



ARTICLE

Turbulent Flow and Thermal-Hydrodynamic Optimization in Evaporator Tubes with Transverse Partitions

Omar Ghoulam¹, Hind Talbi^{1,2}, Kamal Amghar¹, Hamza Faraji^{3,*}, Saloua Senhaji⁴ and Ismael Driouch¹

¹Experimentation and Modeling in Mechanics and Energy Systems Team, Department of Civil, Energy and Environmental Engineering, National School of Applied Sciences, Abdelmalek Essaâdi University, Al Hoceima, Morocco

²École Supérieure d'Ingénierie Appliquée et Innovation Privée (ESIAI), Oujda, Morocco

³LaRTID Laboratory, National School of Applied Sciences, Cadi Ayyad University, UCA, Marrakech, Morocco

⁴Doctoral Studies Center (CEDoc), Sciences, Engineering and Sustainable Development (SIDD), Private University of Fes (UPF), Lotissement Quaraouiye Route Ain Chkef, Fes, Morocco

*Corresponding Author: Hamza Faraji. Email: hamza.faraji@uca.ac.ma

Received: 27 November 2025; Accepted: 26 January 2026; Published: 12 July 2026

ABSTRACT: This study numerically investigates turbulent flow and thermal performance in evaporator tubes equipped with rectangular partitions positioned at different locations. Two configurations are analyzed: (A) partitions on the top wall, center of channel, and bottom wall, and (B) partitions on the bottom wall, center of channel, and top wall. In addition, we examine the effect of varying the positions of the obstacles ($S = \frac{D}{2}$, $S = D$, $S = \frac{5D}{4}$, and $S = 3D/2$) and the inclination angle ($\theta = 60^\circ$, $\theta = 75^\circ$, $\theta = 90^\circ$, $\theta = 105^\circ$ and $\theta = 120^\circ$) of the detached obstacle relative to the walls, an innovative aspect that had not been addressed in previous studies. Using computational fluid dynamics (CFD), heat transfer and hydrodynamic behavior are evaluated under steady-state conditions for Reynolds numbers ranging from 10,000 to 30,000. Results show that configuration A enhances dynamic pressure and Nusselt number while reducing friction, yielding a thermal performance enhancement factor (TEF) greater than 1 across all cases. The originality of this work lies in the comparative evaluation of partition positioning and inclination, demonstrating that optimized geometries can significantly improve heat transfer efficiency while minimizing flow resistance compared to conventional evaporators.

KEYWORDS: Thermal performance; numerical simulation; transverse partition; partitions

1 Introduction

Turbulent forced convection is a crucial mechanism for efficient and effective heat transfer in various industrial sectors. Its prevalence is notable in critical applications such as heat exchangers, nuclear reactors, turbomachinery cooling systems, and solar thermal converters. Optimizing the performance of these systems, vital for energy consumption and sustainability, requires a thorough understanding of thermofluidic phenomena.

Faced with the growing demand for energy efficiency, heat transfer enhancement (HTE) has become a major research focus. In this context, the integration of passive devices, such as fins and baffles, represents a particularly promising strategy in heat exchanger design. These structural elements are designed to modify the flow path, increase turbulence, and promote better fluid mixing, thus significantly enhancing heat exchange. As a result, these innovative solutions have attracted considerable interest and have been the

subject of numerous experimental and numerical studies aimed at precisely characterizing their impact on the overall performance of heat transfer systems.

Many previous studies [1–6] have demonstrated that the integration of obstacles in flow systems significantly improves the efficiency of heat transfer. This work has notably highlighted the interest of devices such as deflectors (or turbulators), installed in ducts, to intensify heat exchanges. Their presence has a double effect: they increase the exchange surface and, more importantly, they generate a disturbance of the main flow, thus promoting turbulence and increased mixing of the fluid. This intensification of turbulence is the key mechanism by which these obstacles contribute to increased thermal performance of the system.

In literature, several pioneering and recent studies have contributed to the understanding of these phenomena. Patankar et al. [7] notably made a fundamental contribution by publishing the first numerical analysis of forced convection flows in a duct, thus introducing the novel concept of periodic flow. Continuing in this direction, more recently, Chandra et al. [8] performed a detailed numerical study of heat exchange in a turbulent air flow through a duct equipped with transverse baffles. Their results revealed that a two-baffle configuration, positioned opposite each other, could increase the heat exchange rate by up to 6% compared to single-baffle housing, highlighting the importance of geometric arrangement of the obstacles. In a similar optimization perspective, Chand et al. [9] innovated by studying the impact of perforated barriers on the thermal performance of airflow-based solar heating systems. Their approach consisted of fixing these barriers with openings on the underside of the absorber plate, to guide and accelerate the heat transfer by precise control of the airflow. The results of their study revealed a direct correlation between thermal efficiency and the mass flow rate. Specifically, for a barrier separation distance of 2 cm, the maximum thermal efficiency reached 70% at a flow rate of 0.0158 kg/s, while an outlet temperature of 58.66°C was recorded at 0.007 kg/s. These observations highlight the potential of perforated barrier configurations to improve thermal performance. Other recent works have also explored thermal performance optimization. Mohit and Gupta [10] conducted an extensive numerical simulation to evaluate the impact of fin height (ranging from 0.2 to 1.0 mm) on heat transfer within a microchannel. Their study, carried out under a constant heat flux at the base and for a range of Reynolds numbers (200–1000), demonstrated that an increase in both fin height and Reynolds number leads to a significant improvement in heat transfer. Meanwhile, Boonloi and Jedsadaratanachai [11] evaluated the thermal and fluid performance of a square-section channel, testing six distinct barrier configurations. Their research focused on laminar flows (Reynolds numbers ranging from 100 to 2000), examining the influence of key parameters such as blockage rates, barrier dimensions and shapes, and different flow directions.

