the Nu , f , and thermal enhancement factor (TEF) values, where the formula used was exactly that shown in Equation (1). Using a TiO_2 /water nanofluid with counter-swirl twisted tapes, Thianpong et al. [16] reported a peak thermal performance factor (TPF) near 1.5. Kadam et al. [17] used wire-coil inserts and concluded that a 110% improvement can be achieved in Nu and TPF values (between 1.2 and 1.6) for Re up to 10,000.

Internal fins integrated into heat exchanger tubes offer many advantages, including reduced installation time due to their shared topology with the tube itself, increased surface area, and anchored coherent secondary flows that continuously disrupt boundary layer structures. Pastuszko and Piasecka [18] examined the heat transfer characteristics of mini-fins, cavities, and perforated arrays of extensions of various heights on a surface undergoing pool boiling. They concluded that the best-performing modification was the introduction of mini-fins. El-Sebaey et al. [19] used both experimental and numerical methods to examine a longitudinal internal fin with a cross-section shaped like a hollow trapezoid, and reported an almost two-fold increase in the Nusselt number. Furthermore, the friction factor increased by only about 50% on average under variations in the controlling geometric parameters (i.e., fin angles, fin thicknesses, and inscribed diameter). Marzouk et al. [20] used extended pin fins in double-pipe heat exchangers and observed Nu improvements of 163–242% along with friction factor increases of 161–290%. Across these reports based on internal fins, parameters such as the number of fins, fin height, and cross-sectional geometry appear to govern the performance of the passive methods. We also observed that increasing the number of fins and fin height increases the friction factor, due to an increased contact surface area of the inner walls. Moreover, although most relevant works in the literature have focused on laminar to low- Re turbulent regimes, industrial applications in chemical plants, large-area heating, ventilation, and air conditioning systems [21], and similar systems are typically characterized by $Re \sim 10^5$.

In recent years, bio-inspired geometries have gained attention in this field based on the premise that surfaces inspired by nature may resolve or disproportionately increase the pressure drop traditionally associated with increased heat transfer. Hurry et al. [22] compared a rectangular fin, a NACA 0030 fin, and a bio-inspired shark denticle fin as an external surface texture in terms of their effects on heat transfer. Their observations revealed that the applied shark denticle fin yielded a mean thermohydraulic performance factor between 0.81 and 1.53 across a high-Reynolds-number range (roughly 9.8×10^4 to 22.9×10^4), with the denticles shifting the onset of turbulence and reducing the pressure drop at the lower end of the tested range. In a companion study [23] involving conjugate heat transfer simulation, they showed that a denticle-inspired plate fin achieved better heat removal performance than rectangular, cylindrical, and elliptical baselines. Wang et al. [24] optimized seagull-inspired curved vortex generators for compact exchangers and extended their study to phase-change and electronics cooling contexts. Li et al. [25] proposed wheat-grain-shaped pin fins for microchannel heat sinks; Liu et al. [26] introduced biomimetic spider web fins that accelerated melting in a latent heat storage application; and Yao et al. [27] optimized bionic intercooler fins using neural networks and genetic algorithms. Cabeza et al. [28] additively manufactured a thermal storage tank whose heat transfer fluid passages mimic the branched venation in plants and animals, increasing the

surface area and discharge rate. This latter example highlights a key trend—additive manufacturing enables the production of intricate internal geometries that would be impossible with conventional machining processes. In this regard, additively manufactured exchangers have already been demonstrated to achieve higher heat duty under a given set of constraints [29]. A key observation from this review of the bio-inspired literature is that, although biomimetic surfaces are increasingly being successfully used in practice, they have been applied mostly to external surfaces, plate fins, microchannels, and storage tanks, rather than to the internal fins of circular heat exchanger tubes.

Considering the current research trends, the literature reveals some gaps. The dominant unresolved problem is the persistent coupling between heat transfer enhancement and the pressure drop penalty; that is, across inserts, fins, and wall modifications, Nusselt number gains are repeatedly overshadowed by much larger friction factor gains. Moreover, the majority of these investigations are confined to laminar and low-to-moderate turbulent regimes (broadly, $Re < 10,000$), with many fin- and insert-based studies utilizing values between $Re \approx 2300$ and 7300 , even though numerous industrial flows (e.g., those in HVAC systems, chemical plants, and geothermal installations) operate at Reynolds numbers that are orders of magnitude higher. Furthermore, internal fin designs for heat transfer enhancement have generally relied on simple geometric cross-sections and have rarely exploited the intricate profiles found in nature. Although dimples, longitudinal fins, twisted tapes, wire coils, and other passive enhancement devices have been extensively investigated, as mentioned above, most previous studies have focused on either surface modifications or conventional inserts. Dimpled surfaces primarily enhance heat transfer through flow separation, recirculation, and boundary layer disruption, whereas conventional internal fins mainly exploit the increased heat transfer area and modification of the near-wall flow. In this regard, the present work makes a novel contribution by introducing a novel bio-inspired internal fin geometry that generates strong coherent vortical structures which continuously perturb the boundary layer to improve heat transfer. A claim of superiority with respect to other passive methods is not the main aim of our investigation; instead, we simply propose a competitive alternative.

This study addresses the above-mentioned gaps by numerically investigating and proposing a novel, bio-inspired internal fin with the following key features:

- It has a cross-section inspired by the horn of a ram sitting on a rectangular stem;
- The radii of the curves of the horn vary according to the Fibonacci spiral;
- It demonstrates thermohydraulic performance above unity for high Re .

The fins are studied as an integral feature of a circular tube over the high-Reynolds-number range of $10,000$ – $100,000$ —conditions representative of HVAC, chemical processing, and geothermal flows that the existing literature has largely left unexplored [30]. The thermohydraulic performance of this design is assessed and compared to that of a plain tube, with parametric studies conducted to determine the effects of the fin number (N) and fin height (characterized by the s/D ratio, where s is the height of the stem from the inner wall of the pipe to the base of the intersection of the two ‘horns’, and D is the inner diameter of the pipe) on the heat transfer, pressure drop, and heat retention rate of the exchanger.

The remainder of this paper is organized as follows: Section 2 describes the geometry, mesh, boundary conditions, and overall numerical methodology. Section 3 presents and discusses the grid independence study, the validation of plain tube simulations against established correlations, and the effects of the geometric parameters on thermohydraulic performance and heat retention. Section 4 draws the principal conclusions from the work.

2. Modeling and Numerical Methodology

Figure 1 shows a flowchart of the steps followed in this numerical investigation. It began with the base-case design; namely, a plain pipe without internal fins. This pipe was designed as a copper pipe with an inner diameter of 25 mm, outer diameter of 26.5 mm, and length of 750 mm. This length was chosen to ensure that a fully developed flow would be obtained (i.e., $\frac{L}{D} > 10$). The geometry was designed in SOLIDWORKS 2024 and transferred to ANSYS FLUENT 2026 for pre- and post-processing.

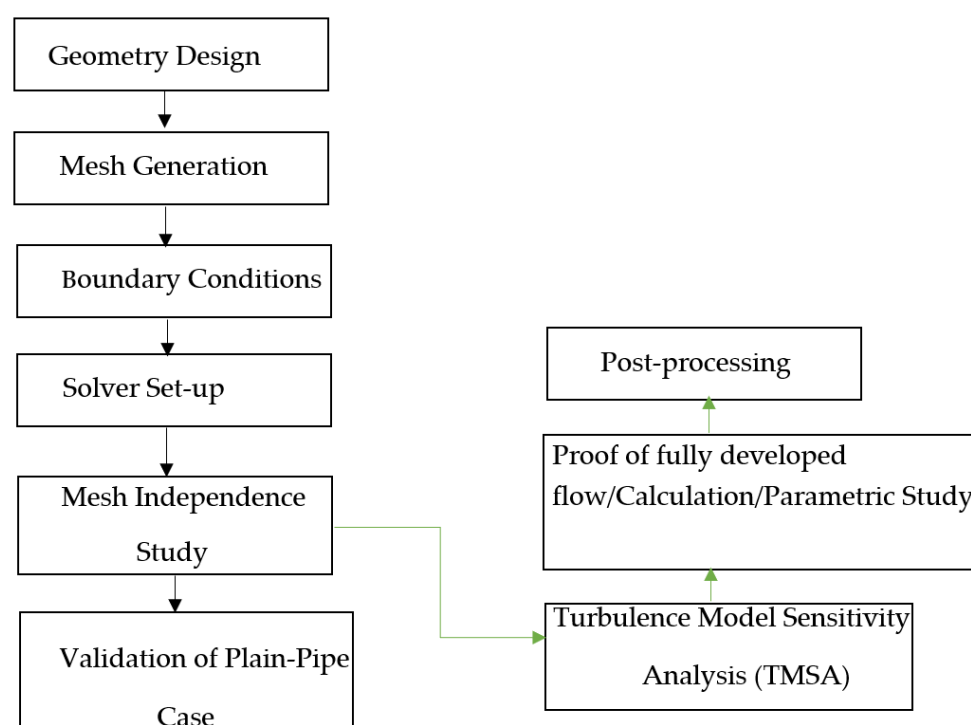


Figure 1. Flow chart of numerical investigation.

The path indicated by black arrows was followed for both the plain and finned pipe cases. However, as experimental data for the novel finned pipe cases were not available, we followed additional steps (indicated by the green arrows) to ensure the reproducibility and numerical rigor of the analyses.

2.1. Geometry

Figure 2 shows the cross-section of the internal fin, which was naturally inspired—in particular, by the horns of a ram. Figure 2b was generated using ChatGPT-5 with a single sentence prompt as follows: “Generate a high-quality, front view image of the head of a ram with curved horns and a white background”. As shown in the figure, the curvature of the horn is characterized by a variable radius as it branches outward; in particular, the curvature variation in this fin geometry was designed such that the radii of curvature followed the Fibonacci spiral, where each radius is the sum of the previous two. This progression is shown in Figure 3, which details the fin more elaborately. R_1 , R_2 , R_3 , and R_4 are related to each other according to Equations (2)–(4).



Figure 2. (a) Cross-section of the internal fin, inspired by (b) a ram's horns.

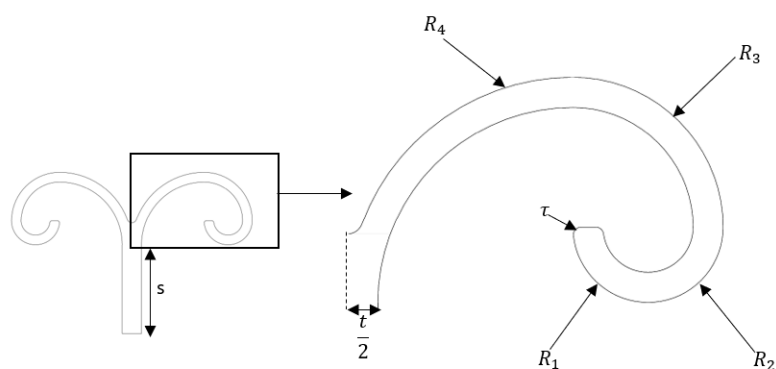


Figure 3. Elaborated detail of the cross-section of the internal fin structure.

$$R_1 = R_2 \quad (2)$$

$$R_3 = R_1 + R_2 \quad (3)$$

$$R_4 = R_2 + R_3 \quad (4)$$

Table 1 provides a summary of the parameters of the ram-horn-inspired fin.

Table 1. Geometric parameters of the proposed finned pipe design.

Parameter	Description	Value(s)
D	Pipe inner diameter	25 mm
N	Number of fins	4, 6, 8
t	Fin stem thickness	0.4 mm
τ	Fin tip fillet radius	0.05 mm
s/D	Stem height-to-diameter ratio	0.06, 0.08, 0.1 (s = 1.5 mm, 2 mm, 2.5 mm)
R ₁	Horn radius (innermost branch)	0.5 mm
R ₂	Horn radius (next branch)	0.5 mm
R ₃	Horn radius (next branch)	1 mm
R ₄	Horn radius (largest branch)	1.5 mm

For this study, three values of N (4, 6, 8) and three values of s/D (0.06, 0.08, 0.1) were used to assess the effects of the number of fins and fin height on the thermohydraulic performance of the design. In the study focused on the fin number, N was varied at constant s/D ($=0.06$), while for the effects of fin height, s/D was varied at constant N ($=8$). Figure 4 shows the geometry of the finned pipe for the design point $N = 8$, and Figure 5 shows an oblique view of the entire 3D geometry, showing how the developed cross-section of the fin was extruded longitudinally and patterned azimuthally within the pipe.

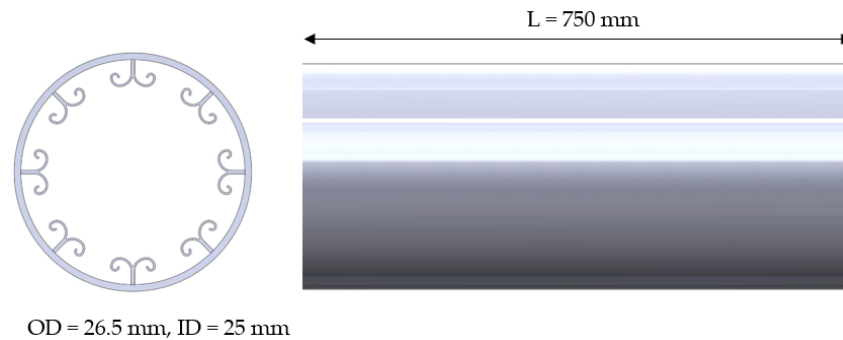


Figure 4. Geometric model of the pipe: left (axial view), right (side view).

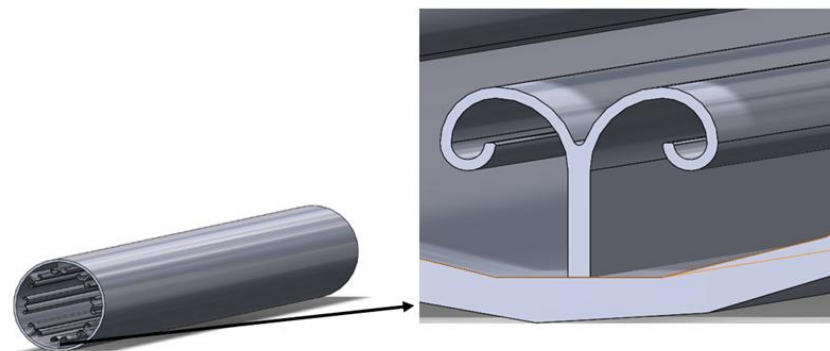


Figure 5. Oblique view of the geometric model of the tube (elaborated picture on the right to show the internal fin structure).

This SOLIDWORKS model was exported as a geometry file to be meshed in ANSYS Fluent Meshing.

Prototype

Manufacturability is an important consideration in the design of internally finned tubes. Thus, in order to prove that this novel design is manufacturable, a 100 mm-long prototype was metal 3D-printed using laser powder bed fusion with GRCo-42 alloy (a copper alloy). The machine used for this process was the EOS M290 with a 1×400 W Yb-fiber laser and a 32 A/400 V power supply. This machine was manufactured by Electro Optical Systems, a German industrial additive manufacturing company headquartered in Krailing, Germany. Figure 6 shows the 3D-printed prototype.

The manufactured part provides evidence that modern-day manufacturing methods can be used to produce the proposed design.

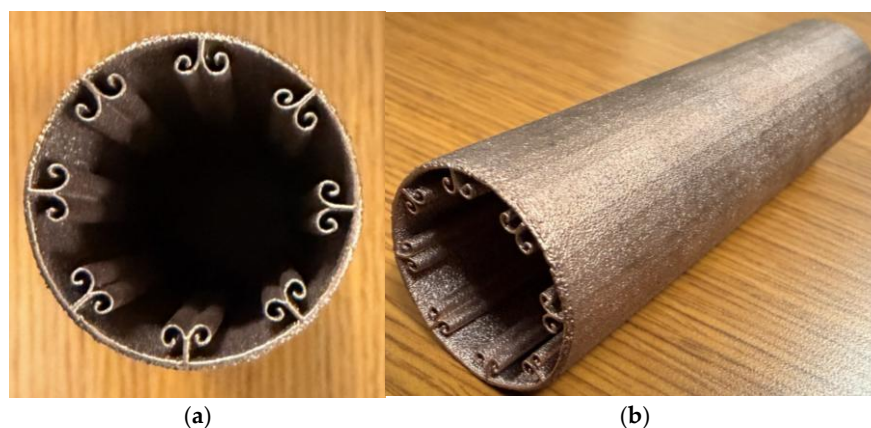


Figure 6. Manufactured prototype ($N = 8$, $s/D = 0.06$) using laser powder bed fusion: (a) Top view; (b) oblique view.

2.2. Meshing

Figure 7 shows the details of the mesh configuration near the most complex design point ($N = 8$, $s/D = 0.1$).

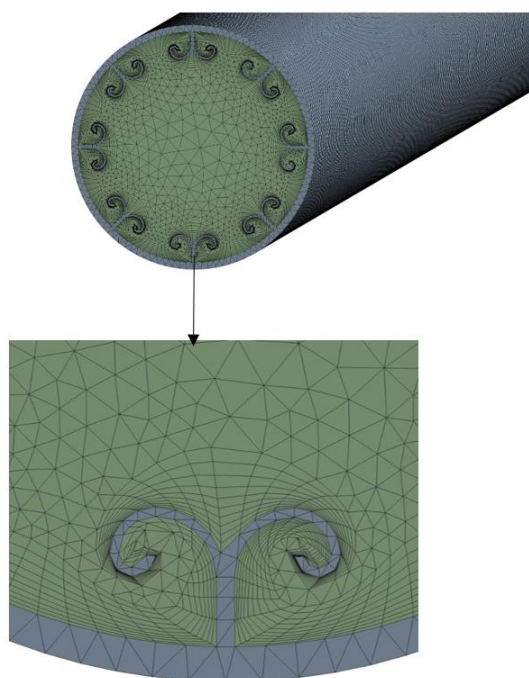


Figure 7. Mesh details ($N = 8$, $s/D = 0.1$).

Both the solid and fluid domains were meshed independently using different settings. For the solid domain, an automatic mesh was used with tetrahedral cells; inflation layers were not meshed within this domain because there are no boundary layer flows within it to capture. For the fluid domain, the choice of the first-layer thickness of the mesh was carefully made to ensure that an appropriate y^+ value was chosen for the cells, in order to accurately capture flow within the viscous sublayer. In particular, it is highly recommended that the y^+ value remains within the range of $1 \leq y^+ \leq 5$ for the turbulence model to precisely predict heat transfer parameters and flow fields (e.g., velocity and pressure). With a value of $80 \mu\text{m}$ for the first-layer thickness and a growth rate of 1.15, 11 layers were used for the mesh. The most complex design point ($N = 8$, $s/D = 0.1$, $\text{Re} = 100,000$) showed a y^+ distribution within the range of 1.9–3.5 with an average y^+ value of 2.6 for

the grid-independent solution. Thus, all cases followed the same setting for first-layer thickness, number of layers, and growth rate, and the average y^+ value for each case ranged between 0.8 and 2.6, indicating that the first cell was within the viscous sublayer. Mesh quality metrics such as orthogonal quality and skewness were checked for all cases. The minimum orthogonal quality was 0.25, and the maximum skewness was 0.75, which was obtained for a minority (15–20%) of the cells, indicating the meshes used for these cases were of good quality.

2.3. Boundary Conditions, Solution Procedure, and Mesh Independence Study

The base design consisted of a hollow cylindrical solid domain with copper as the chosen material, having constant properties and a fluid filling its cavity. Water was used as the working fluid, with temperature-dependent properties within an anticipated temperature range of 350–400 K and a pressure of 4 bar. The maximum temperature observed across all calculations occurred at the wall in the plain pipe case, with a value of 392 K at $Re = 10,000$; although ANSYS Fluent extrapolated the range of validity to 500 K, as the maximum observed temperature was well below this set upper range, the property calculations remain accurate. This thermodynamic state was selected to ensure single-phase flow throughout the steady-state simulation. The justification for the choice of constant solid properties is rooted in the negligible percentage changes (0.8–2%) of properties within a similar temperature range. Table 2 details the properties of the fluid and solid used in the simulations. The boundary conditions selected for the setup were as follows:

1. At the inlet, the temperature of the flow was set to 350 K with a specific mass flow rate based on the chosen Reynolds number ($10,000 \leq Re \leq 100,000$). The mass flow rate was expressed in terms of the Reynolds number and the nominal hydraulic diameter of the plain pipe, D , as follows:

$$\dot{m} = \frac{Re_D \pi D \mu}{4} \quad (5)$$

2. The Reynolds number was based on the diameter of the plain pipe to ensure that the same mass flow rates are compared between plain and internally finned pipe, enabling a fair industry-level operational comparison.
3. At the outlet, a target mass flow rate equal to the inlet mass flow rate was selected to ensure that continuity was strictly obeyed after convergence, and the absolute outlet pressure was set to 4 bar. The turbulence boundary conditions were set at an Re -dependent turbulent intensity value ($0.016 Re^{-1/8}$), and 7% of the hydraulic diameter was used for the turbulent length scale. Reynolds numbers were determined based on the plain tube's diameter, such that all geometries were compared at identical mass flow rates; the true hydraulic diameter of the finned cross-section was used only for specification of the turbulent length scale at the inlet, which was calculated using the following equation:

$$D_h = \frac{4V}{A_s} \quad (6)$$

where V is the volume of the fluid domain and A_s is the wetted surface area of the inner wall (including the fins). Both values were obtained as direct information, calculated in ANSYS Meshing when the inner wall boundary was selected for A_s and the fluid 'body' was selected for V .

4. At the outer wall, a constant heat flux of $50,000 \text{ W/m}^2$ was applied uniformly.
5. At the inner wall, a coupled boundary condition was selected to ensure conjugate heat transfer, where the heat increases the temperature of the fluid while the fluid simultaneously removes heat away from the wall via internal convection.

6. At the wall perpendicular to the inner and outer walls, an insulated boundary condition was selected. This choice was made based on the assumption that these surfaces are small and their contributions to the heat transfer process may be negligible.

Table 2. Material properties for solid (copper) and fluid (water).

Property	Water at 350 K, 4 Bar	Copper at 350 K
Thermal conductivity (W/mK)	$1.037 \times 10^{-8} T^3 - 1.828 \times 10^{-5} T^2 + 9.6099 \times 10^{-3} T - 0.91629$	401
Specific heat capacity (J/kgK)	$-1.48 \times 10^{-6} T^3 + 1.1583 \times 10^{-2} T^2 - 6.8807 T + 5246.72$	386
Dynamic viscosity (kg/ms)	$-3.341 \times 10^{-10} T^3 + 4.0056 \times 10^{-7} T^2 - 1.62343 \times 10^{-4} T + 2.2444 \times 10^{-2}$	-
Density (kg/m ³)	$3.852 \times 10^{-6} T^3 - 6.596 \times 10^{-3} T^2 + 2.5944 T + 708.7145$	8930

Over the fluid domain, ANSYS Fluent solved the governing equations of the conservation of mass (Equation (7)); the radial (r), azimuthal (θ), and axial (z) momenta (Navier–Stokes) (Equations (8)–(10)); and energy (Equation (11)) in their dimensional forms. The equations are as follows:

$$\frac{1}{r} \frac{\partial}{\partial r} (r \rho \bar{u}_r) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho \bar{u}_\theta) + \frac{\partial}{\partial z} (\rho \bar{u}_z) = 0 \quad (7)$$

$$\begin{aligned} \frac{1}{r} \frac{\partial}{\partial r} (r \rho \bar{u}_r \bar{u}_r) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho \bar{u}_\theta \bar{u}_r) + \frac{\partial}{\partial z} (\rho \bar{u}_z \bar{u}_r) - \frac{\rho \bar{u}_\theta^2}{r} \\ = -\frac{\partial \bar{p}}{\partial r} + \frac{1}{r} \frac{\partial}{\partial r} (r \tau_{rr}) + \frac{1}{r} \frac{\partial \tau_{r\theta}}{\partial \theta} + \frac{\partial \tau_{rz}}{\partial z} - \frac{\tau_{\theta\theta}}{r} + \frac{1}{r} \frac{\partial}{\partial r} (-r \rho \bar{u}_r' \bar{u}_r') \\ + \frac{1}{r} \frac{\partial}{\partial \theta} (-\rho \bar{u}_\theta' \bar{u}_r') + \frac{\partial}{\partial z} (-\rho \bar{u}_z' \bar{u}_r') + \frac{\rho \bar{u}_\theta' \bar{u}_\theta'}{r} \end{aligned} \quad (8)$$

$$\begin{aligned} \frac{1}{r} \frac{\partial}{\partial r} (r \rho \bar{u}_r \bar{u}_\theta) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho \bar{u}_\theta \bar{u}_\theta) + \frac{\partial}{\partial z} (\rho \bar{u}_z \bar{u}_\theta) - \frac{\rho \bar{u}_r \bar{u}_\theta}{r} \\ = -\frac{1}{r} \frac{\partial \bar{p}}{\partial \theta} + \frac{1}{r^2} \frac{\partial}{\partial r} (r^2 \tau_{r\theta}) + \frac{1}{r} \frac{\partial \tau_{\theta\theta}}{\partial \theta} + \frac{\partial \tau_{\theta z}}{\partial z} + \frac{1}{r^2} \frac{\partial}{\partial r} (-r^2 \rho \bar{u}_r' \bar{u}_\theta') \\ + \frac{1}{r} \frac{\partial}{\partial \theta} (-\rho \bar{u}_\theta' \bar{u}_\theta') + \frac{\partial}{\partial z} (-\rho \bar{u}_z' \bar{u}_\theta') \end{aligned} \quad (9)$$

$$\begin{aligned} \frac{1}{r} \frac{\partial}{\partial r} (r \rho \bar{u}_r \bar{u}_z) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho \bar{u}_\theta \bar{u}_z) + \frac{\partial}{\partial z} (\rho \bar{u}_z \bar{u}_z) \\ = -\frac{\partial \bar{p}}{\partial z} + \frac{1}{r} \frac{\partial}{\partial r} (r \tau_{rz}) + \frac{1}{r} \frac{\partial \tau_{\theta z}}{\partial \theta} + \frac{\partial \tau_{zz}}{\partial z} + \frac{1}{r} \frac{\partial}{\partial r} (-\rho \bar{u}_r' \bar{u}_z') \\ + \frac{1}{r} \frac{\partial}{\partial \theta} (-\rho \bar{u}_\theta' \bar{u}_z') + \frac{\partial}{\partial z} (-\rho \bar{u}_z' \bar{u}_z') \end{aligned} \quad (10)$$

$$\begin{aligned} \frac{1}{r} \frac{\partial}{\partial r} [r \bar{u}_r (\rho E + \bar{p})] + \frac{1}{r} \frac{\partial}{\partial \theta} [\bar{u}_\theta (\rho E + \bar{p})] + \frac{\partial}{\partial z} [\bar{u}_z (\rho E + \bar{p})] \\ = \frac{1}{r} \frac{\partial}{\partial r} \left(r k_{eff} \frac{\partial T}{\partial r} \right) + \frac{1}{r} \frac{\partial}{\partial \theta} \left(\frac{1}{r} k_{eff} \frac{\partial T}{\partial \theta} \right) + \frac{\partial}{\partial z} \left(k_{eff} \frac{\partial T}{\partial z} \right) \\ + \frac{1}{r} \frac{\partial}{\partial r} [r (\bar{\mathbf{u}} \cdot \tau_{eff})_r] + \frac{1}{r} \frac{\partial}{\partial \theta} [(\bar{\mathbf{u}} \cdot \tau_{eff})_\theta] + \frac{\partial}{\partial z} [(\bar{\mathbf{u}} \cdot \tau_{eff})_z] \end{aligned} \quad (11)$$

In these set of equations ρ is the density, $\bar{\mathbf{u}}$ represents the average velocity vector, $\bar{u}_r$ is the average radial velocity, $\bar{u}_\theta$ is the average azimuthal velocity, $\bar{u}_z$ is the average axial velocity, τ represents the shear stress, $\bar{p}$ represent the average pressure, E is the internal energy, T is the temperature, and k_{eff} is the effective thermal conductivity. The above set of equations represents the Reynolds-Averaged Navier–Stokes (RANS) conservation equations. Each flow variable is decomposed into a time-averaged part (variables with overbars) and a fluctuating part (variables with primes). For an incompressible

steady flow, the mass conservation equation reduces from the time-varying form to Equation (7), which enables a solution for the mean velocity field to be obtained across the fluid domain. In Equations (8)–(10), the left-hand side of the equation represents the mean convection of momentum, whereas the right-hand side comprises the pressure gradient, viscous stresses that arise from the averaging operation, and other terms due to fluctuations. The fluctuation terms, which are differentiated with respect to the spatial coordinates (i.e., $-\rho \overline{u'_i u'_j}$) form the Reynolds stress tensor, which represents the mean transport of momentum by turbulent fluctuations and is a mathematical consequence of averaging the non-linear convective term. This quantity, however, cannot be evaluated directly from the mean field as it contains fluctuating velocity terms. Fluent solves this issue through the use of the Boussinesq Eddy viscosity hypothesis, which posits that the Reynolds stress is proportional to the mean strain rate through a turbulent (eddy) viscosity. This viscosity is estimated via a turbulence model and, as such, a turbulence model sensitivity analysis was performed for the finned pipe in order to determine the best model to use. The plain pipe Nusselt number and friction factor results are later validated against existing correlation equations in the literature, and the accuracy of these results is a consequence of choosing the $k-\omega$ SST turbulence model due to its track record in properly modeling wall functions and preventing excessive diffusion of the turbulent viscosity. As experimental data are not yet available for the proposed fin design, turbulent model sensitivity analysis was performed for three separate turbulent models—namely, the Spalart–Allmaras model, the realizable $k-\varepsilon$ model with enhanced wall treatment functions, and the $k-\omega$ SST model—in order to ensure both numerical rigor and reproducibility of the results. The turbulence model sensitivity analysis results are presented later in this paper.

The Spalart–Allmaras turbulence model [31] solves a single transport equation for a modified turbulent viscosity $\tilde{\nu}$, and was designed for use in applications involving aerodynamic wall-bounded flows. It is computationally inexpensive and easy to implement, but has weaknesses in predicting jets, free shear flows, and decaying turbulence. The equation of this model is given as follows:

$$\begin{aligned}
 u_r \frac{\partial \tilde{\nu}}{\partial r} + \frac{u_\theta}{r} \frac{\partial \tilde{\nu}}{\partial \theta} + u_z \frac{\partial \tilde{\nu}}{\partial z} &= C_{b1}(1 - f_{t2})\tilde{S}\tilde{\nu} \\
 &+ \frac{1}{\sigma} \left\{ \frac{1}{r} \frac{\partial}{\partial r} \left[r(\nu + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial r} \right] + \frac{1}{r} \frac{\partial}{\partial \theta} \left[\frac{1}{r} (\nu + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial \theta} \right] \right. \\
 &\left. + \frac{\partial}{\partial z} \left[(\nu + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial z} \right] + C_{b2} |\nabla \tilde{\nu}|^2 \right\} - \left(C_{w1} f_w - \frac{C_{b1}}{\kappa^2} f_{t2} \right) \left(\frac{\tilde{\nu}}{d} \right)^2
 \end{aligned} \quad (12)$$

where u_r is the mean radial velocity, $\tilde{\nu}$ is the modified turbulent viscosity, C_{b1} is the production coefficient, f_{t2} is the trip function, $\tilde{S}$ is the modified vorticity magnitude, σ is the turbulent Prandtl number based on $\tilde{\nu}$, C_{b2} is the diffusion coefficient, C_{w1} is the destruction coefficient, f_w is the wall destruction function, κ is the Von Karman constant, d is the distance from the local point to the nearest solid wall, and the constants pertinent to the equation are listed in Table 3.

Table 3. Constants used in the Spalart–Allmaras model.

Constant	Value
σ	0.667
C_{b1}	0.136
C_{b2}	0.622
C_{w1}	$\frac{C_{b1}}{\kappa^2} + \frac{1 + C_{b2}}{\sigma}$
κ	0.410

The realizable $k-\varepsilon$ model [32] is a two-equation model that solves for the turbulent kinetic energy (k) and energy dissipation rate (ε) with additional modifications to the standard $k-\varepsilon$ model to satisfy realizability constraints (i.e., positive normal Reynolds stresses and Schwarz inequality). It predicts the spreading rates of planar and round jets well but has weaknesses in adverse pressure gradient applications. It is usually used for internal flows, mixing layer flows, and swirling flows. The equations for turbulent kinetic energy (Equation (13)) and dissipation rate (Equation (14)) used in this model are given as

$$\begin{aligned} \frac{1}{r} \frac{\partial}{\partial r} (r \rho u_r k) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho u_\theta k) + \frac{\partial}{\partial z} (\rho u_z k) \\ = \frac{1}{r} \frac{\partial}{\partial r} \left[r \left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial r} \right] + \frac{1}{r} \frac{\partial}{\partial \theta} \left[\frac{1}{r} \left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial \theta} \right] \\ + \frac{\partial}{\partial z} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial z} \right] + P_k - \rho \varepsilon \end{aligned} \quad (13)$$

$$\begin{aligned} \frac{1}{r} \frac{\partial}{\partial r} (r \rho u_r \varepsilon) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho u_\theta \varepsilon) + \frac{\partial}{\partial z} (\rho u_z \varepsilon) \\ = \frac{1}{r} \frac{\partial}{\partial r} \left[r \left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial r} \right] + \frac{1}{r} \frac{\partial}{\partial \theta} \left[\frac{1}{r} \left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial \theta} \right] + \frac{\partial}{\partial z} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial z} \right] \\ + \rho C_{1\varepsilon} S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} \end{aligned} \quad (14)$$

where k is the turbulent kinetic energy (TKE); μ is the molecular dynamic viscosity; μ_t is the turbulent viscosity; σ_k and σ_ε are the turbulent Prandtl numbers for k and ε , respectively; P_k is the production of TKE; ρ is the density; $C_{1\varepsilon}$ is the production coefficient for ε ; C_2 is the destruction coefficient; S is the magnitude of the mean strain rate tensor; and the constants utilized in the equations are listed in Table 4.

Table 4. Constants used in the realizable $k-\varepsilon$ model.

Constant	Value
σ_k	1.000
σ_ε	1.200
$C_{1\varepsilon}$	1.44
C_2	1.900

The $k-\omega$ SST model [33] blends the robust near-wall behavior of the $k-\omega$ model with the freestream independence of the $k-\varepsilon$ model. A shear-stress transport limiter (SST) is added to improve predictions involving adverse pressure gradient boundary layers. This model is best known for its excellent near-wall resolution without wall functions; however, it is less accurate for free shear flows and jets when compared with some $k-\varepsilon$ variants. The turbulent kinetic energy (k) and dissipation rate (ω) transport equations for this model are as follows:

$$\begin{aligned} \frac{1}{r} \frac{\partial}{\partial r} (r \rho u_r k) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho u_\theta k) + \frac{\partial}{\partial z} (\rho u_z k) \\ = \frac{1}{r} \frac{\partial}{\partial r} \left[r (\mu + \sigma_k \mu_t) \frac{\partial k}{\partial r} \right] + \frac{1}{r} \frac{\partial}{\partial \theta} \left[\frac{1}{r} (\mu + \sigma_k \mu_t) \frac{\partial k}{\partial \theta} \right] \\ + \frac{\partial}{\partial z} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial z} \right] + P_k - \beta^* \rho k \omega \end{aligned} \quad (15)$$

$$\begin{aligned} \frac{1}{r} \frac{\partial}{\partial r} (r \rho u_r \omega) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho u_\theta \omega) + \frac{\partial}{\partial z} (\rho u_z \omega) \\ = \frac{1}{r} \frac{\partial}{\partial r} \left[r (\mu + \sigma_{\omega 1} \mu_t) \frac{\partial \omega}{\partial r} \right] + \frac{1}{r} \frac{\partial}{\partial \theta} \left[\frac{1}{r} (\mu + \sigma_{\omega 1} \mu_t) \frac{\partial \omega}{\partial \theta} \right] \\ + \frac{\partial}{\partial z} \left[(\mu + \sigma_{\omega 1} \mu_t) \frac{\partial \omega}{\partial z} \right] \\ + 2(1 - F_1) \rho \sigma_{\omega 2} \frac{1}{\omega} \left(\frac{\partial k}{\partial r} \frac{\partial \omega}{\partial r} + \frac{1}{r^2} \frac{\partial k}{\partial \theta} \frac{\partial \omega}{\partial \theta} + \frac{\partial k}{\partial z} \frac{\partial \omega}{\partial z} \right) + \frac{\gamma}{\nu_t} P_k - \beta \rho \omega^2 \end{aligned} \quad (16)$$

where β^* is the destruction coefficient for k ; β is the destruction coefficient for ω ; F_1 is the first blending function; γ is the blended production coefficient for ω ; $\sigma_{\omega 1}$ and $\sigma_{\omega 2}$ are the inner and outer turbulent Prandtl numbers for k and ω , respectively; and the relevant constants are listed in Table 5.

Table 5. Constants used in the realizable $k-\omega$ SST model.

Constant	Value
σ_k	0.850
$\sigma_{\omega 1}$	0.500
β	0.075
β^*	0.090
γ	0.44
$\sigma_{\omega 2}$	0.856

This set of equations, together with the appropriate boundary conditions, ensures that the pressure and velocity fields are calculated throughout the fluid domain. Equation (11) also enables calculation of the temperature field. Table 6 details how the numerical solution process and controls were set up in Fluent.

Table 6. CFD setup in ANSYS Fluent 2026 R1.

Setup	Selection
Solver	Pressure-based
Flow type	Steady
Energy equation	Enabled
Turbulence model	SST $k-\omega$, $k-\varepsilon$, and Spalart–Allmaras
Pressure–velocity coupling	SIMPLE
Momentum discretization	Second-order upwind
Energy discretization	Second-order upwind
Convergence criteria	1×10^{-5} for continuity, 1×10^{-6} for all other variables
y^+ setup	First-layer thickness for the most complex case ($N = 8$, $s/D = 0.1$, $Re = 100,000$) $\sim 80 \mu\text{m}$ Inflation layers at boundary = 11 Growth rate = 1.15 y^+ range = 0.8–2.6 for all cases

The 50,000 W/m² heat flux supplied at the outer wall is conducted through the walls and fins, and convection occurs via bulk fluid flow. After calculating the pressure, velocity, and temperature fields, we used different metrics to quantify the thermohydraulic performance and heat retention improvement of the finned pipe, as outlined in Table 4. All calculated properties with subscripts 0 represent those of the plain pipe ($N = 0$ baseline case). The Nusselt number, Nu , was calculated as follows:

$$Nu = \frac{hD}{k} \quad (17)$$

where h is the heat transfer coefficient, which is calculated from Newton's law of cooling as follows:

$$h = \frac{q''}{T_w - T_{avg}} \quad (18)$$

where q'' is the heat flux (set at 50,000 W/m²), T_w is the inner wall temperature, T_{avg} is the average of the inlet and the bulk fluid temperatures, D is the diameter of the pipe, and k is the thermal conductivity of the fluid. The friction factor f was calculated using the pressure drop form of the friction factor equation, given as

$$f = \frac{2\Delta PD}{\rho V^2 L} \quad (19)$$

Here, V is the fully developed mean flow velocity of the bulk fluid, L is the fully developed length, and ΔP is the pressure drop across this length. For the heat retention ratio, which measures the heat absorption rate, $\dot{Q}$ was calculated as

$$\dot{Q} = \dot{m}c_p(T_{out} - T_{in}) \quad (20)$$

For each performance metric in Table 7, a value of 1 means that there was no enhancement compared with the plain pipe case. Conversely, a value greater than 1 indicates an increase, while a value less than 1 indicates a decrease. For example, a Nusselt number ratio greater than 1 indicates enhanced heat transfer, while a friction factor ratio greater than 1 indicates an increased pressure drop penalty.

Table 7. Thermohydraulic and heat retention performance metrics.

Metric	Expression
Nusselt number ratio	$\frac{Nu}{Nu_0}$
Friction factor ratio	$\frac{f}{f_0}$
Thermohydraulic performance factor	$\frac{\left(\frac{Nu}{Nu_0}\right)}{\left(\frac{f}{f_0}\right)^{\frac{1}{3}}}$
Heat retention ratio	$\frac{\dot{Q}}{\dot{Q}_0}$

2.4. Mesh Independence Study

For mesh independence, we analyzed the most complex case to show that the chosen mesh-independent solution provides a mesh count that enables the accurate prediction of flow parameters across cases with less turbulence due to lower geometric complexity and lower Re . The mesh used tetrahedral cells. Table 8 provides the results of a mesh independence study for the most complex design point ($N = 8$, $s/D = 0.1$, $Re = 100,000$), including three mesh counts. Due to the unknown fluid dynamics prior to the numerical investigation, we considered two metrics to determine the most complex case. One was the volume ratio (i.e., the ratio of the plain pipe's fluid domain volume to that of the finned pipe's fluid domain), which we envisioned to require a high mesh density to properly capture the flow mechanisms. The second metric was the level of turbulence. Since the 8-fin configuration with the tallest fins would have the most reduced volume, resulting in the highest volume ratio, and as $Re = 100,000$ was the most turbulent value considered in this investigation, the case ($N = 8$, $s/D = 0.1$, $Re = 100,000$) was chosen as the most hydrodynamically complex case.

Table 8. Mesh independence study for the proposed finned pipe ($N = 8$, $s/D = 0.1$, $Re = 100,000$).

Mesh Count	Nu	f	% Change in Nu	% Change in f
16,906,036	658	0.0682	-	-
24,293,285	657	0.0677	0.15%	0.73%
48,921,194	668	0.0666	1.67%	1.62%

As the number of cells increased, the changes in Nu and f were not as significant. The initial mesh count resulted in the worst combination of values, due to the complex nature of the fin geometry and aspect ratio of the entire domain. After reaching a mesh count of 24,293,285 cells, a further increase in the mesh number yielded an increment of less than

2% in both Nu and f. Thus, this second cell count was used for all other finned pipe cases, as it was the mesh-independent solution. However, using 24+ million cells for the plain pipe case was considered overkill; as such, we conducted a separate mesh independence study and found that only 592,688 cells were needed for a mesh-independent solution (which was later validated); Table 9 shows the results of this study.

Table 9. Mesh independence study for the plain pipe case (Re = 100,000).

Mesh Count	Nu	f	% Change in Nu	% Change in f
192,745	303	0.0177	-	-
592,688	315	0.0156	3.96%	11.86%
2,715,851	293	0.0158	6.98%	1.28%

Although the drop in Nu from 315 to 293 is approximately a 7% change, the coarser mesh was selected as the mesh-independent solution for two reasons. The first is that, when the cell count increased by a factor of 4 (an increase exceeding 300%), the change in the Nusselt number was not very significant (7%). Furthermore, the finer mesh data point was slightly further from the correlation results than that of the coarser mesh. The correlation analysis yielded a value of 316 for Nu, which was closer to that obtained with the coarser mesh (315) than the finer mesh (293). Thus, the coarser mesh was chosen as the mesh-independent solution, in order to balance between accuracy and computational cost.