In the same line of geometric optimization, Sriromreun et al. [12] conducted a specific study to determine the optimal thermal performance in a rectangular channel for Reynolds numbers ranging from 4400 to 20,400. Their work focused on evaluating the effects of size and separation length of Z-type barriers. In particular, they compared in-phase and out-of-phase 45° inclined Z-type barriers. In-phase configurations were shown to offer significantly higher heat transfer coefficients, friction coefficients, and thermal performance enhancement factor (TEF) than out-of-phase configurations under comparable operating conditions. Moreover, within in-phase configurations, a larger barrier height resulted in increased heat transfer and frictional pressure drop. Interestingly, despite this pressure drop, a shorter separation length achieved an overall higher thermodynamic performance than a wider separation. Other notable research has deepened the understanding of heat transfer enhancement in various contexts. Following on from this research, Handoyo et al. [13] further analyzed the thermo-aerodynamic behavior of a V-shaped air tube, an integral part of a solar heating system. Their study specifically examined the impact of obstacles placed at different distances along the tube, seeking to optimize fluid dynamics and heat exchange. Similarly, Peng et al. [14] introduced a novel solar air collector configuration. Their innovation lies in the integration of fins

on the absorber surface, a strategy designed to significantly improve the thermal performance of the entire system. Beyond the previously mentioned studies, another fundamental research has also enriched the field. Kumar and Kim [15] conducted a comprehensive review study, specifically analyzing the impact of various devices, such as obstacles, on the performance of solar tubes. From a geometric optimization perspective, Promvonge et al. [16] numerically characterized the performance of inclined baffles in a 3D rectangular duct, exploring the effect of blocking ratios ranging from 0.1 to 0.5 on heat transport and pressure drop, and comparing these results to a vertical obstacle configuration. Mokhtari et al. [17] used the finite volume method to simulate mixed heat convection in a 3D tube, studying the influence of different fin positions. As for Mousavi and Hooman [18], they described the convective heat transfer and the control of a laminar flow in the inlet section of a 2D duct incorporating wall obstacles. In the context of turbulent flows, Yongsiri et al. [19] provided numerical data on the heat transfer through a tube equipped with inclined ribs with detached geometries. Finally, more targeted studies include the work of Karwa and Maheshwari [20] on two distinct cases of perforated obstacles (totally or partially), and the simulation by Kamali and Binesh [21] of the thermo-aerodynamic behavior in a square duct with various geometric ribs arranged on the same surface. Other notable contributions, exploring the simulation of fluid flows and heat transfer under different conditions, are also worth noting [22–25]. More specific investigations have continued to shed light on the mechanisms of heat transfer intensification. Bazdidi-Tehrani and Naderi-Abadi [26] performed a numerical analysis of the thermal and dynamic behavior of a fluid flowing through a baffled conduit. Their findings indicated that baffle obstacles retain some effectiveness even at high blocking rates, which contradicts some expectations. Similarly, Tsay et al. [27] conducted a numerical study aimed at enhancing heat transfer in a channel equipped with a vertical baffle. For Reynolds numbers between 100 and 500, they extensively examined the effects of baffle size and setbacks on the flow structure. In an experimental setting, Wilfried [28] explored the crucial impact of the distance between the baffle and the casing, as well as the spacing between the baffles themselves, on the thermal performance of a tubular heat exchanger. Complementing this work, Ko and Anand [29] conducted a novel experimental study to estimate the average heat transfer coefficients in a rectangular channel specifically equipped with porous baffles. Their findings highlighted the effectiveness of this configuration compared to a channel without baffles. In the context of innovative configurations, Karwa et al. [30] conducted an analysis of heat transport and friction in a rectangular channel characterized by perforated baffles and asymmetry. Their results showed a significant improvement in the Nusselt number, ranging from 73.7% to 82.7% compared to a solid baffle duct for specific heights. In addition, Liu et al. [31] conducted a comprehensive numerical study on turbulent flow in a rectangular channel equipped with cylindrical grooves. Their work demonstrated that these grooves improve heat transfer in a comparable manner to square ribs. However, they also highlighted that the pressure drop penalty associated with square ribs is significantly higher than that of cylindrical grooves, suggesting a better compromise for the latter. In a similar heat transfer optimization approach, Ghoulam et al. [32] numerically investigated the effectiveness of flat and diamond-shaped obstacles along a channel. By evaluating overall performance using the thermal performance factor, their results, for Reynolds numbers ranging from 20,000 to 35,000, highlighted the superiority of diamond-shaped obstacles. The latter exhibited a performance factor of up to 1.65, thus outperforming rectangular obstacles. Furthermore, the same author [33] specifically examined the effect of baffle spacing on improving heat transfer in a channel. By testing three different spacings ($d_1 = 0.051$ m, $d_2 = 0.101$ m, and $d_3 = 0.152$ m), this study concluded that the performance factor reached its maximum value for an intermediate.