2.5. Validation of Plain Pipe Case

To ensure that the numerical investigation accurately predicted heat transfer and hydraulic parameters, it was validated against well-established experimental correlations from the literature, as plotted in Figure 8a,b. In particular, the friction factor calculated from pressure drop values in the simulations was compared to the Petukhov correlation [34] given in Equation (21), and the calculated Nusselt numbers were compared to the Dittus–Boelter correlation [35] in Equation (22). Prandtl number calculations were made at film temperature.

$$f = (0.79 \ln Re - 1.64)^{-2} \quad (21)$$

$$Nu_D = 0.023 Re_D^{0.8} Pr^{0.4} \quad (22)$$

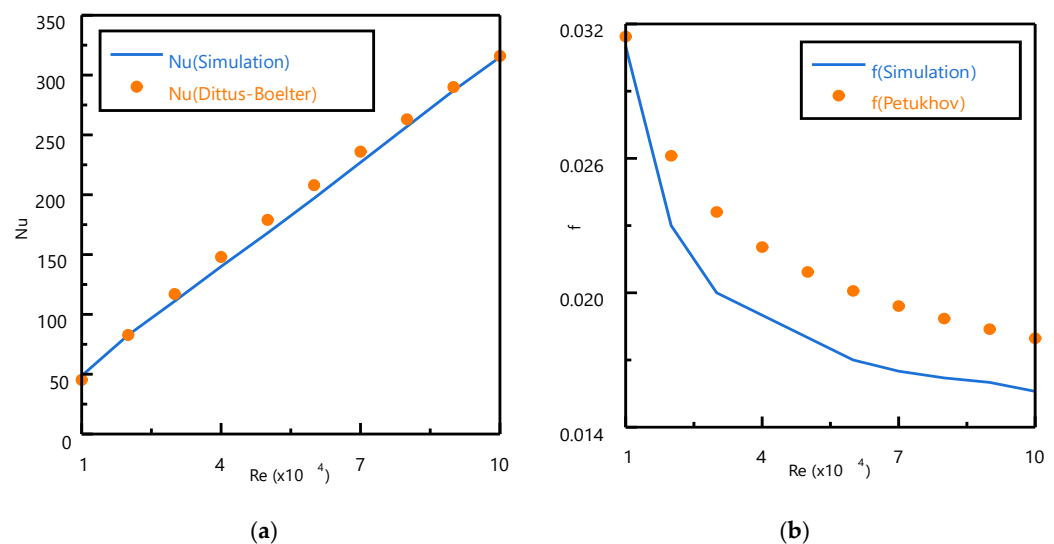


Figure 8. Validation of simulation results (plain pipe): (a) Nusselt number vs. Re; and (b) friction factor vs. Re.

The graphs above indicate good agreement between the simulation and correlations, with the CFD results well within 13.02% of the Petukhov correlation (Equation (21)), while the Nusselt number calculated from wall temperature, fluid temperature, and heat flux data from the simulation was within 3.7% of the Dittus–Boelter correlation, thus validating the modeling approach used. This error was calculated using the following equation:

$$\% \text{ Error} = \frac{1}{M} \sum_{i=1}^{10} \frac{|\phi_{\text{simulation}} - \phi_{\text{correlation}}|}{\phi_{\text{correlation}}} \times 100\% \quad (23)$$

where M is the number of data points (10), $\phi_{\text{simulation}}$ is either Nu or f (calculated from the simulation data), and $\phi_{\text{correlation}}$ is either Nu or f (calculated from the relevant correlation equation). The discrepancy in the friction factor can be attributed to the location chosen for fully developed flow (mid-section), from which the pressure gradient was calculated; these changes do not significantly affect the solution.

2.6. Turbulence Model Sensitivity Analysis

To ensure that the turbulence model accurately captures the expected flow structure, a turbulence model sensitivity analysis was performed for the most complex ($N = 8$, $s/D = 0.1$, $Re = 100,000$; as justified in Section 2.4) finned pipe case. This sensitivity analysis ensured the reliability and reproducibility of the results. Figure 9 shows the transverse contours of turbulent viscosity ratio (TVR), which was chosen as a metric to rank the three models tested, i.e., the Spalart–Allmaras, $k-\epsilon$, and $k-\omega$ SST models. A good turbulence model is expected to have very low (even close to zero) values of TVR at the walls while high values are expected at the vortex core regions (which, in this case, are the regions close to the horns of the fins), and it is expected to capture an asymmetry that physically manifests as the anisotropy present in Reynolds stresses. Islands of high-TVR regions are ‘red flags’ for a model, as they indicate overproduction of turbulence. Additionally, a highly symmetric transverse contour is not a good sign either, as it indicates diffusiveness. Figure 9 shows the transverse contours of TVR for the three considered models. In particular, Figure 9a represents the transverse contour of TVR for the $k-\epsilon$ model at the location $x/L = 1/2$, where x is the axial distance from the inlet of the finned pipe; Figure 9b shows that for the Spalart–Allmaras model; and Figure 9c shows that for the $k-\omega$ SST model.

The contours above indicate key features used to justify the model choice for the numerical investigation. First, Figure 9a shows ‘islands’ of high TVR at the corners of the fins, which is not a good indicator, as the viscous layer is supposed to dampen turbulence around this region. Although the TVR is low at the wall, similarly low values appear in the vortex core above the fin horns, where more circulation is expected. In Figure 9b, the Spalart–Allmaras model shows diffusiveness due to a non-realistic, perfectly symmetric contour; this is because the one-equation model diffuses the production of turbulence and does not take into account the anisotropy in the Reynolds stress as a result of secondary flow. From Figure 9c, the $k-\omega$ SST model can be seen to meet all the criteria for a good model. First, it shows TVR values close to zero at the walls, correctly modeling turbulence damping by the viscous sublayer. Second, the model shows the highest TVR values in the vortex core region, where secondary flows are expected. Moreover, the asymmetry of the TVR distribution at this location captures the anisotropy. Therefore, the choice of the $k-\omega$ SST model was justified.

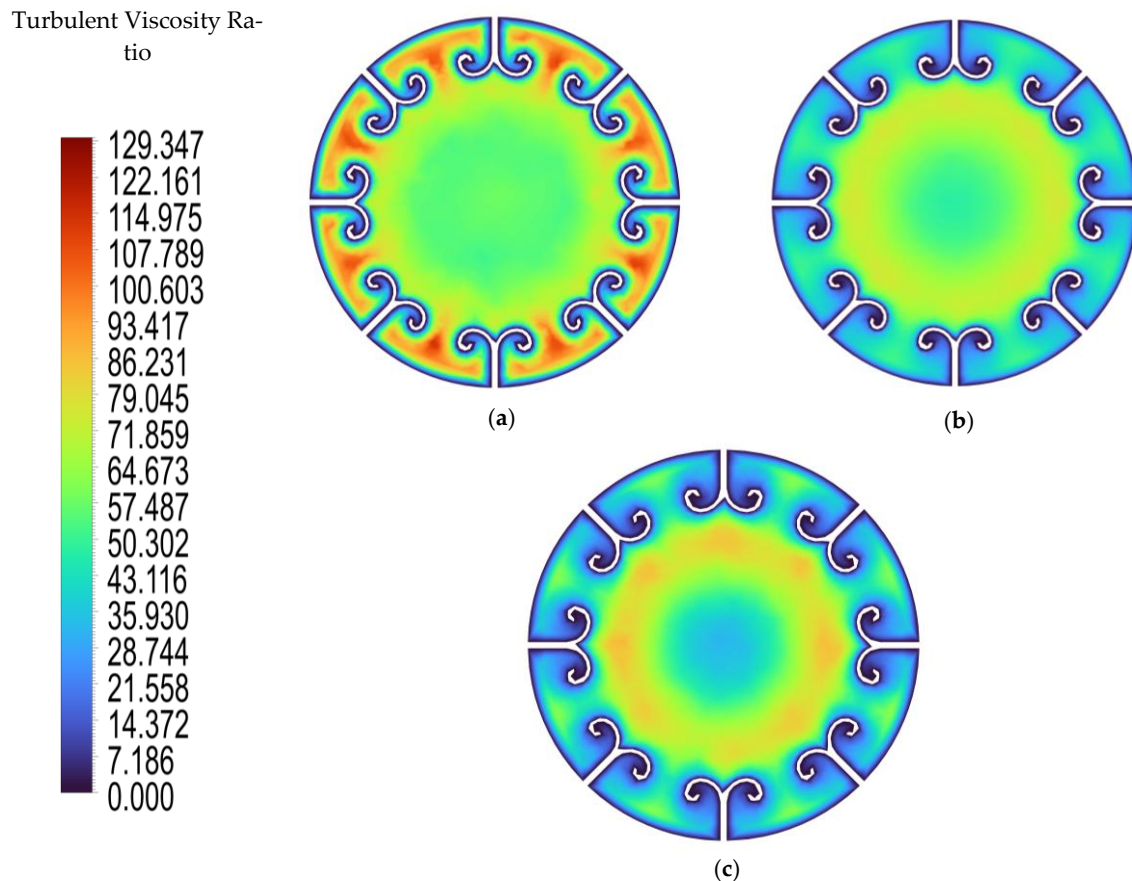


Figure 9. TVR contours at $x/L = 1/2$ ($N = 8$, $s/D = 0.1$, $Re = 100,000$): (a) $k-\varepsilon$ model; (b) Spalart–Allmaras model; and (c) $k-\omega$ SST model.

2.7. Proof of Fully Developed Flow

The pipe length was selected such that $L/D = 30$ ($L = 750$ mm, $D = 25$ mm) to eliminate entrance effects when computing velocity, temperature, and pressure values, and the fully developed region was verified in the most complex finned pipe case. To verify that a fully developed flow occurred, transverse contours of axial velocity were observed at three different locations ($x/L = \frac{1}{2}, \frac{3}{4}$, and $\frac{17}{20}$), which are shown in Figure 10a–c.

The results indicate qualitatively and quantitatively identical contours of the axial velocity profile on the transverse planes, thereby demonstrating fully developed flow. Temperature, velocity, and pressure computations for the measurement of Nu , f , and heat retention were therefore taken from the developed region starting at $x/L = 1/2$, where $L/D = 15$, in order to eliminate entrance effects. After completing the solution setup, a parametric study was conducted to investigate the thermohydraulic and heat retention performance of the novel finned pipe.

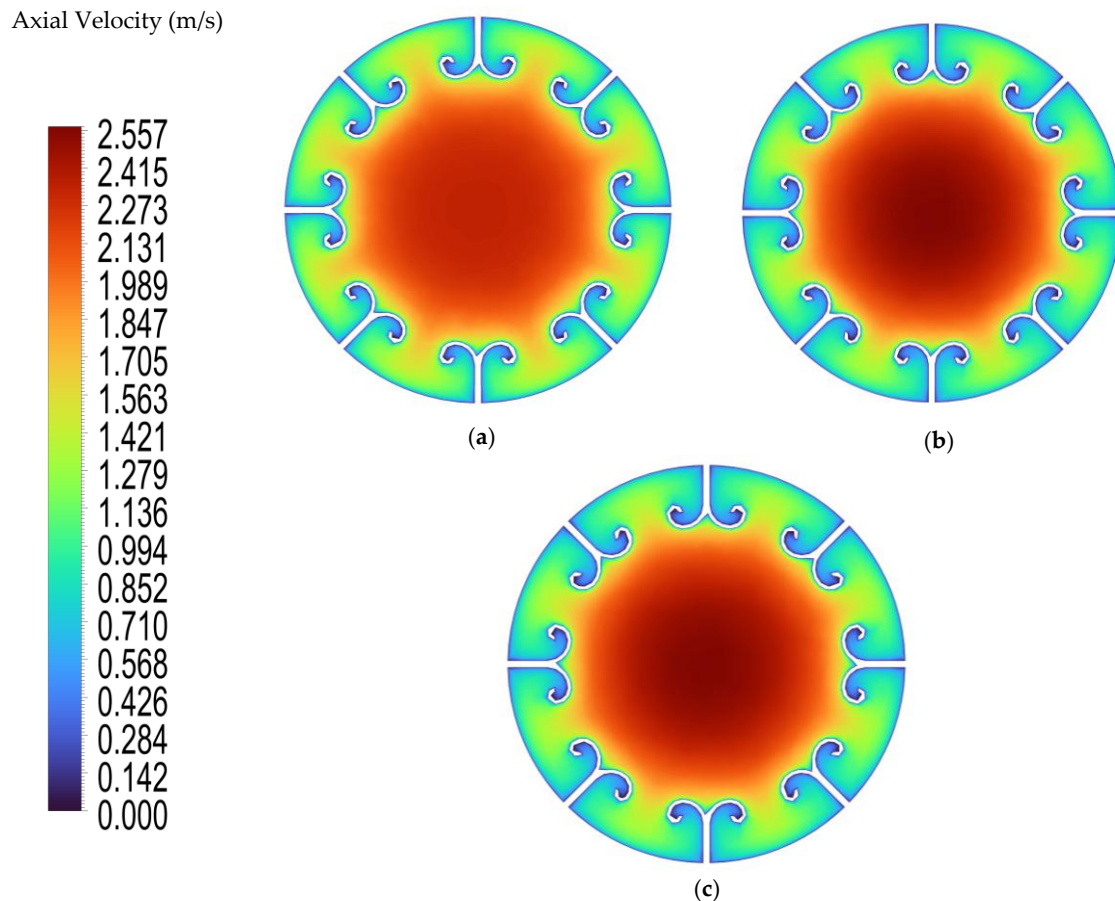


Figure 10. Contours of axial velocity: (a) $x/L = 1/2$; (b) $x/L = 3/4$; and (c) $x/L = 17/20$.

3. Results

3.1. Effects of Number of Fins on Thermohydraulic Performance and Heat Retention

This study investigated the effects of the number of fins on the pipe's thermohydraulic performance. Figure 11 shows how the Nusselt number varies with the Reynolds number for the cases $N = 4, 6$, and 8 evaluated at $s/D = 0.06$.

From Figure 11, it can be seen that the Nusselt number increases monotonically with the Reynolds number, and the results are nuanced under different fin numbers. The increase in the Nusselt number is attributed to the increased vortex core regions created by adding fins, which also increases the contact area; however, their geometry can create wake regions that may stagnate pockets of fluid, hindering Nu enhancement. As observed in the figure, $N = 6$ obtains optimum Nu amongst the three cases at certain Reynolds numbers ($Re = 20,000; 30,000; 50,000; 80,000$; and $100,000$), whereas at other Re numbers ($90,000$), $N = 8$ obtains the optimum Nu value. This shows that the Nusselt number depends not only on the Reynolds and Prandtl numbers, but also on the geometric parameter N . The sudden peak of Nu occurring at $N = 6$, $Re = 80,000$ is because, amongst the three cases observed, this fin number configuration at this Re value produced the least interfering secondary flow. This resulted in larger coherent fluid flow structures that sweep in the streamwise direction along locations of the vortex core to improve heat transfer in comparison to the other two cases (i.e., $N = 4$ and $N = 8$ at $Re = 80,000$). This result is further corroborated by the iso-surfaces of the λ_2 criterion, which indicate the mechanism by which flow action improves heat transfer, as shown in Figure 12.

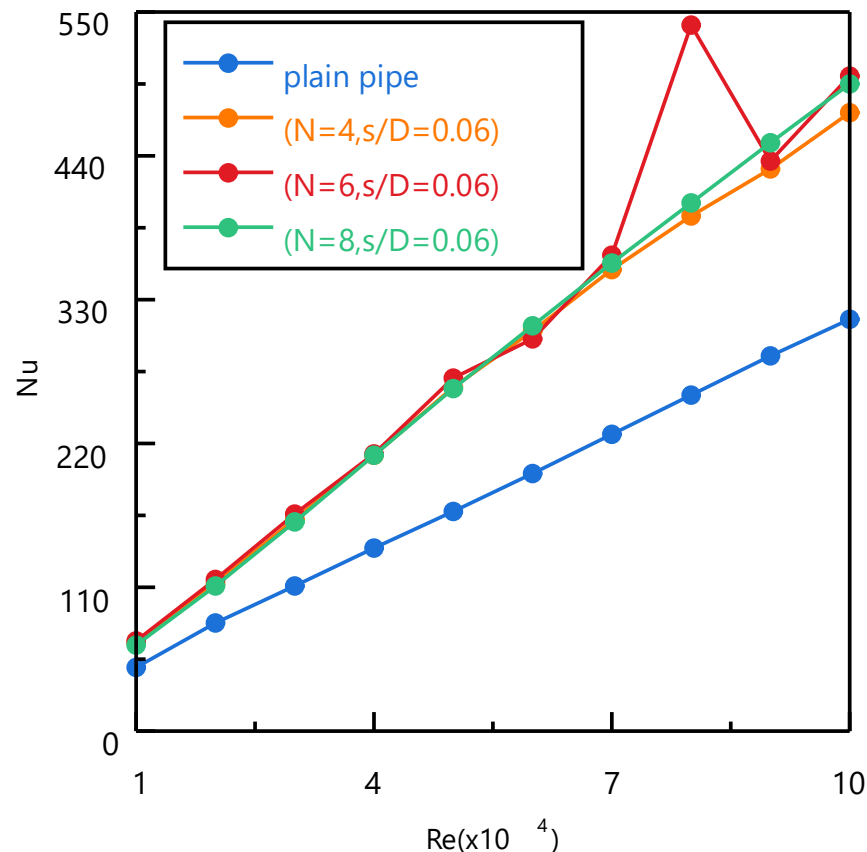


Figure 11. Nusselt number vs. Reynolds number ($N = 4, 6, 8$ at $s/D = 0.06$).

The λ_2 criterion [36] is used to identify vortical regions using the velocity gradient tensor $\nabla \mathbf{u}$, distinguishing regions of concentrated rotational motion from regions where strong shear or strain alone might otherwise be mistaken for a vortex. The velocity gradient tensor is first broken into symmetric and antisymmetric parts, which represent the strain/deformation matrix $\mathbf{S}$ and the rotation matrix of the fluid element $\mathbf{\Omega}$, respectively. After the three eigenvalues of the matrix $\mathbf{S}^2 + \mathbf{\Omega}^2$ are calculated, regions where the second eigenvalue, λ_2 , are negative are then identified as the vortex core regions. In Figure 12, the three cases $N = 4, 6$, and 8 are observed at the same Reynolds number ($Re = 80,000$). At this Re , a high value of -5000 s^{-2} for the λ_2 criterion, representing high vortical activity, is observed close to the outlet of each tube where the flow is fully developed.

These figures show that the strong coherent structures, responsible for boundary layer perturbation and heat transfer enhancement, are largest and more consistent in Figure 12b, where $N = 6$. This is because the angular pitch between the fins in this case (at $N = 6$, the angular pitch is 60°) is optimal for the formation of streamwise vortical cores and population of the boundary layer. In the case of $N = 8$, the fins are closely packed and, thus, have a smaller angular pitch (45°), which can cause interference and destructive conjunction between vortices, thereby weakening and breaking them up. In the case of $N = 4$, although the spacing between the fins is much larger (90° angular pitch), there are not enough fins to generate sufficient vortex core regions for this heat transfer enhancement mechanism to perpetuate along the tube. This explains why, at $Re = 80,000$, the $N = 6$ ranks first, followed by $N = 8$ and $N = 4$, in terms of the Nusselt number (according to Figure 10). Figure 13 shows a plot of the friction factor against Re for each of the three configurations.

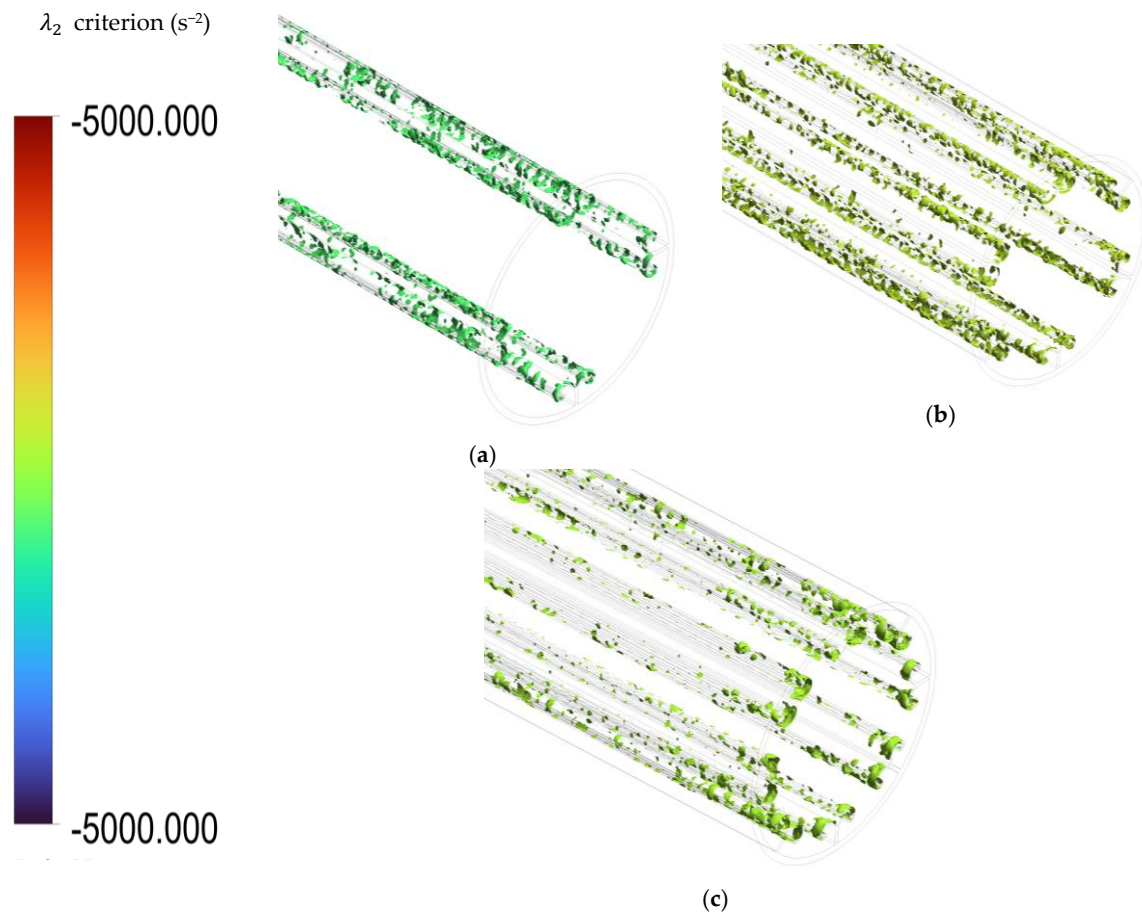


Figure 12. The -5000 s^{-2} iso-surfaces of the λ_2 criterion at $\text{Re} = 80,000$ for (a) $N = 4$; (b) $N = 6$; and (c) $N = 8$.

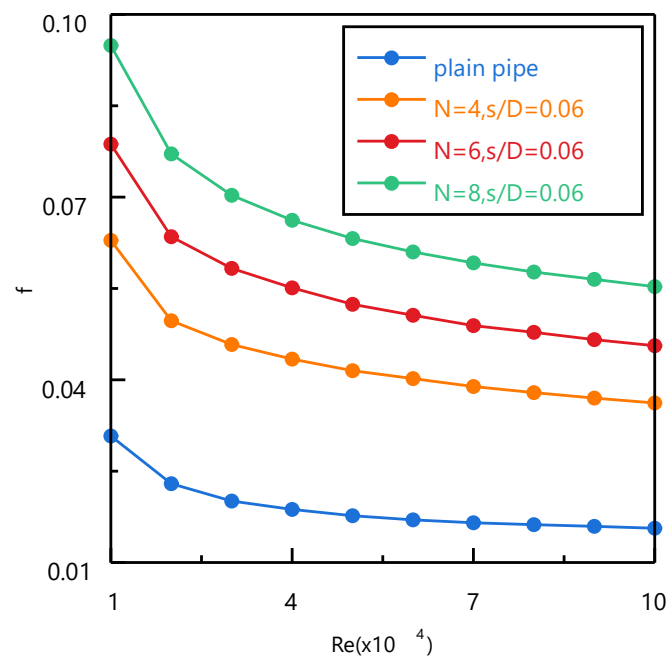


Figure 13. Friction factor vs. Re ($N = 4, 6, 8$ at $s/D = 0.06$).

The penalties introduced by the fins manifest as an increase in the friction factor, which decreases with increasing Reynolds number. Increasing the number of fins increases the friction factor due to the increased obstructive area, which creates local

pressure minima that interact with the bulk flow and increase the pressure drop. Doubling the number of fins from 4 to 8 increased the friction factor by 53%, as expected. To evaluate how the finned pipe improves heat transfer, we compare its Nusselt number to that of the plain pipe in Figure 14; similarly, we evaluate friction factor penalties according to the friction factor ratios in Figure 15.

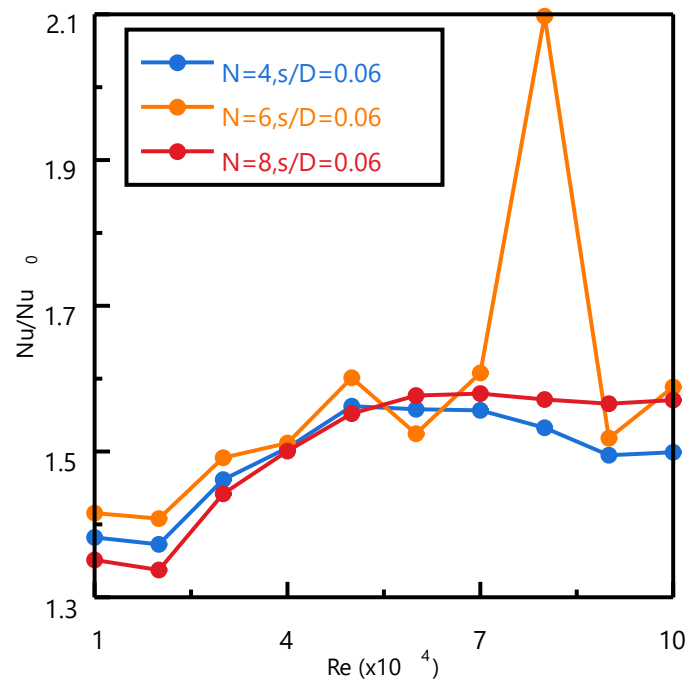


Figure 14. Effect of number of fins on the Nusselt number ratio.

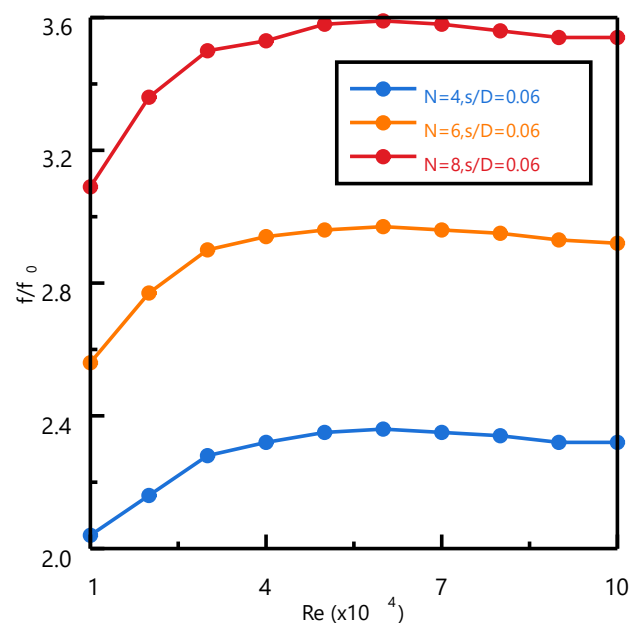


Figure 15. Effect of number of fins on the friction factor ratio.

All data points for all Re in Figure 14 are greater than 1, indicating improved heat transfer in the proposed exchanger. The Nusselt number varies strongly with changes in the geometric parameters and Re. From Re = 10,000 to Re = 20,000, the Nu improvement

reduces slightly for all N , with $N = 6$ yielding the highest enhancements (ranging from 42% to 110% at all Re values, except at $Re = 60,000$ and $Re = 90,000$); this is attributed to complex secondary flows and inter-fin interactions occurring as the Reynolds number increases. As the fluid flows over the curved fins, their curvature creates pressure and velocity gradients that vary not only radially toward the center of the pipe but also across the multiple radii of curvature, causing interactions that disrupt the boundary layer and enhance heat transfer. The curves in Figure 14 show that the best heat transfer enhancement occurs at $N = 6$ when $Re = 80,000$, resulting in a 110% increase in the Nusselt number. This performance of the second most populated fin case corroborates the position that Nu varies strongly with geometric parameters as well as fluid and flow properties, where the scientific reason for this optimal behavior was explained above. In Figure 15, the friction factor penalties can be seen to vary across several Reynolds numbers, as vortical structures create local pressure minima. The friction factor penalties increase sharply as more fins are added, when compared with the plain pipe, with a 260% increase occurring at $N = 6$ and $Re = 60,000$. Although thermal performance and hydraulic performance for the finned pipe were observed separately, analysis of combined effects is also necessary. Therefore, the combined thermohydraulic performance factor (TPF) was calculated to determine whether the friction factor or pressure drop penalties are worth sacrificing to obtain better heat transfer. A value greater than one indicates an overall thermohydraulic performance improvement, thus justifying the tradeoff. Figure 16 plots the TPF in terms of the Nusselt number ratio and friction factor ratio against Re .

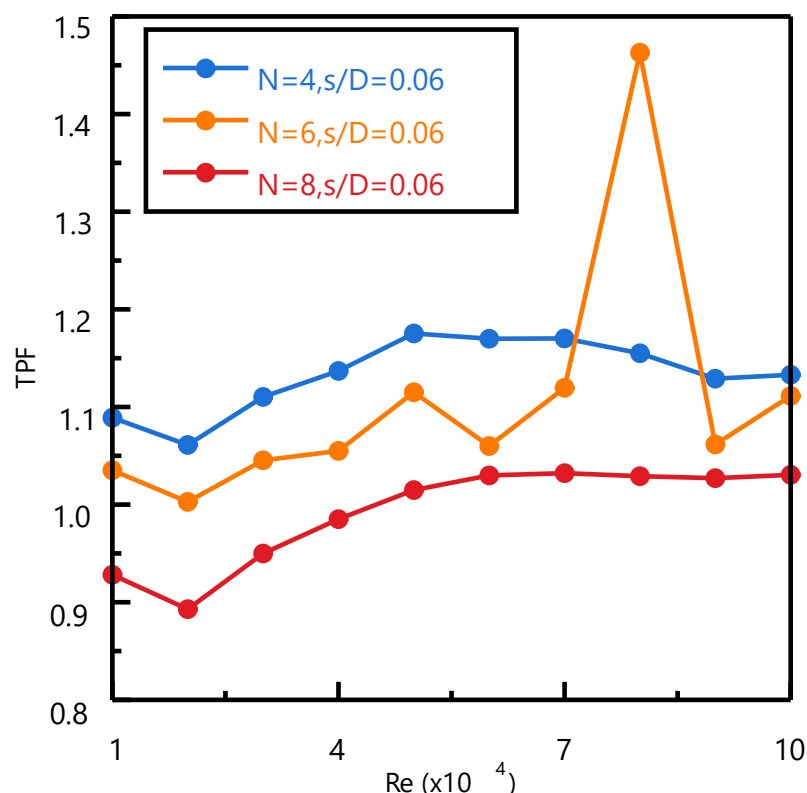


Figure 16. Effect of fin number on overall thermohydraulic performance.

From the results, $N = 6$ showed the best TPF, with a maximum value of 1.46 at $Re = 80,000$, which can be attributed to its significant heat transfer enhancement far outweighing the hydraulic cost. The overall thermohydraulic performance improved due to several factors, including increased surface area and the generation of local swirls that

continuously disrupt the boundary layer. Another metric used to quantify thermal performance was the heat retention ratio, $\frac{\dot{Q}}{\dot{Q}_0}$, which measures the heat retained by the fluid as it interacts with the pipe's heated walls. It is one of the most significant industrial metrics, as a good heat exchanger tube absorbs heat energy well from the heated walls. In particular, in this study, a value greater than one indicates that the fluid absorbs more heat when the finned pipe is used instead of the plain pipe under the same mass flow conditions. Figure 17 shows a plot of the heat retention ratio against Re for N = 4, 6, and 8 at s/D = 0.06.

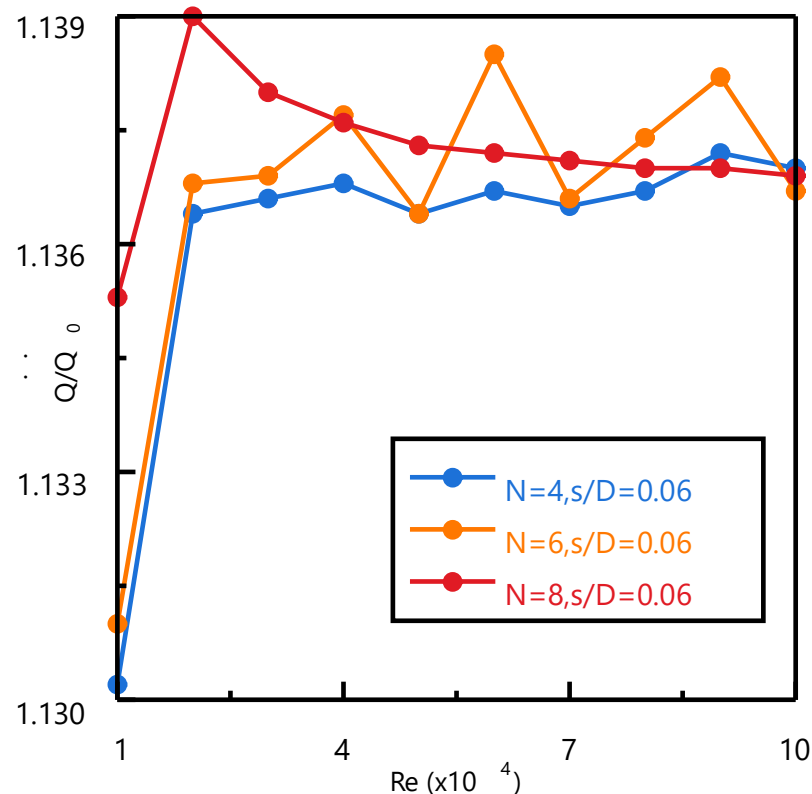


Figure 17. Effect of fin number on heat retention rate.

The plot shows that adding fins improves the heat retention rate. This is consistent with the physics of the flow, which describes heat absorption as a consequence of an increased wetted area; that is, a greater wetted area increases molecular collisions between fluid particles and the heated walls, increases residence time, and thereby increases heat transfer to the fluid. The graph shows a 13.9% maximum improvement in heat absorption occurring at N = 8 and Re = 20,000. For each fin number configuration, as the Reynolds number increases, the heat absorption ratio varies strongly, with N = 6 showing almost oscillatory behavior, which can be attributed to the strong influence of geometric parameters on the secondary flow structure and, consequently, heat retention. An increase in heat retention is significant for the application of such finned pipes in geothermal heat exchangers, where heat from the earth must be absorbed into a bulk fluid of interest. Figures 18–23 further confirm the heat transfer enhancement mechanism by visualizing the flow. These post-processed plots include contour plots of temperature, velocity, turbulent kinetic energy, vorticity (curl), and the λ_2 criterion iso-surface, respectively. The plots (except for the three-dimensional iso-surface plots of the λ_2 criterion) are over the plane at L/2 in the transverse direction, for Re = 50,000 and s/D = 0.06.

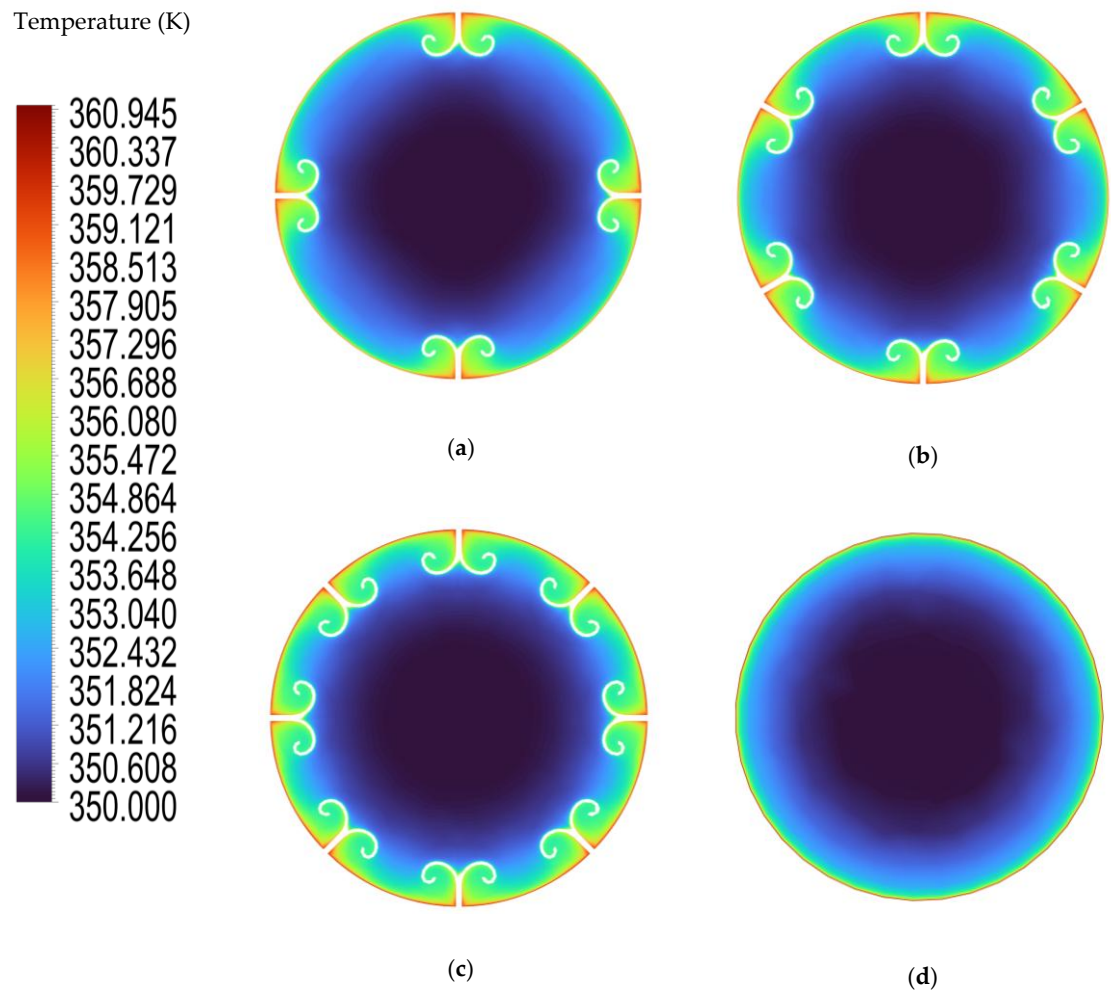


Figure 18. Transverse temperature contours at $Re = 50,000$: (a) $N = 4$, $S/D = 0.06$; (b) $N = 6$, $S/D = 0.06$; (c) $N = 8$, $S/D = 0.06$; and (d) plain pipe.

In Figure 18a–d, temperature is highest at the walls and at the base of the fin, which is expected as this is the first point of contact with the heat flux being applied. Temperature reduces toward the bulk flow. The plain pipe shows a relatively thicker thermal boundary layer, which justifies why the finned cases enhance heat transfer. Furthermore, the thermal boundary layer forms around the fins and walls, demonstrating the fins' contribution to improved heat transfer. As the fin number increases, warmer regions in the thermal boundary layer increase, thereby reducing thermal gradients and improving heat transfer. This observation does not conclusively imply that the most populated fins will be associated with the best heat transfer, as other factors such as secondary flows can contribute to heat transfer performance.

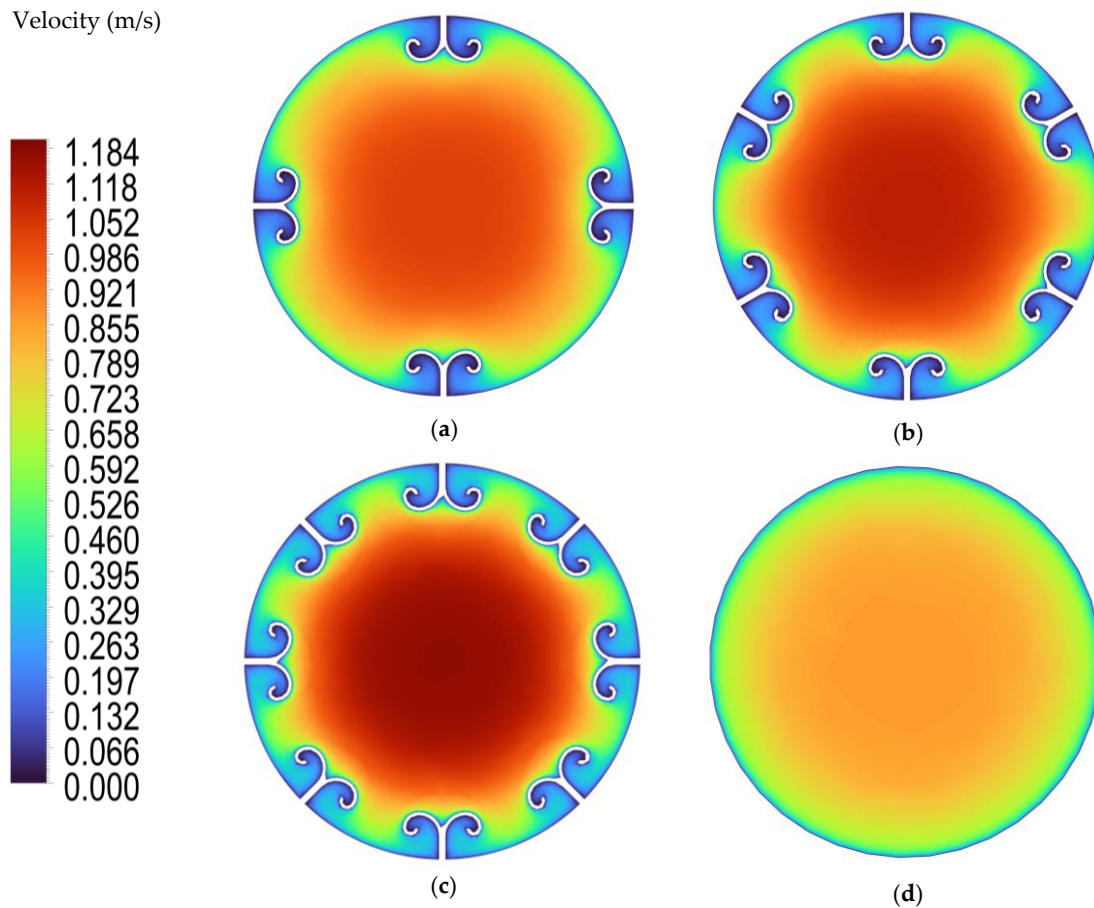


Figure 19. Transverse velocity contours at $Re = 50,000$: (a) $N = 4$, $S/D = 0.06$; (b) $N = 6$, $S/D = 0.06$; (c) $N = 8$, $S/D = 0.06$; and (d) plain pipe.