More numerical investigations of convective flows in baffled and finned heat exchanger channels are recently documented in the literature (Kitayama et al. [34], Ismael [35], Mo et al. [36], Khadang et al. [37],

Amghar et al. [38], Bi et al. [39], Yaseen and Ismael [40], Mazdak et al. [41], Maouedj et al. [42], Ahmad et al. [43], Saravanakumar et al. [44], Peiravi and Alinejad [45], and Ghoulam et al. [32]) as detailed in Table 1.

Table 1: Compilation of several numerical analyses pertaining to finned-channel heat exchangers.

Author(s)	Year	Type Study	Findings
Kitayama et al.	2022	Numerical	Sequential approximate optimization improved baffle configurations in heat exchanger headers, enhancing flow distribution and thermal performance.
Ismael	2022	Numerical	Forced convection in compliant channels with alternated baffles enhanced heat transfer efficiency under turbulent flow.
Mo, Li, Yuan	2023	Numerical	Optimized baffle parameters in hydrogen/air micro-combustor improved wall temperature uniformity and radiation efficiency.
Khadang et al.	2024	Experimental	Baffles combined with nanofluids significantly enhanced thermal performance in double-pipe heat exchangers; neural networks accurately predicted outcomes.
Amghar et al.	2021	Numerical	Numerical study of thermal and dynamic structure of a laminar flow in solar collector.
Bi et al.	2022	Numerical	Optimal cylindrical baffle array layout reduced rock avalanche energy via spacing and arrangement optimization.
Yaseen, Ismael	2023	Numerical	Flexible baffles in trapezoidal cavities enhanced heat transfer in non-Newtonian fluids by deformation-induced mixing.
Mazdak et al.	2023	Numerical	Curved segmental baffles with Cassini oval tubes improved shell-and-tube heat exchanger thermal performance.
Maouedj et al.	2021	Numerical	Vortex generators enhanced turbulent hydrothermal performance of oil/MWCNT nanofluid in solar channel heat exchangers.
Ahmad et al.	2021	Numerical	New solutions for excitable systems using modified expansion methods improved nonlinear system modeling.
Saravanakumar et al.	2020	Experimental	Arc-shaped ribs with fins and baffles optimized exergetic performance in solar air heaters, improving efficiency.
Peiravi, Alinejad	2020	Numerical	Rectangular cylinders (as baffles/obstacles) improved hybrid heat transfer modes (conduction, convection, radiation) in channels.

(Continued)

Table 1 (continued)

Author(s)	Year	Type Study	Findings
Ghoulam et al.	2025	Numerical	Detached obstacles (baffle-like) enhanced turbulent heat transfer in heat exchanger ducts through flow separation and mixing.

The present study thus aims to fill a gap identified in the literature by proposing an in-depth numerical analysis of the thermal and dynamic characteristics of a turbulent air flow within a specifically configured rectangular channel. This innovative configuration integrates not only two vertically placed and oppositely oriented rectangular baffles, but also an additional rectangular obstacle positioned in the middle of the channel, between these two baffles. In addition, we investigate the influence of obstacle positioning ($S = D/2$, $S = D$, $S = 5D/4$, and $S = 3D/2$), and the inclination angle ($\theta = 60^\circ$, 75° , 90° , 105° , and 120°) of the detached obstacle relative to the walls. This systematic parametric study represents an innovative aspect of our work, as such variations have not been previously addressed. The main objective is to understand how this combined geometry of the baffles and the central obstacle significantly improve the interaction and mixing between the high-temperature zones and the circulating fluid, thus optimizing the heat transfer. To achieve this, the hydrodynamic parameters of the flow (such as velocity profiles and recirculation zones) and the spatial temperature distribution, as well as the overall exergy destruction in the channel, were rigorously examined for a diverse range of Reynolds numbers. The results of this work will offer valuable insights for the design of more efficient and energy-efficient heat exchangers.

2 Computational Procedure

2.1 Numerical Model and Geometric Description

The modeling of this study builds on previous research by Demartini et al. [46], focusing on a two-dimensional computational domain. The channel under study has a rectangular cross-section, and its design aims to optimize heat transfer through the integration of turbulator devices. This arrangement includes two rectangular baffles, mounted alternately on the upper and lower walls of the duct. An additional rectangular obstacle is strategically positioned in the center of the channel, between these two baffles. The structural design of the evaporator tube partitions incorporates both conventional practices and innovative modifications. Rectangular baffles were chosen because of their ease of manufacture and wide industrial application. This study presents different positioning strategies, including top, bottom, and center, and examines the effect of obstacle spacing and detached obstacle inclination. This investigation is distinguished by the comparative analysis of two distinct geometric configurations of this turbulator system, designated Case (A) and Case (B), respectively. These geometries are illustrated in detail in Fig. 1.

The precise dimensions of the geometric elements (channel length, baffle height, obstacle position) are detailed directly in Fig. 1.