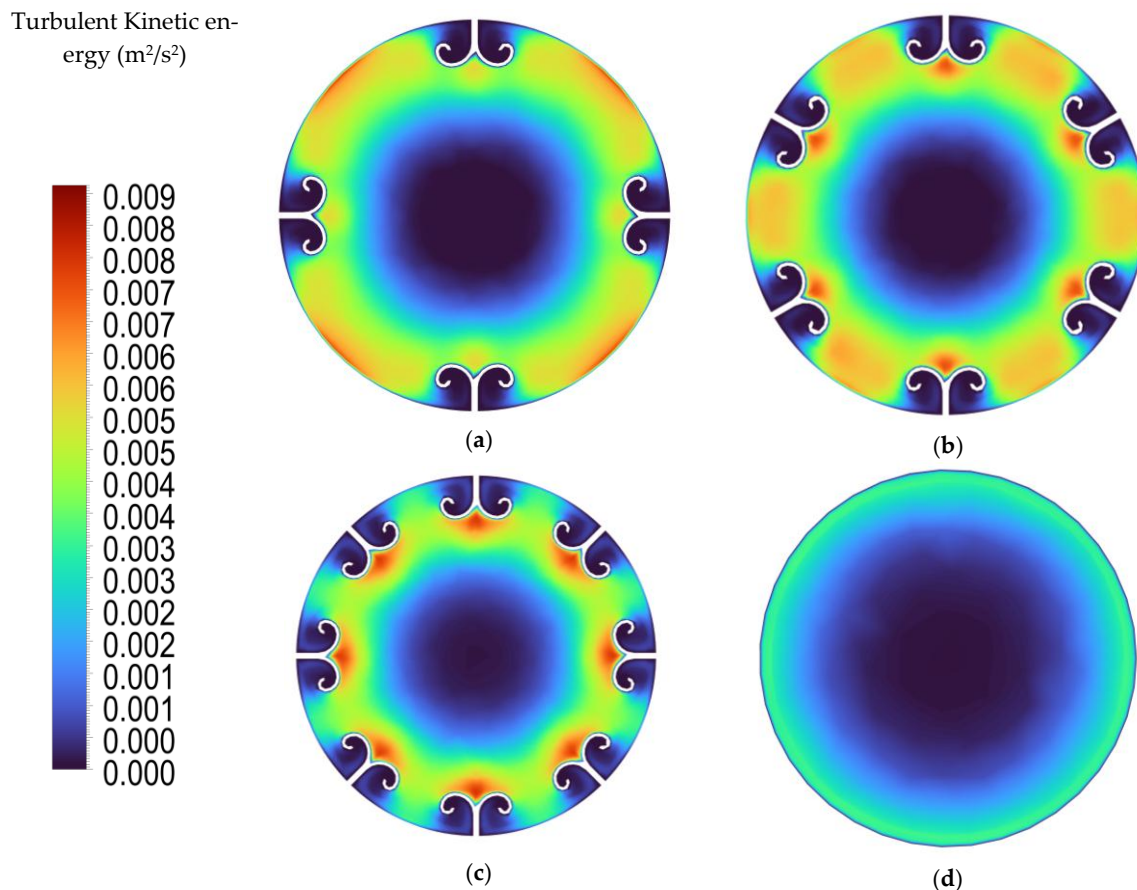


Figure 20. Transverse turbulent kinetic energy contours at $Re = 50,000$: (a) $N = 4$, $S/D = 0.06$; (b) $N = 6$, $S/D = 0.06$; (c) $N = 8$, $S/D = 0.06$; and (d) plain pipe.

The velocity contours in Figure 19a–d show the no-slip boundary condition, which is enforced at the walls, with the maximum velocity occurring in the bulk flow at the center. For the plain pipe case, a smooth velocity boundary layer is observed on this plane, while for the finned cases, velocity changes occur between the fins and the vortex core region, with low-to-high velocity transitions from under the ‘umbrella’ created by the horns to wall regions without the fins. These observations hint at boundary layer perturbation via fluctuations in velocity as a mechanism of heat transfer, as further supported by the turbulent kinetic energy contours in Figure 20a–d, which are representative of average velocity fluctuations and turbulence production. This flow variable is maximized in local regions near the fins, indicating significant turbulence production. The plain pipe case (Figure 20d) shows low TKE values, supporting less heat transfer via perturbations in the boundary layer when compared to the finned cases. At the considered Reynolds number, $N = 6$ yielded the highest Nusselt number (270); the reason for this result is that, although regions of high TKE increase as N increases, at $N = 6$ an optimum angular pitch of 60° spaces the fins, thus enabling the fluctuations to sweep across the non-finned wall areas. In contrast, in the case of $N = 8$, the more closely packed fins reduce turbulence production in such regions; furthermore, very little to no activity is observed in the bulk flow at the center of the tube. However, increasing the number of fins increases local regions of velocity fluctuations, indicating secondary flows that improve heat transfer. The contours of surface streamlines, vorticities, and three-dimensional -1000 s^{-2} iso-surfaces of λ_2 criterion were further used to identify vortex core regions where secondary flows disrupt the boundary layer, further supporting the posited mechanism.

Streamlines (m/s)

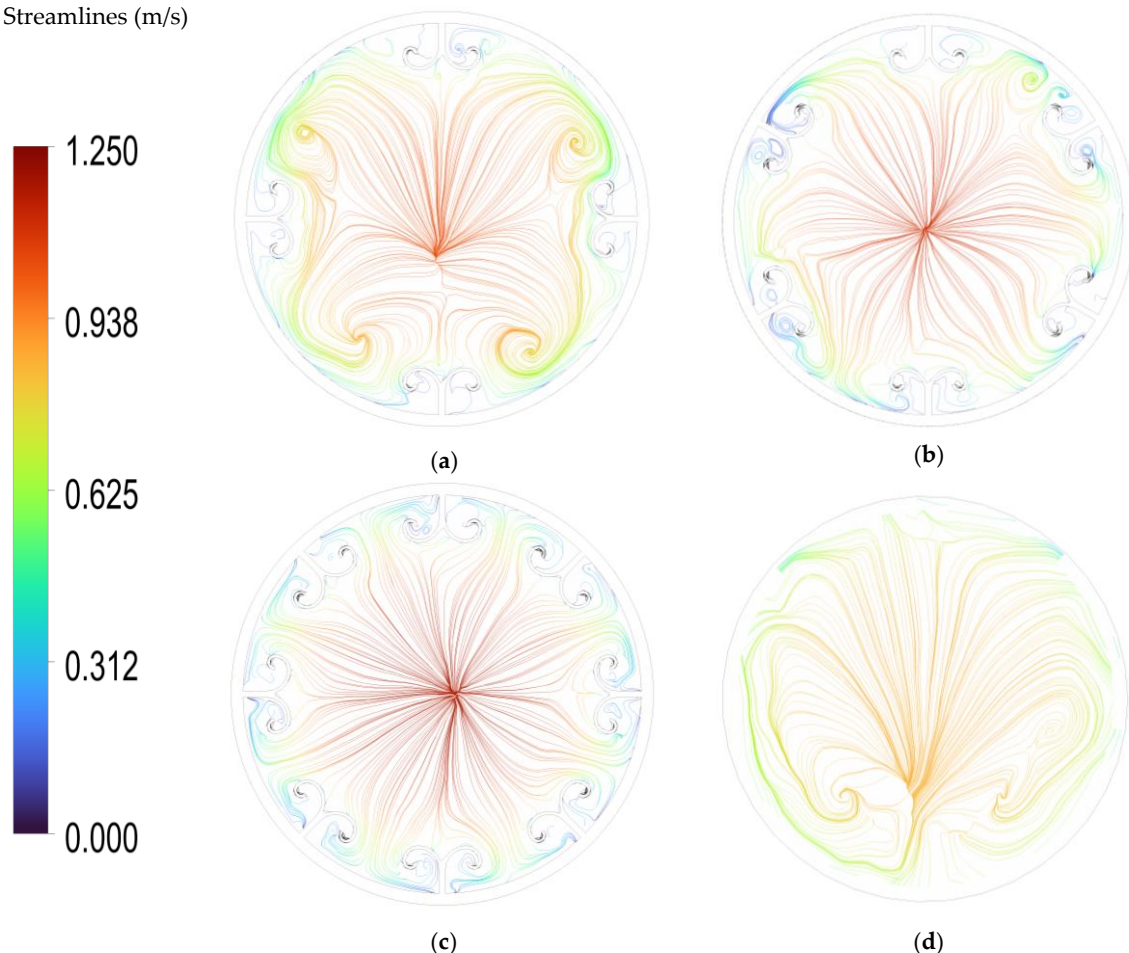


Figure 21. Surface streamlines at mid-plane ($Re = 50,000$): (a) $N = 4$, $S/D = 0.06$; (b) $N = 6$, $S/D = 0.06$; (c) $N = 8$, $S/D = 0.06$; and (d) plain pip

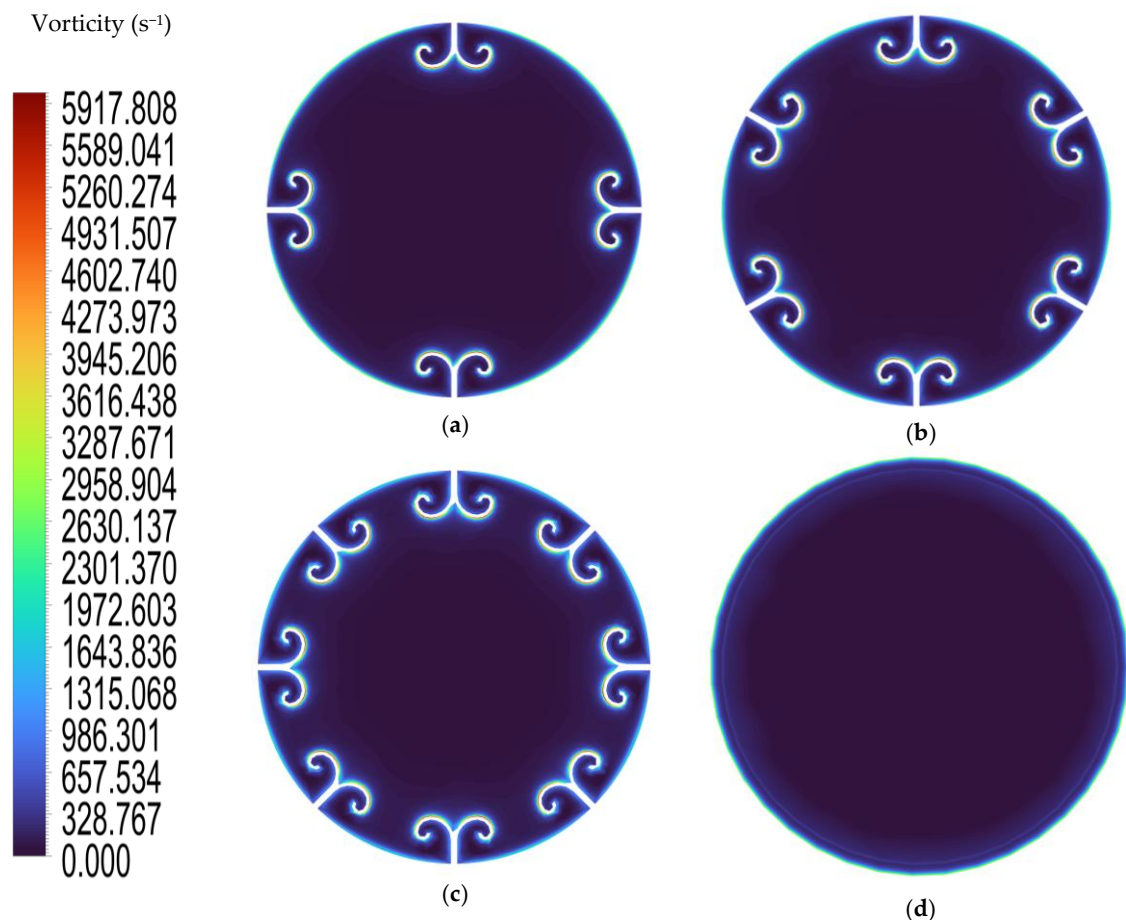


Figure 22. Transverse vorticity contours at $Re = 50,000$: (a) $N = 4$, $S/D = 0.06$; (b) $N = 6$, $S/D = 0.06$; (c) $N = 8$, $S/D = 0.06$; and (d) plain pipe.

Figure 21 shows the azimuthal and radial velocity components that make up the surface streamlines, from which circulatory motion can be observed between the fins as well as on top of the horns. As the number of fins increases, these local regions of secondary flow increase. The plain pipe case shows fewer signs of secondary flow when compared to the finned pipe cases, which indicates less fluid motion in the circumferential and radial directions to move colder fluid to and from the heated walls—an important mechanism for sustained and enhanced heat transfer. The transverse vorticity contours in Figure 22 also demonstrate an increase in high-value rotational and shear motion around the fin horns, which also increases with increasing fin number. The plain pipe shows non-zero vorticity regions at the wall, but the vorticity in these regions is not as significant in magnitude as that in the finned cases. These non-zero vorticity regions do not necessarily imply the formation of a vortex, as shear flow/strain can also generate non-zero vorticity. Thus, the 3D iso-surfaces of the λ_2 criterion were used to identify and separate coherent vortical structures from regions of high strain/shear flow. This vortex identification method further supports the mechanism of heat transfer enhancement, with Figure 22 showing that these strong ($\sim 1000 \text{ s}^{-2}$) structures are found directly on top of the horns. In the plain pipe case, although regions of non-zero vorticity were observed, the λ_2 criterion maps indicate no iso-surfaces, indicating that the reported vorticity values in Figure 22 were due to shear flow and not necessarily a formed set of coherent vortical structures. In contrast, the finned pipe cases show strong vortical structures that continuously sweep

boundary layers to enhance heat transfer; indeed, these regions form the vortex core, where the TKE is also maximal. These coherent structures create a perturbation action within the boundary layer and ultimately improve heat transfer.

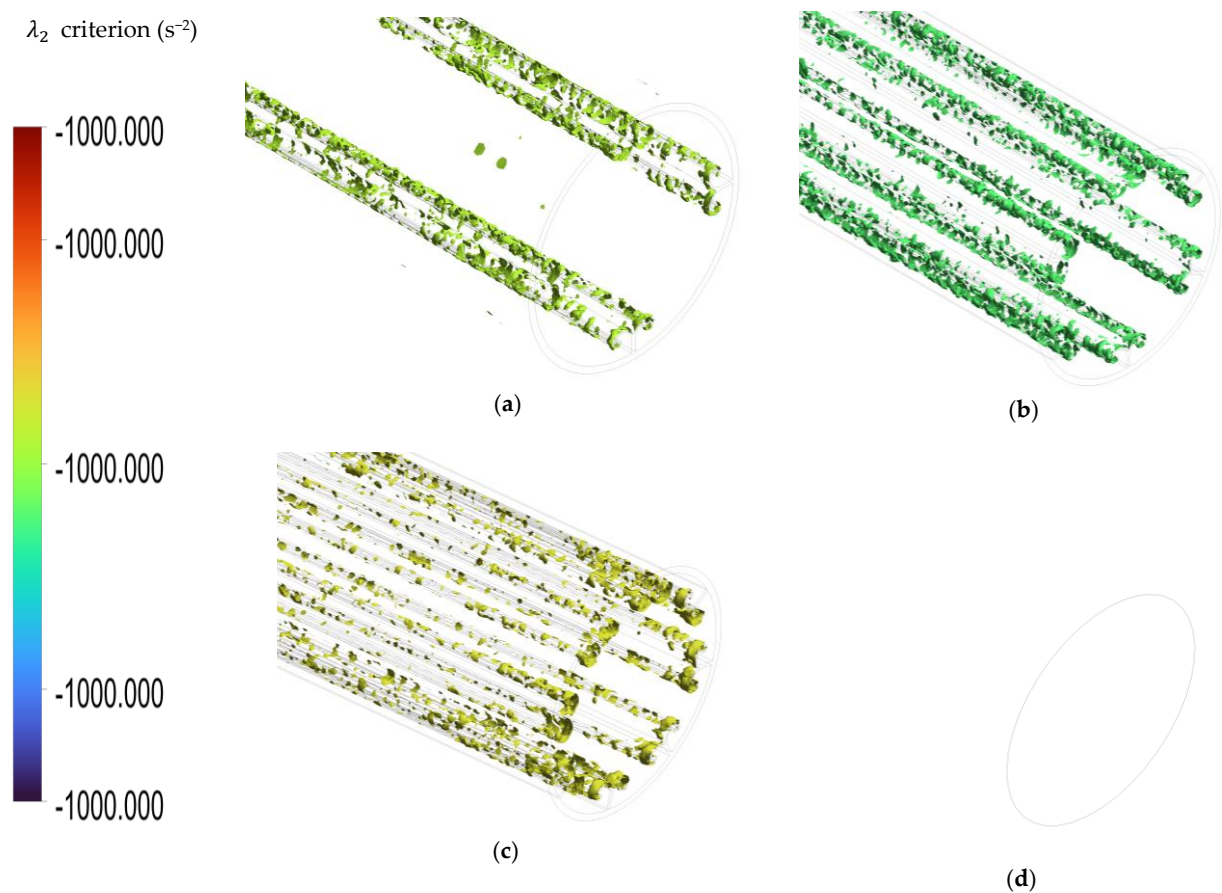


Figure 23. The $-1000 \text{ s}^{-2} \lambda_2$ criterion iso-surfaces at $Re = 50,000$: (a) $N = 4$, $S/D = 0.06$; (b) $N = 6$, $S/D = 0.06$; (c) $N = 8$, $S/D = 0.06$; and (d) plain pipe.

3.2. Effects of Fin Height on Thermohydraulic Performance and Heat Retention

The fin height was modified by extending (or shortening) the vertical stem upon which the horns sit. With $N = 8$ held constant, this stem height was varied to form three cases ($s/D = 0.06$, 0.08 , or 0.1). Increasing the fin stem height also increases the surface area and moves the fin toward the bulk flow region in the radial direction. Figures 24 and 25 show the variations in the Nusselt number and friction factor with Re for the three fin heights.

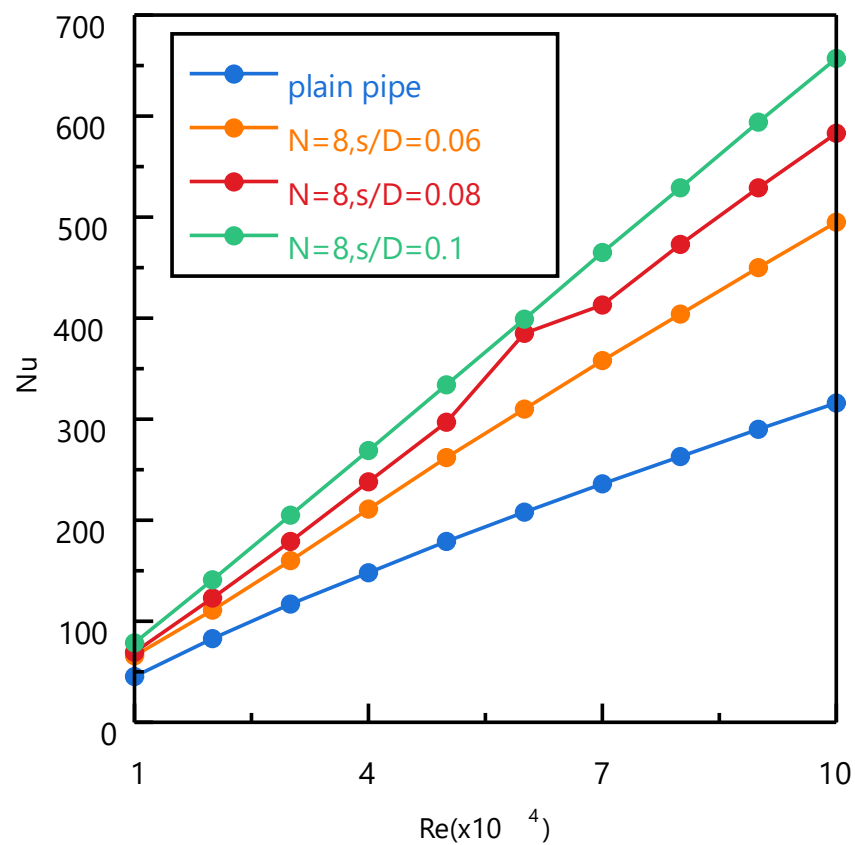


Figure 24. Nu vs. Re ($s/D = 0.06, 0.08, 0.1$ at $N = 8$).

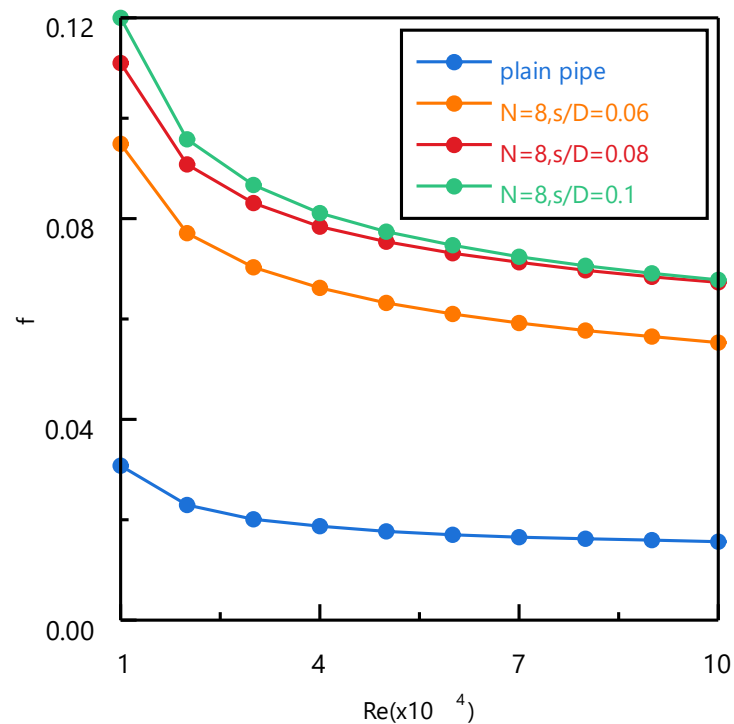


Figure 25. Friction factor vs. Re ($s/D = 0.06, 0.08, 0.1$ at $N = 8$).

According to Figure 24, the Nusselt number increased with increasing Reynolds number and fin height, with a 30% average increase in heat transfer after the fin height

increased by 67% from $s/D = 0.06$ to $s/D = 0.1$. Increasing the fin height increases the surface area, which increases Nu ; moreover, this height increase allows the fin horns to transfer more heat closer to the center of the bulk flow, which has the lowest temperature in this operation. However, elongation also increases the pressure drop, as indicated by the results shown in Figure 25. As the fin height increases, vertical obstruction of the flow also increases, causing local pressure gradients that further increase the pressure drop. The maximum penalty occurred when the fin stem increased from 6% to 10% of the pipe's diameter, with the friction factor increasing by 26.4% at $Re = 10,000$. Another key detail is that, as Re increased, the pressure drop penalties converged for both $s/D = 0.08$ and $s/D = 0.1$. This is a plausible indicator that the friction factor may change only slightly under high- Re flows, even when fin height is increased. Figures 26 and 27 compare the relative increments in Nu and f with respect to the plain pipe case.

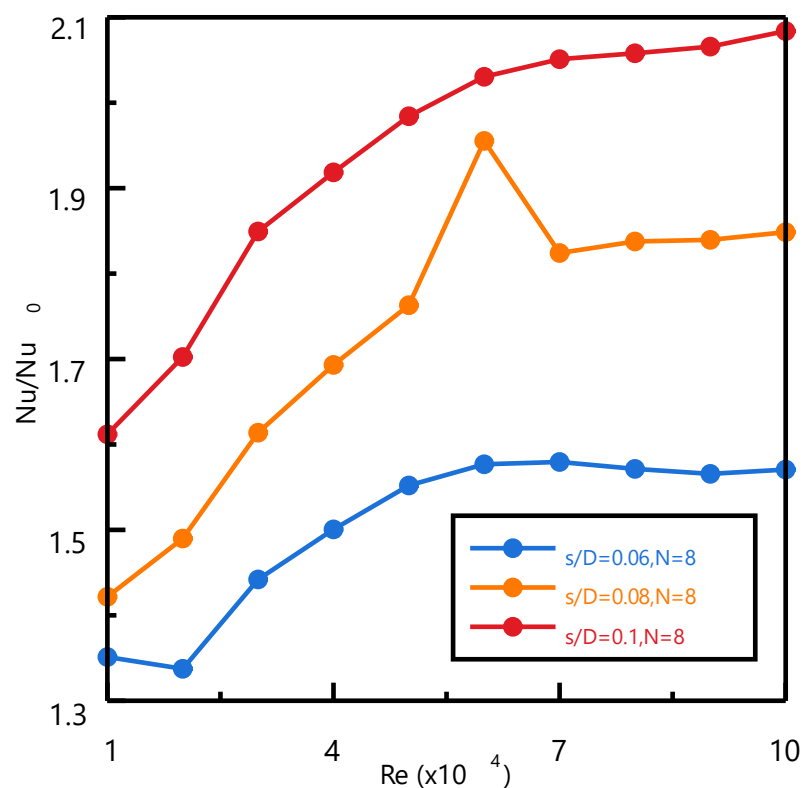


Figure 26. Effect of fin height on Nusselt number ratio.

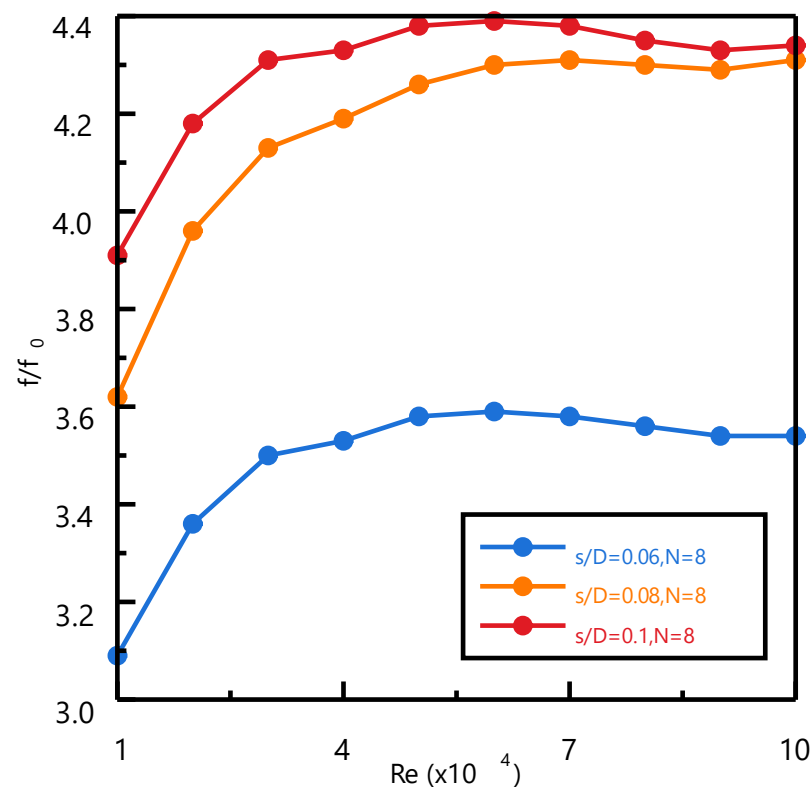


Figure 27. Effect of fin height on friction factor ratio.

From Figure 26, it can be seen that the Nusselt number increases with Reynolds number, although the slight dip around $Re = 10,000$ – $20,000$ for $s/D = 0.06$ indicates an enhancement that is not as large (in percentage terms) with respect to that of a plain pipe at the same Reynolds number. An increase in the Nusselt number with fin height was observed, which can be attributed to the increase in surface area, with a 108% improvement for $s/D = 0.1$. The spike at $Re = 60,000$ for $s/D = 0.08$ is most likely due to increased vortical activity at this Reynolds number, in combination with the favorable geometric parameters (e.g., angular pitch and fin height), leading to a quantitatively higher-than-usual improvement in heat transfer. Increasing the Reynolds number further bypasses these optimal conditions, where flow inertia is not as beneficial for coherent structures to persist. The friction factor ratio, as shown in Figure 27, was found to increase with the Reynolds number in the case of the proposed exchanger, incurring a higher pressure drop when compared to the plain pipe; in particular, the friction factor increased by as much as 339% at $s/D = 0.1$ and $Re = 50,000$ for the 8-fin configuration. This is due to obstructions in an inertia-strong, highly turbulent flow. The effect of fin height on combined thermohydraulic performance was also assessed, with Figure 28 showing the TPF values for $s/D = 0.06$, 0.08 , and 0.1 .

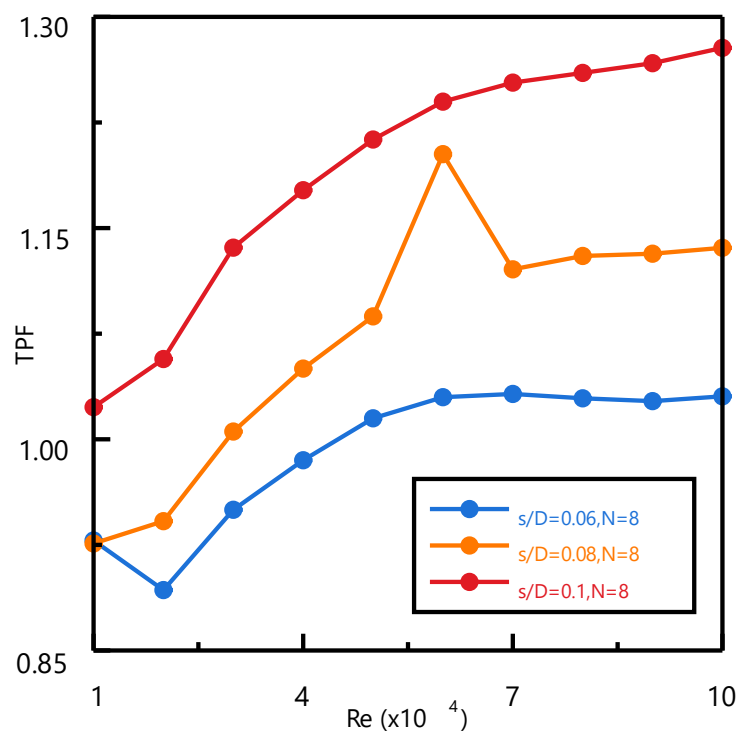


Figure 28. Effect of fin height on overall thermohydraulic performance.

The TPF was found to increase with increasing fin height and Re (Figure 28), and all values were greater than unity for $Re > 40,000$, indicating that the pressure drop penalty tradeoff is worthwhile. The maximum TPF of 1.28 was observed at $Re = 100,000$ and $s/D = 0.1$ for the 8-fin configuration. Figure 29 shows a plot of the heat retention ratio against the Reynolds number for the three different fin heights.

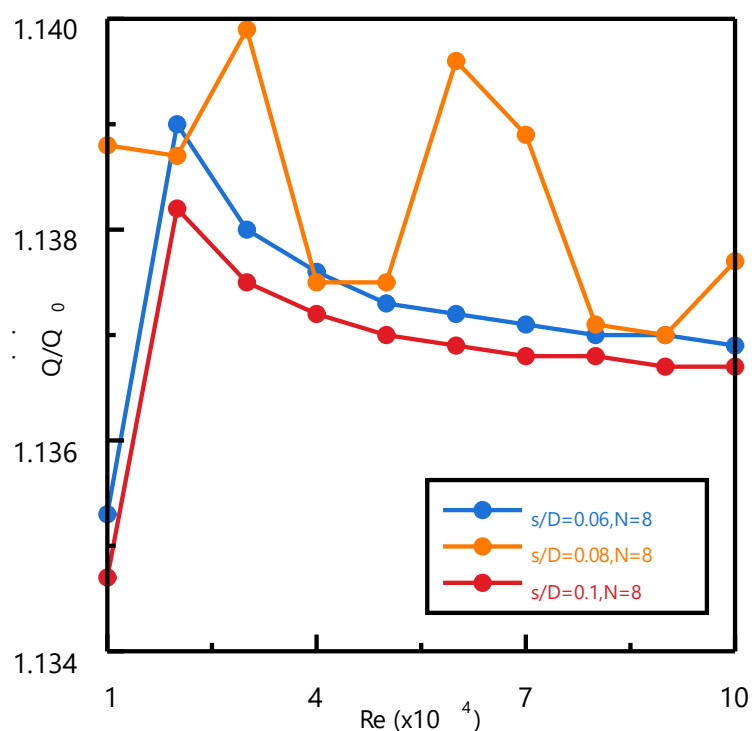


Figure 29. Effect of fin height on overall thermal performance (based on the heat retention rate ratio).

The heat retention ratio was found to vary strongly with fin height and Re . Over the considered Reynolds number range, the fins improved heat retention, with a 13.9% maximum improvement for $N = 8$, $s/D = 0.08$, and $Re = 30,000$. The temperature, velocity, turbulent kinetic energy, and vorticity contours, along with surface streamlines and 3D λ_2 criterion iso-surfaces, provide quantitative and qualitative insights into the mechanisms of heat transfer enhancement as the fin height increases. Figure 30a–d show transverse contours of temperature at $L/2$ for $Re = 50,000$, while Figures 31a–d and 32a–d show velocity and turbulent kinetic energy contours, respectively, at the same plane location. Figures 33–35a–d show surface streamlines, vorticity contours, and 3D λ_2 iso-surfaces, respectively.

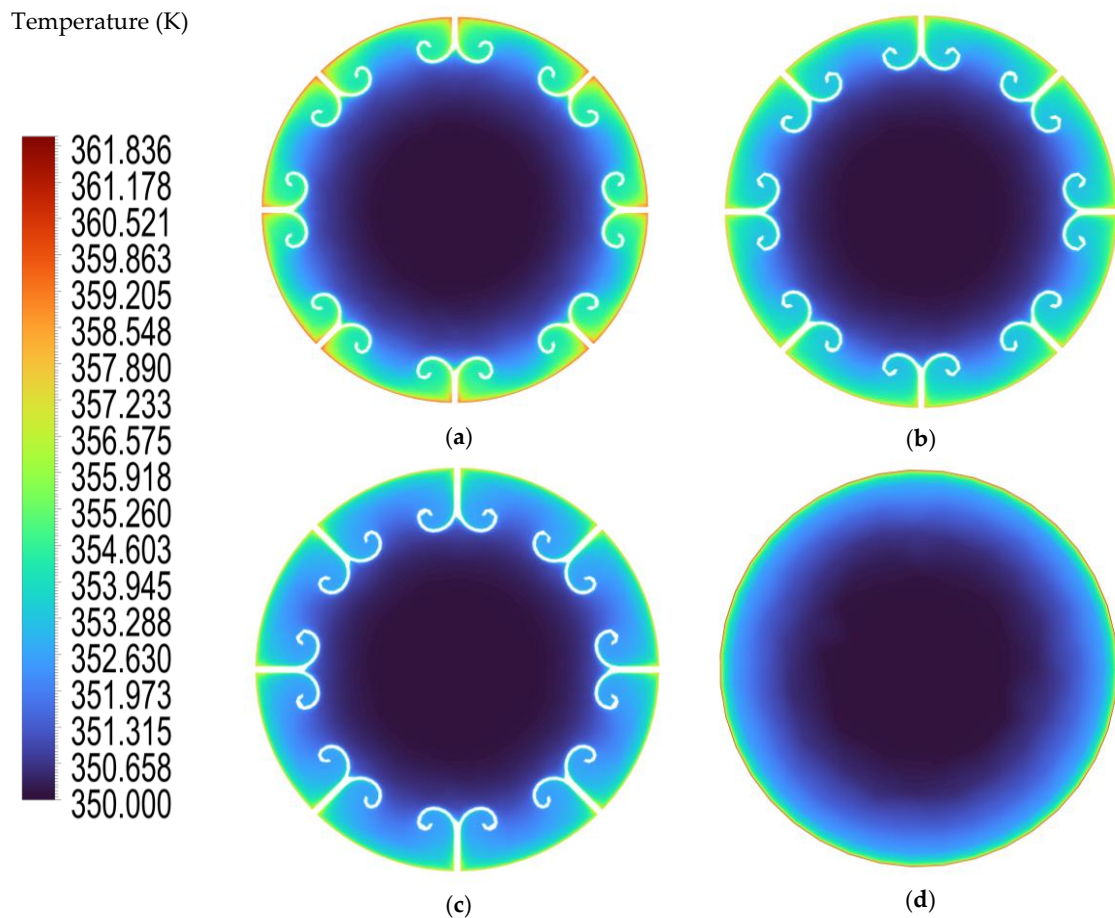


Figure 30. Transverse temperature contours at $Re = 50,000$: (a) $N = 8$, $S/D = 0.06$; (b) $N = 8$, $S/D = 0.08$; (c) $N = 8$, $S/D = 0.1$; and (d) plain pipe.

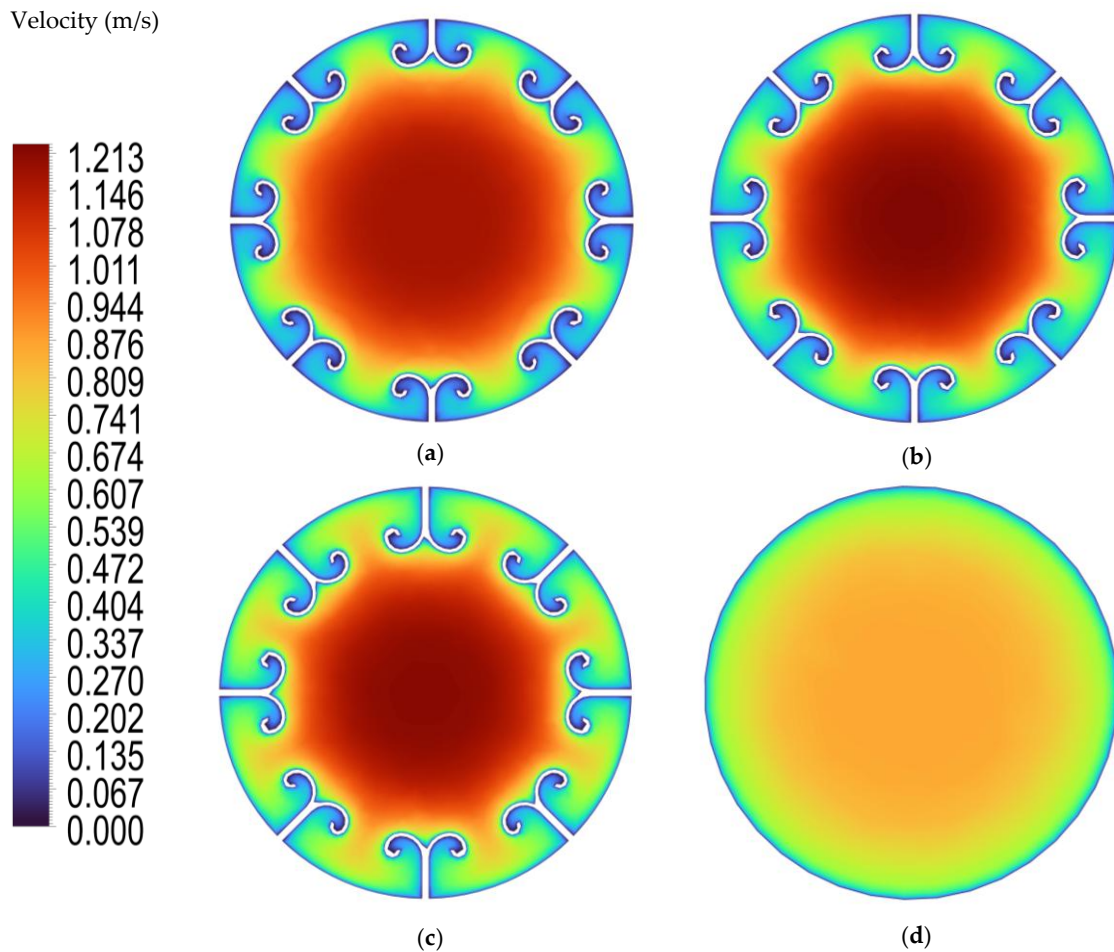


Figure 31. Transverse velocity contours at $Re = 50,000$: (a) $N = 8$, $S/D = 0.06$; (b) $N = 8$, $S/D = 0.08$; (c) $N = 8$, $S/D = 0.1$; and (d) plain pipe.

The temperature contours in Figure 30 show temperature changes within the fluid domain as the fin height changes. In the plain pipe case (Figure 30d), the wall temperature is the highest, providing evidence for why this is achieved the lowest Nu value amongst all compared cases at this Re , due to its thicker boundary layer. As the fin height increases, the contact surface increases and more circulation is promoted underneath the horns of the fins, allowing for a reduced wall temperature and increased heat transfer. Figure 30a–c support this idea, as the wall temperature is observed to decrease as the fin height increases. At $Re = 50,000$, Figure 31a–d show how increasing the fin height impacts the velocity flow field. As s/D increases from 0.06 to 0.1, the low-velocity regions under the horn “umbrellas” become larger and extend further towards the bulk flow, thus creating stronger transverse velocity gradients between the vertical stems of the fins. These gradients promote cross-stream momentum transport and secondary flows (including possible longitudinal vortices induced by the horn curvature), which disrupt the thermal boundary layer more effectively than shorter fins. The core flow velocity increases slightly with taller fins due to greater blockage (i.e., smaller effective flow area), consistent with mass conservation at the mass flow rate corresponding to the geometry-specific hydraulic diameter at fixed Re . The no-slip condition is strictly enforced at all fin surfaces, and flow recovery is facilitated by the smooth Fibonacci-based curvature.