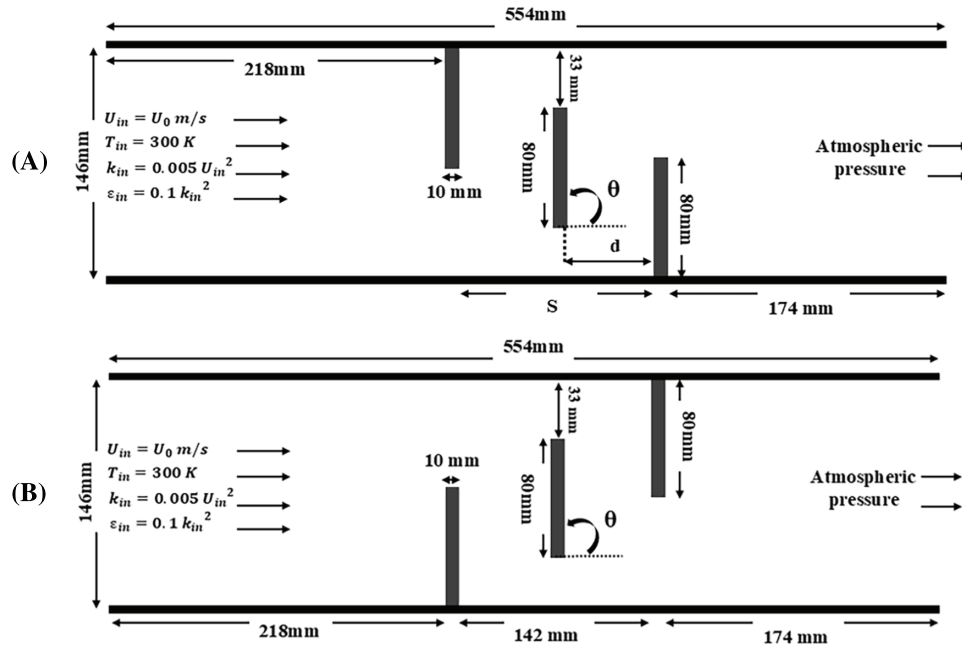


Figure 1: Geometries of computational domain for two cases: (A) and (B).

Table 2 presents the different spacings between obstacles that were applied specifically for Case (A).

Table 2: Studied spacing between baffles.

Cases	Spacing S	Values (m)	Spacing d	Values (m)
Case (a)	$S = D/2$	0.071	$d_1 = (S - 0.01) / 2$	0.0305
Case (b)	$S = D$	0.142	$d_2 = (S - 0.01) / 2$	0.066
Case (c)	$S = 5D/4$	0.1775	$d_3 = (S - 0.01) / 2$	0.08375
Case (d)	$S = 3D/2$	0.213	$d_4 = (S - 0.01) / 2$	0.1015

Table 3 provides the thermophysical properties of the channel wall material, as well as the fundamental assumptions used for numerical simulation.

Table 3: Thermal and physical properties of aluminum [47].

	Aluminum	Assumptions
Density, $\rho_s (\text{kg/m}^3)$	2719	Constant
Specific heat at constant pressure, $C_p (\text{J/kg} \cdot \text{K})$	871	Constant
Thermal conductivity, $\lambda_{Al} (\text{W/m} \cdot \text{K})$	202.4	Constant

In this computation, the study examines two main objectives. The first is based on the deflection of the airflow by deflectors, thus creating vortices in the domain. The second analysis focuses on the improvement of thermal performance by the location of the deflectors. In this investigation, the geometry and boundary conditions are based on the experimental and numerical work of Demartini et al. [46], which allows the study

of a stationary and two-dimensional turbulent air flow. The working fluid is air, whose physical properties are considered constant.

For this study, the partial differential equations governing fluid flow in turbulent and two-dimensional regimes, as well as heat transfer, are formulated according to the methodologies and principles described in references [48,49]. These equations, which include the conservation of mass, momentum, and energy, are solved in steady-state and time-averaged (Reynolds-Averaged Navier-Stokes, RANS). Regarding turbulence modeling, a crucial aspect for the accuracy of simulations, we relied on previous works. Notably, based on the results of reference [44], it was established that the $k-\epsilon$ turbulence model provides robust accuracy for the simulation of similar flows. Therefore, this model was adopted in the present work to reliably study the characteristics of turbulent flow and heat transfer within the ribbed duct.

Continuity:

$$\frac{\partial u_j}{\partial x_j} = 0 \quad (1)$$

Momentum equations:

$$\frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left((\mu + \mu_t) \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \right) \quad (2)$$

Energy:

$$\frac{\partial}{\partial x_i} (u_i T) = \frac{\partial}{\partial x_i} \left(\left(\frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right) \frac{\partial T}{\partial x_i} \right) \quad (3)$$

Turbulent Kinetic Energy:

$$u_i \frac{\partial k}{\partial x_i} = P_k - \epsilon + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (4)$$

Dissipation rate equation:

$$u_i \frac{\partial \epsilon}{\partial x_i} = \frac{c_{\epsilon 1}^* P - C_{\epsilon 2} \epsilon}{T} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] \quad (5)$$

The constant turbulent values are explicated as: $\sigma_k = 1$; $\sigma_\epsilon = 1, 3$; $C_{\epsilon 1} = 1, 4$; $C_{\epsilon 2} = 1, 9$.