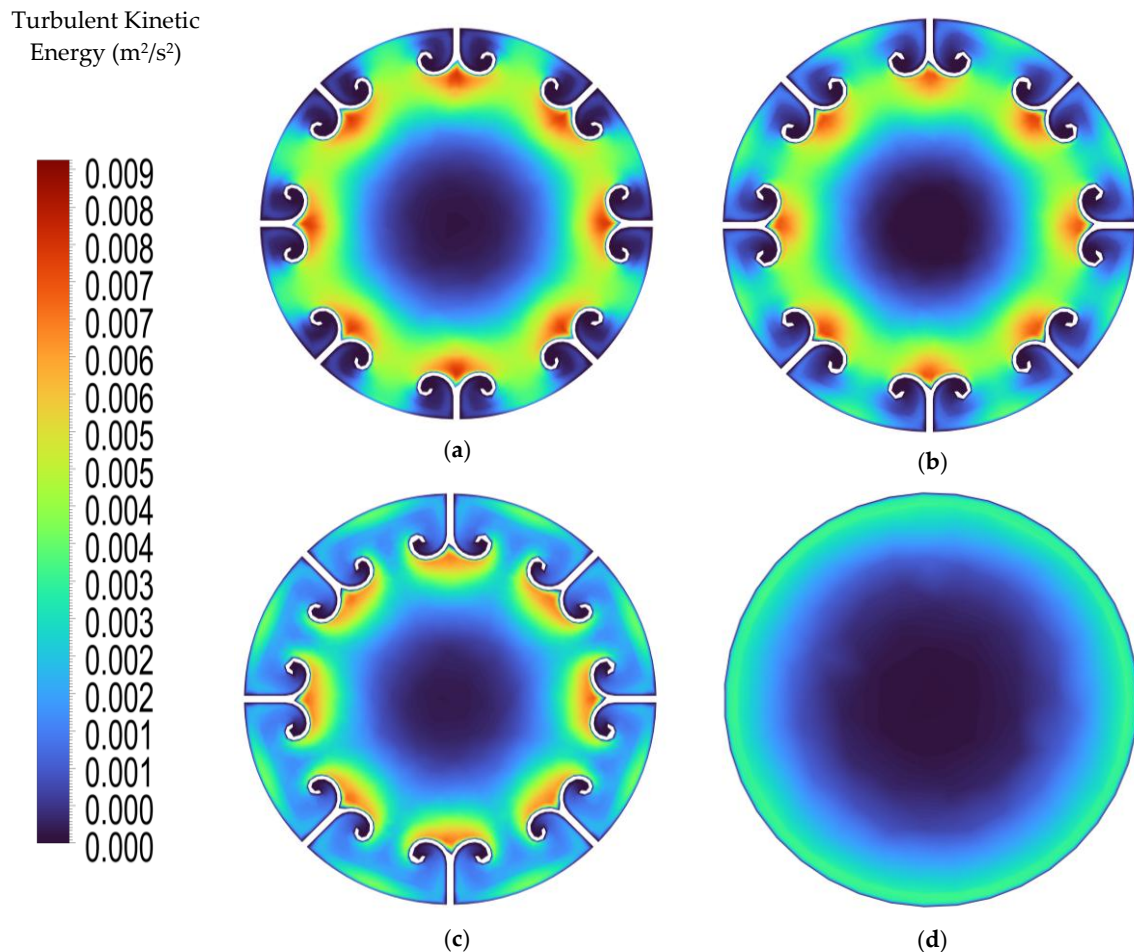


Figure 32. Transverse turbulent kinetic energy contours at $Re = 50,000$: (a) $N = 8$, $S/D = 0.06$; (b) $N = 8$, $S/D = 0.08$; (c) $N = 8$, $S/D = 0.1$; and (d) plain pipe.

The turbulent kinetic energy (TKE) contours in Figure 32a–d provide further insight into the turbulence production mechanisms. The plain pipe case, shown in Figure 33d, indicates low turbulence production, while TKE is predominantly generated in the shear layers originating from the horn tips, the stem–fin junctions, and the curved surfaces of the finned pipe cases in Figure 32a–c. As the fin height increases, these high-TKE regions intensify and spread over a larger portion of the near-wall and inter-fin spaces, indicating enhanced turbulent mixing and momentum exchange. The Fibonacci spiral curvature likely contributes to a favorable pressure gradient distribution that sustains attached flow over much of the horn surface while still generating beneficial vortical structures at the trailing edges, thus avoiding the excessive form drag typical of sharp-edged turbulators. Importantly, the TKE levels in the pipe core remain relatively low across the board, suggesting that the enhancement mechanism is efficient.

The surface streamlines for the finned pipe cases in Figure 33a–c show increased circulation between the fins and at the vortex core region, an expected boundary layer perturbation that improves heat transfer. Taller fins create larger umbrellas that promote the circulatory motion of fluid at scales equivalent to the perceived diameter of coherent structures responsible for secondary flows. In contrast, the plain pipe case shows low secondary flow activity, limiting the radial and azimuthal motion of fluid that is mainly responsible for transporting thermal energy within the fluid to and from the walls to facilitate heat transfer. This mechanism is further corroborated by the vorticity contours in Figure 34a–c, which indicate increased strain and rotation of fluid particles on top of the fin horns; again, while the plain pipe case showed non-zero vorticity regions, these were later

confirmed to have no coherent vortical structures from the λ_2 criterion iso-surface in Figure 35d. The λ_2 criterion iso-surfaces in Figure 35a–c provide evidence for the secondary flows through vortical structures that perturb the boundary layer with increasing fin height, further improving Nu. Strong coherent structures can be seen at the tops of the fins, and even between the fins at the inner wall in the case of $s/D = 0.1$ due to increased secondary flow under its curvature.

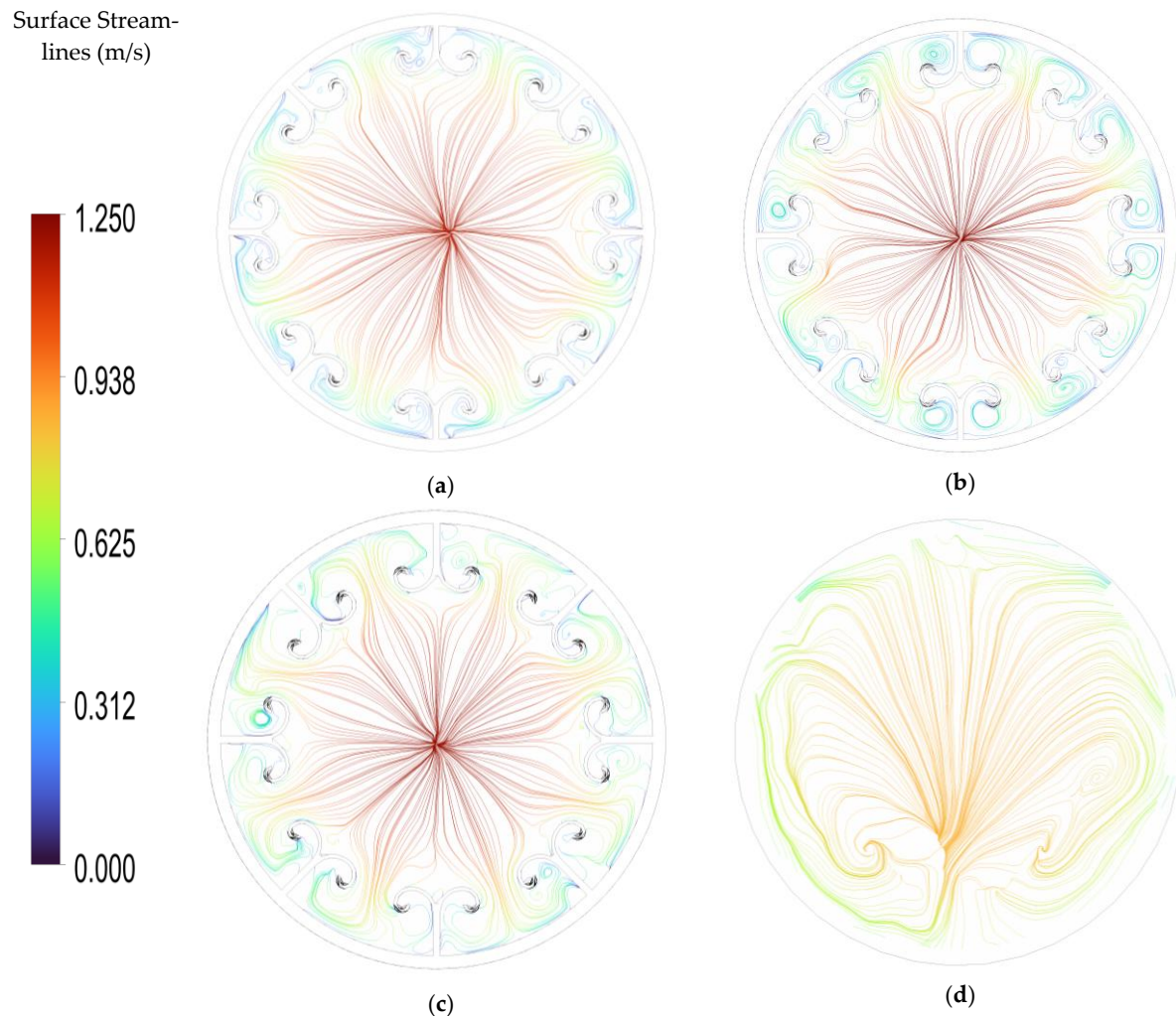
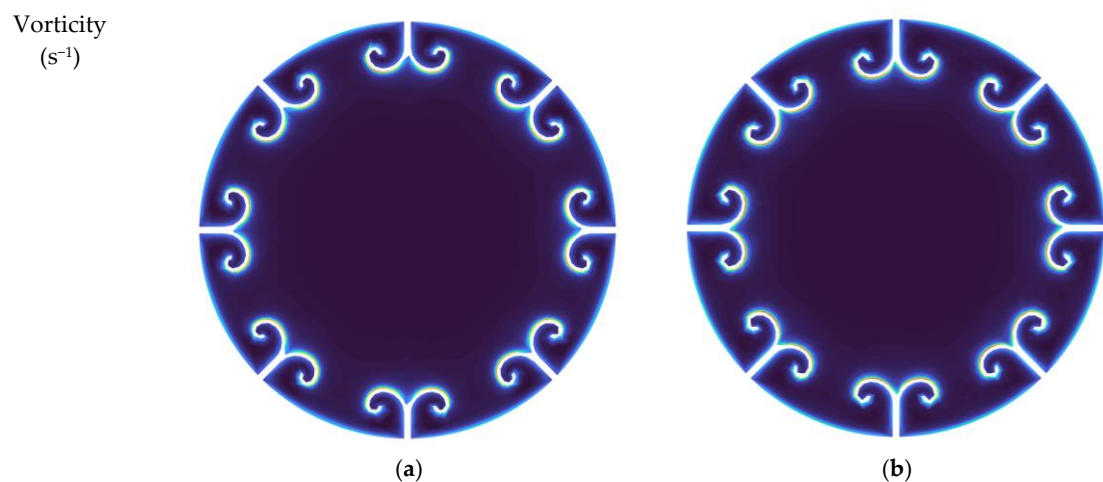


Figure 33. Surface streamlines at the mid-plane ($Re = 50,000$): (a) $N = 8$, $S/D = 0.06$; (b) $N = 8$, $S/D = 0.08$; (c) $N = 8$, $S/D = 0.1$; and (d) plain pipe.



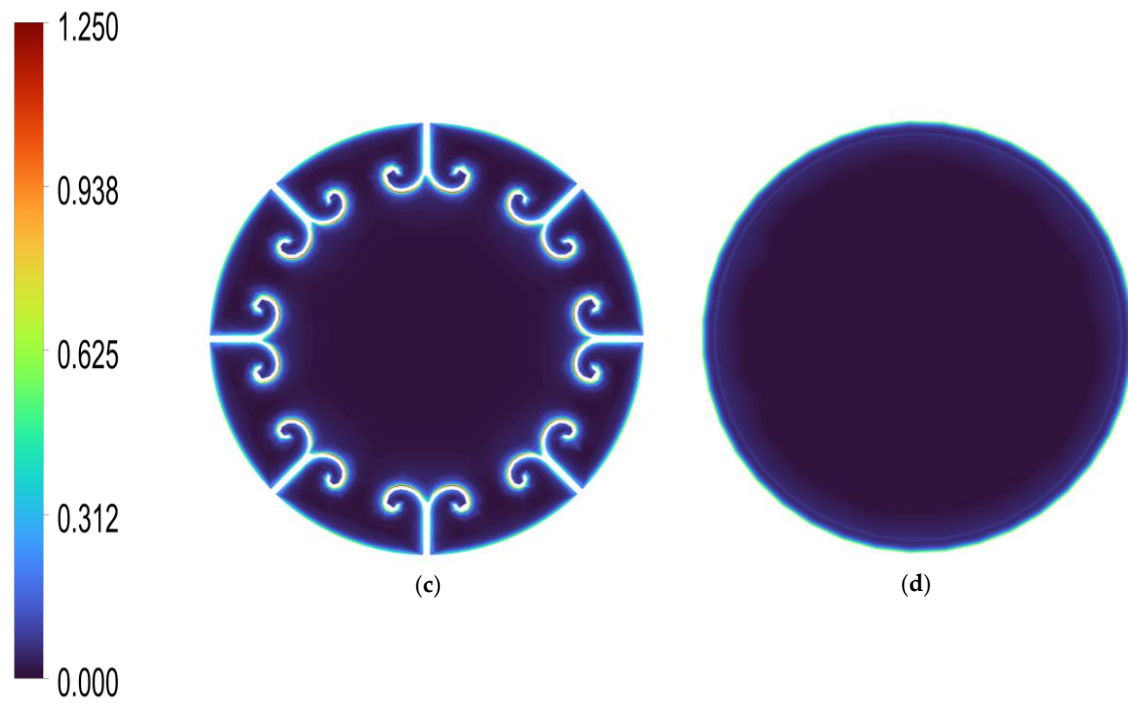


Figure 34. Transverse vorticity contours at $Re = 50,000$: (a) $N = 8$, $S/D = 0.06$; (b) $N = 8$, $S/D = 0.08$; (c) $N = 8$, $S/D = 0.1$; and (d) plain pipe.

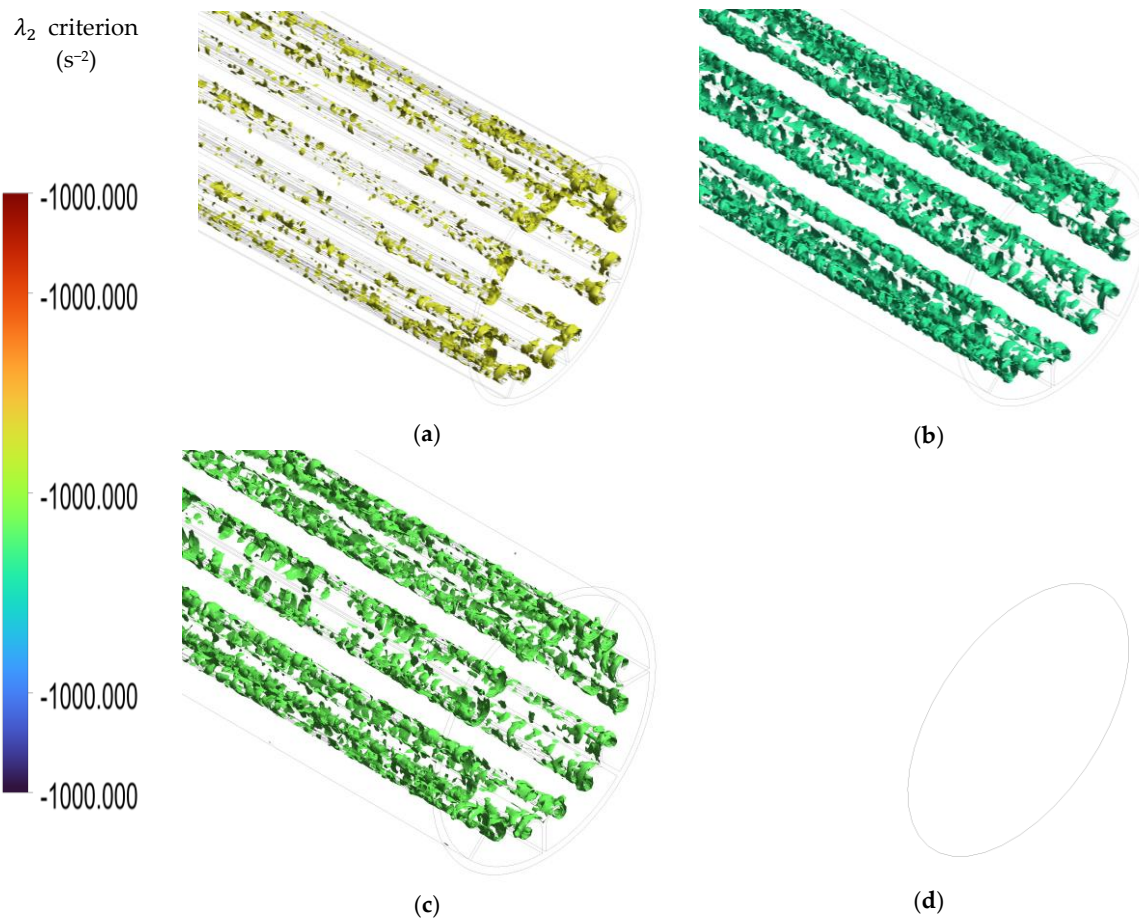


Figure 35. The -1000 s^{-2} iso-surfaces of the λ_2 criterion at $Re = 50,000$: (a) $N = 8$, $S/D = 0.06$; (b) $N = 8$, $S/D = 0.08$; (c) $N = 8$, $S/D = 0.1$; and (d) plain pipe.

3.3. Constant Pumping Power Analysis

The heat transfer enhancement at a constant pumping power was compared between the plain pipe case and all finned pipe cases examined. As the pumping power is directly proportional to fRe^3 , it follows that two flow regimes with the same value of fRe^3 have the same pumping power. For a fair comparison, graphs of Nu vs. $f^{1/3}Re$ (a scaled version of fRe^3 , considering the large Re numbers) are plotted in Figure 36 for all cases, with vertical lines through the abscissa representing constant pumping power. Most points that intersect the finned case curves at a constant pumping power show an improvement in heat transfer; for example, all cases showed improved heat transfer at $f^{1/3}Re = 20,000$, ranging from 3% ($s/D = 0.06$, $N = 8$) to 25% ($s/D = 0.1$, $N = 8$). For known pumping power values of the plain pipe case, a maximum heat transfer enhancement for all finned cases was observed at $f^{1/3}Re \approx 25,000$, where a 28% increase occurs at $s/D = 0.1$, $N = 8$. These improvements are related to several factors, such as the heat transfer mechanisms of boundary layer perturbations and vortical structures that rotate fluid particles described above. However, a few cases showed lower heat transfer than that in the plain pipe case, especially at the lower end of the constant pumping power lines. This is attributed to increased flow blockage, which may hinder heat transfer due to thermal boundary layer thickening.

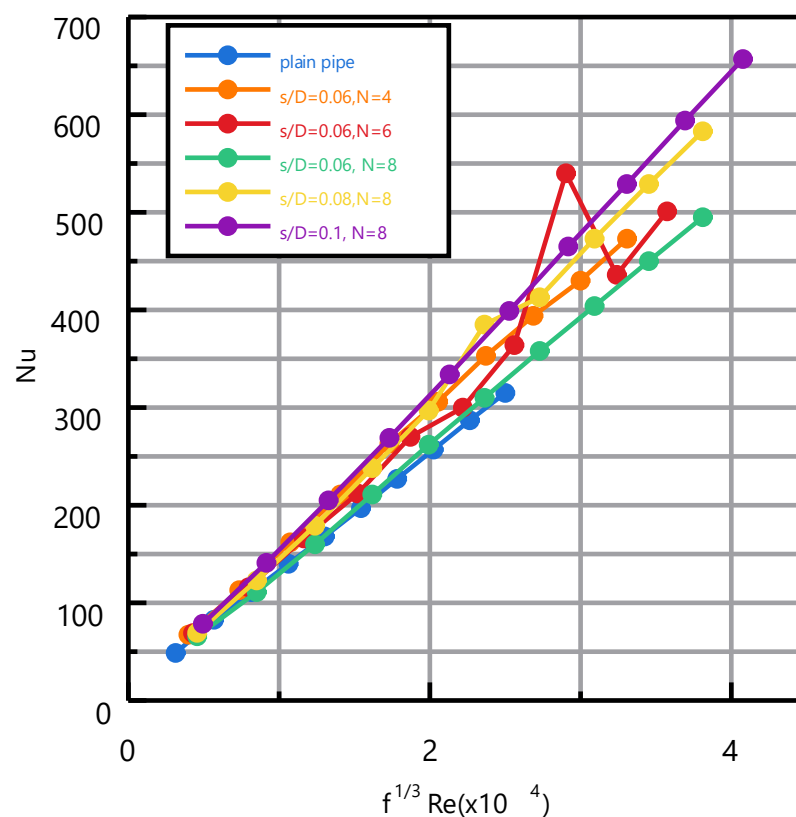


Figure 36. Nu vs. $f^{1/3}Re$.

3.4. Development of Nusselt Number Correlation

A Nusselt number correlation analysis was conducted via multiple linear regression after logarithmic transformation using MATLAB code (MATLAB 2026a), thus enabling prediction of the Nusselt number as a function of the Reynolds number (Re), Prandtl number, stem-to-height diameter ratio (s/D), and fin number (N). The data used in this prediction model included all 60 points simulated numerically; that is, 10 points stemming from the Re range for each of the six cases analyzed (plain pipe; $N = 4$, $s/D = 0.06$; $N = 6$, $s/D =$

0.06; $N = 8$, $s/D = 0.06$; $N = 8$, $s/D = 0.08$; and $N = 8$, $s/D = 0.1$). The correlation equation is given as:

$$Nu = 0.0225Re^{0.8}Pr^{0.4}\left(1 + \frac{s}{D}\right)^{6.151}(1 + N)^{0.02642} \quad (24)$$

It should be noted that Equation (24) returns the Dittus–Boelter equation in the special case of the plain pipe (i.e., $N = 0$ and $s/D = 0$). Figure 37 shows how well this model predicts the results obtained in the CFD simulation, together with $\pm 15\%$ band lines.

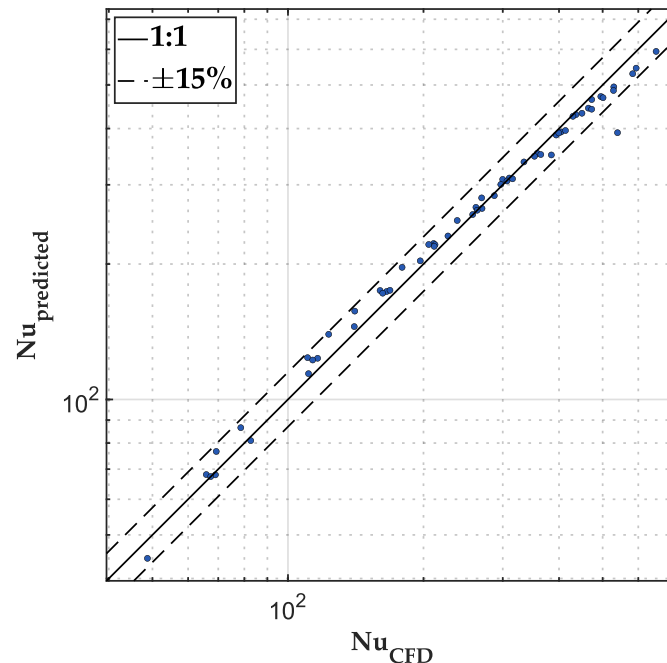


Figure 37. Predicted Nu vs. Data from CFD.

The correlation equation was found to predict the CFD data well, with most percentage errors ranging from 0.1% to 13.36%; the only outlier is the Nu value at $N = 6$, $s/D = 0.06$, $Re = 80,000$, $Pr = 2.244$, where the predicted value (391.674) was within 27% of the CFD data (539.508). This equation had a goodness-of-fit R^2 value of 0.9885 with 5.02% MAPE, indicating a good fit and further corroborating the idea that Nu depends on both flow and geometric parameters. To evaluate the applicability of Equation (24) to a different internal-fin geometry, the correlation was compared against the results of Bellos et al. [37]. That study examined passive heat-transfer enhancement using simple longitudinal fins (rectangular, triangular, and semi-circular) inside a tube. The rectangular fin case, with a relative fin height that corresponds to $s/D = 0.1$ and water as the working fluid, falls within the geometric and fluid-property bounds of the present correlation. Table 10 presents the comparison for the rectangular fin configuration. Equation (24) reproduces the reported Nusselt numbers with percentage differences ranging from 2.06% to 27.27% over $13,000 \leq Re \leq 55,000$. Agreement improves at the higher end of the range. The authors do not claim that Equation (24) can be applied indiscriminately to arbitrary internal fin geometries. The present comparison indicates only that the correlation can furnish a useful first-order estimate of the Nusselt number for longitudinally finned tubes of comparable relative height and operating conditions.

Table 10. Comparison of Equation (24) to Nu values of rectangular finned tubes in Bellos et al. [37].

Re	Pr	s/D	N	Nu Predicted	Nu (Bellos et al.)	% Error
13,000	2	0.1	8	110.55	152	27.27%
17,000	2	0.1	8	137.02	181	24.3%
48,000	2	0.1	8	314.34	308	2.06%
55,000	2	0.1	8	350.51	325	7.85%

4. Conclusions

This numerical investigation revealed that, when implemented in circular tubes, the proposed “ram-horn”-inspired internal fins, whose radii follow the Fibonacci spiral, provide thermohydraulic advantages at Reynolds numbers in the range of 10,000–100,000. Using 750 mm-long copper tubes with a 25 mm internal diameter, the effects of two geometric parameters of these fins—namely, the fin number and fin height (characterized by the s/D ratio)—on the Nu, f , TPF, and heat retention rate values were assessed. Three fin numbers ($N = 4, 6, 8$) at $s/D = 0.06$ and three stem-to-diameter ratios ($s/D = 0.06, 0.08, 0.1$) at $N = 8$ were studied over the considered Re range. The operating conditions under which this investigation was conducted were as follows: water at 350 K was used as the working fluid, with a pressure of 4 bar at the inlet, and an external wall heat flux of 50,000 W/m² was applied uniformly. The key findings of this investigation are detailed below.

1. The Nusselt number enhancements with the finned tubes reached 110% ($N = 6$, $s/D = 0.06$, $Re = 80,000$) relative to the plain tube. A Nusselt number correlation equation was developed by fitting a curve to all 60 simulated data points, which indicated that heat transfer enhancement is dependent on the geometric parameters. This equation is valid for $10,000 \leq Re \leq 100,000$, $1.70 < Pr < 2.3$, $0 \leq \frac{s}{D} \leq 0.1$, and $0 \leq N \leq 8$ and predicted the CFD data accurately with a mean absolute percentage error of 5.02%; it also predicted an outlier (Nu at $N = 6$, $s/D = 0.06$, $Re = 80,000$) with 73% accuracy. This correlation could also serve as a first-order estimate for similar design operating conditions, based on comparison with existing literature Nu values.

2. The thermohydraulic performance factor values ranged between 0.89 and 1.46 for all cases examined, with the maximum TPF of 1.46 occurring at $N = 6$, $s/D = 0.06$, and $Re = 80,000$. In all cases satisfying $TPF > 1$, the pressure drop penalty is more than offset by the heat transfer gain.

3. The heat retention ratio improved by an average of 14% when fins were added, with better heat retention particularly observed at the lower end of the Reynolds number range.

4. Constant pumping power analyses show a maximum Nu improvement of 28% when all finned cases were examined.

5. Post-processed velocity, turbulent kinetic energy, vorticity, and λ_2 -criterion field analyses confirmed that the multi-radius horns generate coherent secondary flows and localized vortex cores that persistently disrupt the thermal boundary layer.

The significant contributions of our work lie in the robust CFD approach focused on passive heat transfer enhancement techniques and the novel geometry, which introduces special mechanisms of heat transfer enhancement via boundary layer perturbation and vortex generation. This bio-inspired geometry offers a practical passive strategy for improving energy recovery in high-Reynolds-number tubular heat exchangers. The key limitations of our investigation include a lack of experimental validation and the use of a single working fluid (water), which narrowed the range of applicable Prandtl numbers to the developed correlation equation; however, as water is commonly applied in relevant

industrial settings, the results can still be considered useful. Future work may include incorporating this tube into an actual heat exchanger and performing experimental studies with a wide range of working fluids.

Author Contributions: Conceptualization, A.A.-M. and J.P.K.; methodology, A.A.-M.; validation, A.A.-M., K.Y.A., and Y.T.; writing—original draft preparation, A.A.-M. and K.Y.A.; writing—review and editing, Y.T.; supervision, J.P.K. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by The Engineering Research Center (ERC) from the Department of Mechanical Engineering at North Carolina Agricultural and Technical State University.

Data Availability Statement: All data files are available upon request.

Acknowledgments: During the preparation of this manuscript/study, the authors used ChatGPT-5 to generate the image of the ram and Grammarly for text editing. The authors have reviewed and edited the output and take full responsibility for the content of this publication.

Conflicts of Interest: The authors declare no conflicts of interest.

Abbreviations

The following abbreviations are used in this manuscript:

API	Aerothermal Performance Index
CFD	Computational Fluid Dynamics
HVAC	Heating, Ventilation, and Air Conditioning
k- ϵ	k-epsilon turbulence model
MAPE	Mean Absolute Percentage Error
NACA	National Advisory Committee for Aeronautics
Nu	Nusselt number
λ_2 -criterion	Second invariant of the velocity gradient tensor (vortex identification criterion)
RANS	Reynolds-Averaged Navier–Stokes
Re	Reynolds number
s/D	Stem height-to-diameter ratio
SST k- ω	Shear Stress Transport k-omega turbulence model
TEF	Thermal Enhancement Factor
TKE	Turbulent Kinetic Energy
TPF	Thermohydraulic Performance Factor (also referred to as Thermal Performance Factor)
TVR	Turbulent Viscosity Ratio

Symbols

The following symbols were used in the manuscript

Symbol	Description	Unit
A_s	Wetted surface area of the inner wall (including fins)	m ²
β^*	Destruction coefficient for Turbulent kinetic energy (k- ω SST model)	
β	Destruction coefficient for turbulent dissipation rate (k- ω SST model)	
γ	Blended production coefficient for turbulent dissipation rate	
$C_{1\epsilon}$	Production coefficient (realizable k- ϵ model)	
C_2	Destruction coefficient (realizable k- ϵ model)	

C_{b1}	Production coefficient (Spalart–Allmaras Model)	
C_{b2}	Diffusion coefficient Spalart–Almaras Model)	
C_w	Destruction coefficient (Spalart–Allmaras Model)	
d	Distance from local point to nearest wall	m
D	Pipe inner diameter	m
E	Internal energy	Joules
ε	Turbulent dissipation rate	Kg s^{-3}
f	Friction factor	
F_1	First blending function	
f_{t2}	Trip function (Spalart–Allmaras Model)	
f_w	Wall destruction function	
h	Convective heat transfer coefficient	$\text{W m}^{-2} \text{K}^{-1}$
k	Thermal conductivity of the fluid	$\text{W m}^{-1} \text{K}^{-1}$
k_{eff}	Effective thermal conductivity of the fluid	$\text{W m}^{-1} \text{K}^{-1}$
k	Turbulent kinetic energy	$\text{m}^2 \text{s}^{-2}$
κ	Von Karman constant	
L	Length of the tube	m
$\dot{m}$	Mass flow rate	kg s^{-1}
M	Number of data points	
μ	Dynamic viscosity	$\text{Kg m}^{-1} \text{s}^{-1}$
μ_t	Turbulent viscosity	$\text{Kg m}^{-1} \text{s}^{-1}$
N	Number of fins	–
$\bar{p}$	Pressure	Pa
P_k	Production of turbulent kinetic energy	
ϕ	Data for calculating percentage error (Nu or f)	
$\dot{Q}$	Heat retention rate	W
q''	Wall heat flux	W m^{-2}
ρ	Density	Kg m^{-3}
R_1, R_2, R_3, R_4	Radii of curvature of the successive horn branches (Fibonacci sequence)	m
σ_k	Turbulent Prandtl number for turbulent kinetic energy	
σ_ε	Turbulent Prandtl number for turbulent dissipation rate (realizable k- ε model)	
$\sigma_{\omega 1}, \sigma_{\omega 2}$	Turbulent Prandtl number for turbulent kinetic energy and dissipation rate (realizable k- ε model)	
$\tilde{S}$	Modified vorticity magnitude	s^{-1}
S	Mean strain rate	s^{-1}
s	Stem height of the fin (from inner wall to base of horns)	m
t	Fin stem thickness	m
τ	Shear stress	Pa
T	Temperature	K
T_{avg}	Average fluid temperature (arithmetic mean of inlet and outlet)	K
T_w	Inner wall temperature	K
$\bar{u}_r$	Mean radial velocity	ms^{-1}
$\bar{u}_\theta$	Mean azimuthal velocity	ms^{-1}
$\bar{u}_z$	Mean axial velocity	ms^{-1}
V	Mean bulk velocity or fluid-domain volume (context-dependent)	m s^{-1} or m^3
x	Axial coordinate	m
y^+	Dimensionless wall distance	–
ω	Turbulent dissipation rate (k- ω SST model)	Kg s^{-3}

References

1. Admin TLG. Energy Recovery via Industrial Heat Exchangers: The Future Path. 2026. Available online: <https://shipand-shore.com/energy-recovery-via-industrial-heat-exchangers/> (accessed on 10 April 2026).
2. Liu, Y.; Xu, G.; Fu, Y.; Wen, J.; Huang, H. Thermal dynamic and failure research on an air-fuel heat exchanger for aero-engine cooling. *Case Stud. Therm. Eng.* **2023**, *42*, 102715. <https://doi.org/10.1016/j.csite.2023.102715>.
3. Zhao, S.; Duan, F.; Tian, Z.; Huang, H.; Wang, J.; Wang, C. Comprehensive analysis of a novel coal-fired cogeneration system with integrated dual heat pumps. *Case Stud. Therm. Eng.* **2025**, *73*, 106413. <https://doi.org/10.1016/j.csite.2025.106413>.
4. Chandran, S.S.R.; Roy, K.E.R.; Barik, D.; Arun, M.; Pullagura, G.; Rout, S.K.; Dennison, M.S. Empirical correlation to analyze performance of shell and tube heat exchanger using TiO₂ Nanofluid-DI water in solar water heater. *Case Stud. Therm. Eng.* **2024**, *60*, 104652. <https://doi.org/10.1016/j.csite.2024.104652>.
5. Mousa, M.H.; Miljkovic, N.; Nawaz, K. Review of heat transfer enhancement techniques for single phase flows. *Renew. Sustain. Energy Rev.* **2021**, *137*, 110566. <https://doi.org/10.1016/j.rser.2020.110566>.
6. Marzouk, S.A.; Al-Sood, M.M.A.; El-Said, E.M.S.; Younes, M.M.; El-Fakharany, M.K. A comprehensive review of methods of heat transfer enhancement in shell and tube heat exchangers. *J. Therm. Anal. Calorim.* **2023**, *148*, 7539–7578. <https://doi.org/10.1007/s10973-023-12265-3>.
7. Al-Chlahawi, K.K.I.; Hasan, M.R.; Ekaid, A.L. Compound passive heat transfer augmentation techniques: A comprehensive review. *Heat Transf.* **2023**, *53*, 363–421. <https://doi.org/10.1002/htj.22955>.
8. Khan, M.A.; Ilyas, M.; Waheed, K.; Haq, I.; Aydogan, F. Advances in passive heat transfer enhancement for heat exchangers: A comprehensive review. *J. Therm. Eng.* **2025**, *11*, 1193–1230. <https://doi.org/10.14744/thermal.0000972>.
9. Diwaker, M.K.; Kumar, A. *An Updated Review of Heat Transfer Enhancement Techniques in Tube-Type Heat Exchangers*; Springer Nature: Berlin/Heidelberg, Germany, 2024; pp. 65–78. https://doi.org/10.1007/978-981-99-7827-4_6.
10. Yildirim, O.; Karagöz, Ş.; Abdi, H. Numerical Investigation of the Effects of Different Turbulators on Heat Transfer in an Internal Flow System. *Int. J. Innov. Res. Rev.* **2025**, *9*, 59–66.
11. Lotfi, B.; Zeng, M.; Sundén, B.; Wang, Q. Thermo-hydraulic Characterization of the Smooth Wavy Fin-and-elliptical Tube Heat Exchangers Using New Type Vortex Generators. *Energy Procedia* **2014**, *61*, 2343–2346. <https://doi.org/10.1016/j.egypro.2014.11.1199>.
12. Jin, Z.; Chen, F.; Gao, Z.; Gao, X.; Qian, J. Effects of pitch and corrugation depth on heat transfer characteristics in six-start spirally corrugated tube. *Int. J. Heat Mass Transf.* **2017**, *108*, 1011–1025. <https://doi.org/10.1016/j.ijheatmasstransfer.2016.12.091>.
13. Abotsi, O.Y.W.; Kizito, J.P. Numerical study of heat transfer augmentation in an axially rotating pipe equipped with Kenics mixer. *Case Stud. Therm. Eng.* **2020**, *21*, 100695. <https://doi.org/10.1016/j.csite.2020.100695>.
14. Samutpraphut, B.; Eiamsa-Ard, S.; Chuwattanakul, V.; Thianpong, C.; Maruyama, N.; Hirota, M. Influence of sawtooth twisted tape on thermal enhancement of heat exchanger tube. *Energy Rep.* **2023**, *9*, 696–703. <https://doi.org/10.1016/j.egy.2022.11.029>.
15. Abdulhamed, A.J.; Al-Akam, A.; Khudhayer, W.J.; Allw, A.S. Tubular Heat Enhancement Using Twisted Tape Inserts with Large Holes. *Energy Eng.* **2024**, *121*, 273–290. <https://doi.org/10.32604/ee.2023.045583>.
16. Thianpong, C.; Wongcharee, K.; Kunrak, K.; Chokphoemphun, S.; Chamoli, S.; Eiamsa-Ard, S. Parametric study on thermal performance augmentation of TiO₂/water nanofluids flowing a tube contained with dual counter twisted-tapes. *Case Stud. Therm. Eng.* **2024**, *59*, 104471. <https://doi.org/10.1016/j.csite.2024.104471>.
17. Kadam, S.; Kore, A.N.; Sawant, S.; Kadam, H.; Mohite, S. Numerical Analysis of Helical Coil-Inserted Tubes Based on Thermo-hydraulic Performance and Development of Correlations for Heat and Fluid Flow. *Heat Transf.* **2025**, *55*, 1442–1457. <https://doi.org/10.1002/htj.70143>.
18. Pastuszko, R.; Piasecka, M. Pool boiling on surfaces with mini-fins and micro-cavities. *J. Phys. Conf. Ser.* **2012**, *395*, 012137. <https://doi.org/10.1088/1742-6596/395/1/012137>.
19. El-Sebaey, M.S.; El-Din, S.S.; El-Kholy, M.K. Heat transfer and fluid flow performance of an internally longitudinal finned tube: Numerical study and experimental validation. *Int. J. Therm. Sci.* **2024**, *201*, 109025. <https://doi.org/10.1016/j.ijthermalsci.2024.109025>.
20. Marzouk, S.; Awjah Almeahadi, F.; Aljabr, A.; Alam, T.; Singh, T.; Khargotra, R. Effects of extended pin fins on the hydrothermal performance of double pipe heat exchanger. *Therm. Sci. Eng. Prog.* **2024**, *55*, 102915. <https://doi.org/10.1016/j.tsep.2024.102915>.
21. Mameli, M.; Marengo, M.; Khandekar, S. Local heat transfer measurement and thermo-fluid characterization of a pulsating heat pipe. *Int. J. Therm. Sci.* **2014**, *75*, 140–152. <https://doi.org/10.1016/j.ijthermalsci.2013.07.025>.

22. Hurry, A.S.; Hayward, K.; Guzzomi, F.; Rauthan, K.; Vafadar, A. Thermo-hydraulic performance evaluation of a NACA 63-015 heat exchanger with shark denticles as surface textures. *Int. J. Heat Mass Transf.* **2023**, *216*, 124591. <https://doi.org/10.1016/j.ijheat-masstransfer.2023.124591>.
23. Hurry, A.S.; Vafadar, A.; Hayward, K.; Guzzomi, F.; Rauthan, K. Numerical investigation of the thermo-hydraulic performance of a shark denticle-inspired plate fin heat exchanger. *Appl. Therm. Eng.* **2024**, *239*, 122192. <https://doi.org/10.1016/j.applthermaleng.2023.122192>.
24. Wang, Z.; Yang, X.; Gu, X.; Liu, Y. A Drag-Reduction Mechanism of Seagull-Inspired Curved Vortex Generators Integrating Response Surface Method and Genetic Algorithms Optimization in Compact Heat Exchangers. *Energies* **2025**, *18*, 4281. <https://doi.org/10.3390/en18164281>.
25. Li, Y.; Yang, Y.; Gong, L.; Cui, W.; Yang, B. Design of biomimetic wheat-grain-shaped pin fins for enhanced heat transfer in microchannel heat sink. *Appl. Therm. Eng.* **2026**, *284*, 129074. <https://doi.org/10.1016/j.applthermaleng.2025.129074>.
26. Liu, Y.; Chen, Y.; Zhang, M.; Zhao, X.; Li, T. Enhanced thermal performance of shell and tube phase change heat accumulator with novel biomimetic spider web fins. *Appl. Therm. Eng.* **2023**, *233*, 121224. <https://doi.org/10.1016/j.applthermaleng.2023.121224>.
27. Yao, J.; Zhang, Z.; Saari, J.; Wang, J.; Čuček, L.; Zheng, D. Structure optimization of intercooler bionic fins based on artificial neural network and genetic algorithms. *Energy* **2024**, *307*, 132615. <https://doi.org/10.1016/j.energy.2024.132615>.
28. Cabeza, L.F.; Mani Kala, S.; Zsembinski, G.; Vérez, D.; Risco Amigó, S.; Borri, E. Development of a Bio-Inspired TES Tank for Heat Transfer Enhancement in Latent Heat Thermal Energy Storage Systems. *Appl. Sci.* **2024**, *14*, 2940. <https://doi.org/10.3390/app14072940>.
29. Sun, S.; Moreira, T.A.; Rankouhi, B.; Yu, X.; Jentz, I.W.; Thoma, D.J.; Anderson, M.H.; Qian, X. Topology optimization, additive manufacturing and thermohydraulic testing of high-temperature heat exchangers. *Int. J. Heat Mass Transf.* **2025**, *242*, 126809. <https://doi.org/10.1016/j.ijheatmasstransfer.2025.126809>.
30. Sultan, A.A.; Mostafa, H.M.; Alboz, A.M.; Abdelhady, S.S. Experimental Investigation of Air Humidification Through Convergent and Divergent Rectangular Duct. *Mansoura Eng. J.* **2020**, *38*, 1–13. <https://doi.org/10.21608/bfemu.2020.115626>.
31. Spalart, P.R.; Allmaras, S.R. A one-equation turbulence model for aerodynamic flows. In Proceedings of the 30th Aerospace Sciences Meeting and Exhibit, Reno, NV, USA, 6–9 January 1992.
32. Shih, T.-H.; Liou, W.W.; Shabbir, A.; Yang, Z.; Zhu, J. A new K- ϵ Eddy Viscosity Model For High Reynolds Number Turbulent Flows. *Comput. Fluids* **1995**, *24*, 227–238. [https://doi.org/10.1016/0045-7930\(94\)00032-t](https://doi.org/10.1016/0045-7930(94)00032-t).
33. Menter, F.R. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. *AIAA J.* **1994**, *32*, 1598–1605. <https://doi.org/10.2514/3.12149>.
34. Petukhov, B.S. Heat Transfer and Friction in Turbulent Pipe Flow with Variable Physical Properties. *Adv. Heat Transf.* **1970**, *6*, 503–564.
35. Dittus, F.W.; Boelter, L.M.K. Heat Transfer in Automobile Radiators of the Tubular Type. *Univ. Calif. Publ. Eng.* **1930**, *2*, 443–461.
36. Jeong, J.; Hussain, F. On the identification of a vortex. *J. Fluid Mech.* **1995**, *285*, 69–94.
37. Bellos, E.; Lykas, P.; Tzivanidis, C. Heat and Flow Study of the Internally Finned Tubes with Different Fin Geometries. *Appl. Syst. Innov.* **2022**, *5*, 50. <https://doi.org/10.3390/asi5030050>.

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.