2.2 Boundary Conditions

The dynamic boundary conditions used to solve the system of equations are defined based on experimental data published by Demartini et al. [46]. The Reynolds number (Re), calculated from the hydraulic diameter ($D_h = 0.167$ m), is also set at $Re = 8.73 \times 10^4$, by the same reference [46]. The channel walls and baffles are considered to be impermeable and adherent (no-slip conditions), implying zero velocities ($u = v = 0$) as well as zero values for the turbulent kinetic energy ($k = 0$) and its dissipation rate ($\epsilon = 0$) at these boundaries. On the other hand, the thermal boundary conditions are specified according to the work of Nasiruddin and Kamran Siddiqui [50].

Moreover, the boundary conditions applied for the numerical solution of the problem are detailed below and are shown below:

- At channel inlet:

$$U_{in} = U_0 \quad (6)$$

$$v = 0 \quad (7)$$

$$T_{in} = 300K \quad (8)$$

$$k_{in} = 0.005U_{in}^2 \quad (9)$$

$$\varepsilon_{in} = 0.1k_{in}^2 \quad (10)$$

- At the walls

$$u = v = 0 \quad (11)$$

$$T_w = 375K \quad (12)$$

The heat flow through the walls is zero ($\Phi = 0$).

- At channel outlet:

$$\frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = \frac{\partial k}{\partial x} = \frac{\partial \varepsilon}{\partial x} = 0 \quad (13)$$

$$P = P_{atm} \quad (14)$$

2.3 Quantity Definition and Grid Independence Study

The dimensionless quantities utilized in this analysis are described as follows:

The Reynolds number (Re) is defined as:

$$Re = \frac{\rho U D_h}{\mu} \quad (15)$$

The average skin friction coefficient is given by [51]:

$$f = \frac{2\Delta p}{L} \frac{D_h}{\rho u^2}; u = \int u_x dA \quad (16)$$

The average Nusselt number (Nu) is written as:

$$Nu = \frac{1}{L} \int \frac{h_x D_h}{\lambda} dx \quad (17)$$

And, local Nusselt number is expressed by:

$$Nu = \frac{h * D_h}{\lambda} \quad (18)$$

To evaluate the performance of our model, we must consider the friction factor and the Nusselt number along the heat exchanger. For this, the thermal performance factor (TPF) is a general term defined by the following formula [52]:

$$TPF = \frac{\frac{N_u}{Nu_0}}{\left(\frac{f}{f_0}\right)^{\frac{1}{3}}} \quad (19)$$

where: Nu_0 is the Nusselt number along the smooth channel, and f_0 is the skin friction coefficient along the smooth channel.

When the TPF parameter exceeds 1, we conclude that the overall thermal performance of the studied geometry is superior to that of a smooth channel.

To ensure the accuracy and stability of the numerical results and their independence from spatial discretization, a mesh sensitivity analysis was rigorously conducted. A structured quadrilateral mesh was generated for the computational domain, prioritizing higher element quality and optimal alignment with the flow direction. The mesh was refined progressively, with particular densification near walls and obstacles where velocity and temperature gradients are most pronounced. The mesh independence study was performed for the case of fluid flow with an inlet velocity of 7.8 m/s (corresponding to a specific Reynolds number, to be mentioned if relevant). The variables monitored to assess convergence were the maximum axial velocity ($U_{\max}$) and the maximum dynamic pressure ($Pd_{\max}$) within the channel. The obtained results are given in Fig. 2.

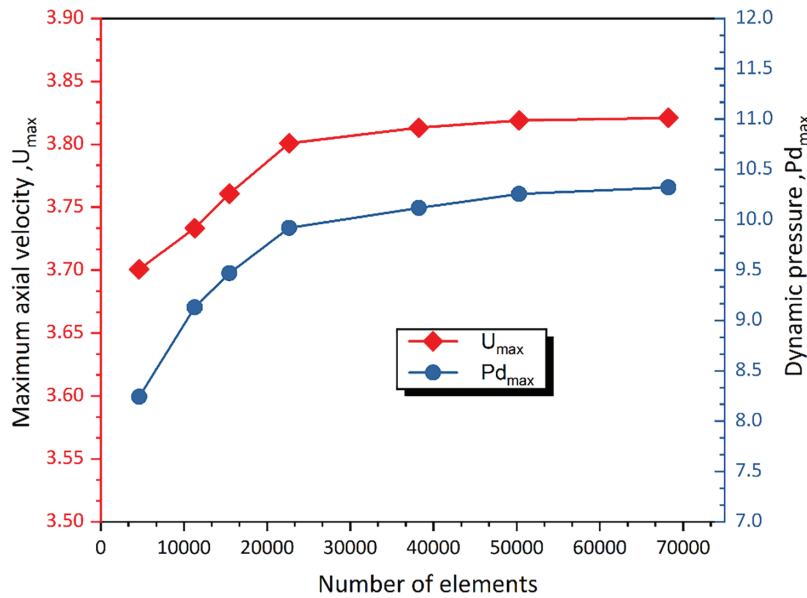


Figure 2: Mesh convergence analysis with $Re = 10,000$.

The effect of mesh refinement, indicating the variation of these key characteristics as a function of the number of nodes. It can be seen that for a node count of 50,298, the values of $U_{\max}$ and $Pd_{\max}$ are 3.8190 m/s and 10.2567 Pa, respectively. When moving to a finer mesh of 68,220 nodes, the relative variation for $U_{\max}$ is only 0.052%, and that of $Pd_{\max}$ is 0.62%. These small variations, less than 1% for both quantities, demonstrate the convergence of the results and the mesh independence of the solution for a node count of 50,298.

Consequently, considering the optimal balance between the accuracy of the results and the required computational resources, the mesh composed of 50,298 nodes was selected and adopted for all subsequent numerical simulations in this study.

The improved mesh used along all solid walls is seen in Fig. 3. Accurately capturing temperature and velocity variations in crucial flow regions requires this refinement.

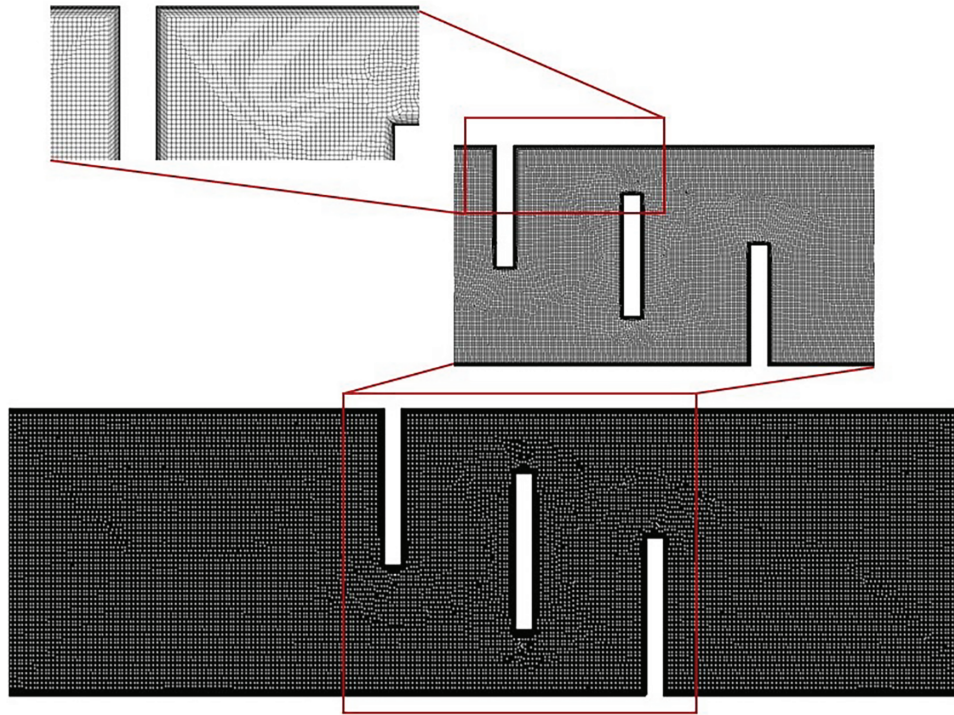


Figure 3: Simulation domain.

3 Numerical Validation

The robustness and reliability of the developed numerical model were rigorously validated by comparison with literature data. The system of differential equations governing flow and heat transfer was solved numerically using the finite volume method, relying on the SIMPLE algorithm for pressure-velocity coupling. To discretize the convective terms of the transport equations (momentum, energy, k , and ϵ), the QUICK (Quadratic Upstream Interpolation for Convective Kinematics) scheme was used, recognized for its accuracy in turbulent flows. The computational domain was discretized using an unstructured 2D quadrilateral mesh, an approach deemed particularly suited to the complexity of the geometry integrating the deflectors. To accurately capture the high velocity and temperature gradients that occur near the channel walls and baffle surfaces, the mesh was significantly refined in these critical regions. The turbulence model used for this validation is the standard k - ϵ model. To demonstrate the reliability of our approach, the obtained numerical results were compared with data published by Demartini et al. [46]. Figs. 4 and 5 present a comparison of the axial velocity profiles at a specific position ($x = 0.525$), located near the channel outlet, and the pressure coefficient at $x = 0.225$ across the height of a channel. This figure highlights an excellent agreement between the results of our numerical simulation and the reference numerical data [46]. Although a slight discrepancy can be observed compared to experimental data, the strong correlation with the numerical data reinforces the validity of our model for the studied configuration.

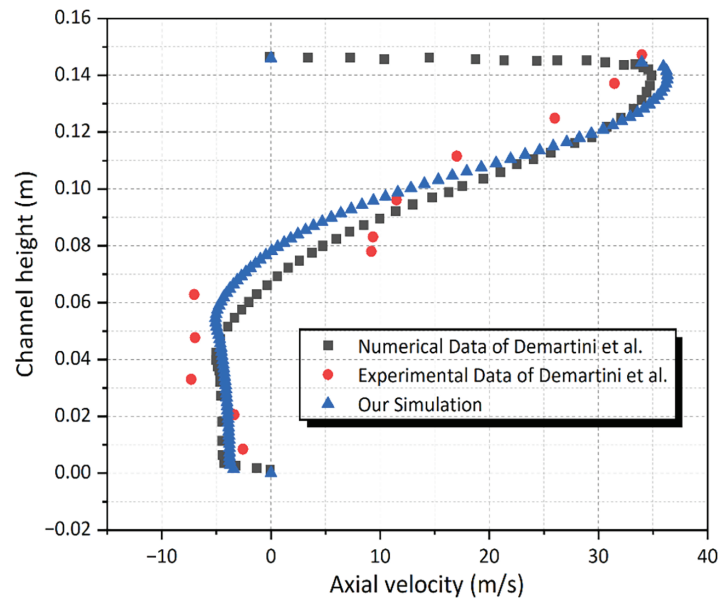


Figure 4: Comparison of the axial velocity: Our simulation and the results of Demartini et al. [46].

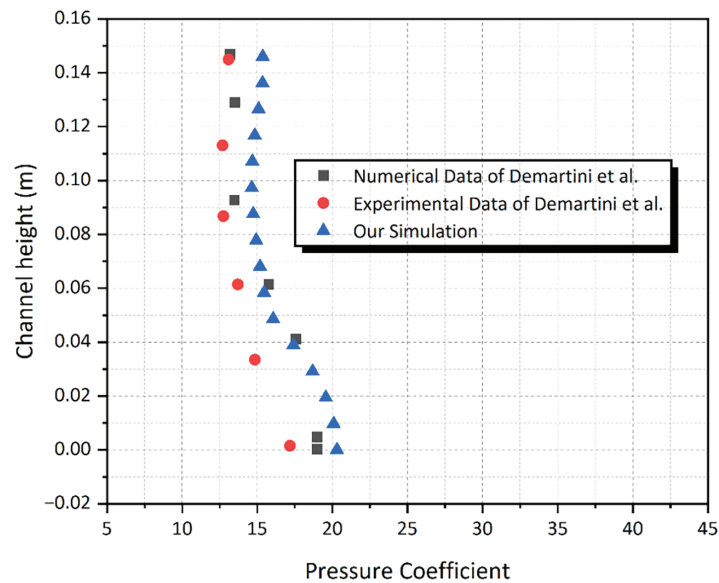


Figure 5: Comparison of the pressure coefficient variation at $x = 0.255$ m: Our simulation and the results of Demartini et al. [46].

4 Results and Discussion

Flow Characteristic

This section presents and discusses the results of the numerical simulations obtained for the turbulent airflow in the two geometric configurations of the heat exchanger channel, as illustrated in Fig. 1 (Cases A and B). A detailed analysis of the velocity and temperature fields was performed to understand the combined and distinct impact of these devices (baffles and central obstacles) on the hydrodynamic and thermal aspects of the flow. To visualize the flow topology and identify recirculation zones, the streamlines for the two cases

studied (Cases A and B) are presented in Fig. 6, for a Reynolds number of $Re = 10,000$. Examination of Fig. 6 reveals distinct flow dynamics induced by the presence of obstacles. For both configurations (Case A and Case B):

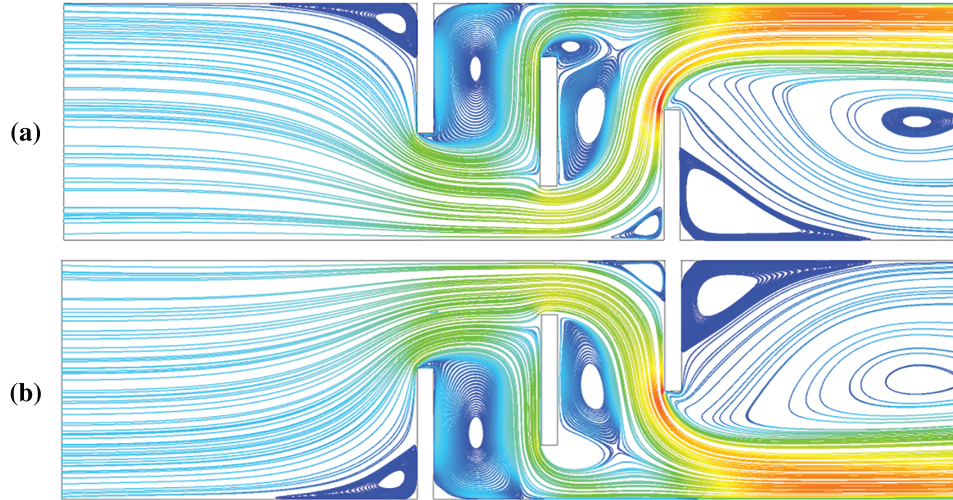


Figure 6: (a): Contour plots of streamlines for case A ($Re = 1000$), (b): Contour plots of streamlines for case B ($Re = 10,000$).

From the first deflector, flow separation phenomena are observed, leading to the formation of leading vortices upstream and wake vortices downstream of the upper surface of the deflector. The fluid is forced to bypass this obstacle, generating a pronounced recirculation zone behind it. The flow passing over and around the central obstacle and the second deflector creates additional vortex structures. A small vortex is generated specifically at the forward base of the second deflector.

Downstream of the second deflector, the separation phenomenon is particularly pronounced, leading to the formation of a large recirculation bubble behind this obstacle. In this wake zone, a large arch-shaped vortex dominates the flow structure, indicating strong fluid recirculation.

These streamline observations are crucial, as the formation and intensity of these vortices and recirculation zones are directly related to increased turbulence and mixing. This increased recirculation prolongs the residence time of the hot fluid near the exchange surfaces, thus promoting a significant improvement in heat transfer. The comparison between case A and case B (according to the figure, the central obstacle in case A is vertical, and inclined in case B, which affects the vortices in its immediate vicinity) reveals differences in the size and stability of the vortex structures, suggesting distinct hydrodynamic performances for each geometry.

These contours highlight the disturbances generated by the presence of the baffles and the central obstacle. In particular, the formation of intense recirculation zones downstream of each obstacle is observed, characterized by negative axial velocities (shown in blue in the legend). The extent and intensity of these zones are visibly influenced by the Reynolds number, indicating more complex flow dynamics as Re increases.

Examination of the velocity profiles reveals a significant deceleration of the flow downstream of the baffles, accompanied by the systematic formation of recirculation zones characterized by negative axial velocities immediately behind each baffle. Interestingly, an acceleration of the fluid in the lower part of the channel following the passage of the first baffle is noted, suggesting a channeling effect and a redistribution of the flow. The presence of the central rectangular obstacle further intensifies the flow velocity and induces a notable reorientation of the flow towards the upper part of the channel, a phenomenon that could be

attributed to the geometric disturbance and the creation of preferential flow paths. In contrast, a substantial increase in the axial velocity is observed downstream of the top of the second baffle. This acceleration is likely due to the contraction of the effective flow area and the formation of a vortex, which induces suction and acceleration of the fluid in this region.

Axial velocity fields were examined to characterize the hydrodynamic impact of the baffles and the central obstacle on the flow. Moreover, Fig. 7 presents the axial velocity contours for cases (A) and (B), at a Reynolds number of $Re = 10,000$. These visualizations are essential for understanding momentum redistribution and mixing intensity. In both configurations (Case A and Case B), the fluid's interaction with the baffles and the central obstacle induces significant changes in the velocity field. A marked fluid deceleration and the appearance of negative velocity zones (indicating recirculation) are systematically observed immediately downstream of each obstacle. These recirculation zones are key features because they extend the fluid's residence time, thus promoting heat transfer. More specifically for Case (A) (vertical central obstacle):

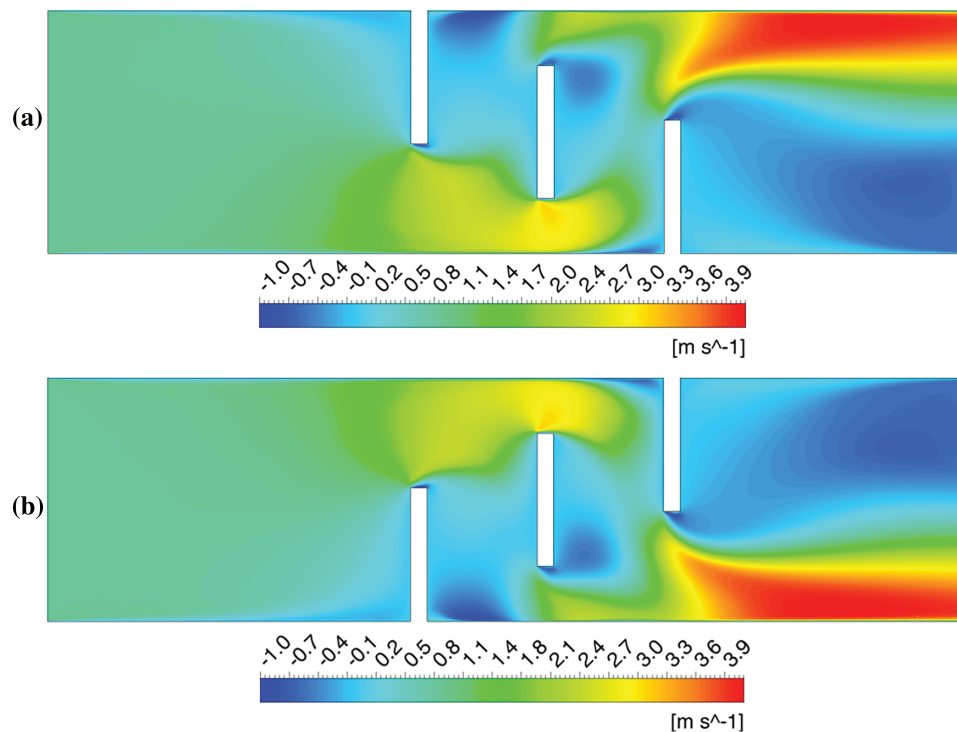


Figure 7: Velocity profiles with $Re = 10,000$: (a) Case A, (b) Case B.

After the first deflector, a clear acceleration of the air is observed along the lower wall of the channel, where the flow is constrained. The central obstacle acts as a second disruptor, intensifying this acceleration and directing the main flow towards the upper surface of the channel. Downstream of the top of the second deflector, a significant increase in axial velocity is visible, resulting from the constriction effect and the reformation of the shear layer after the intense wake zone, which can also generate a dynamic vortex.

As for case (B) (inclined central obstacle), although it shares general characteristics with case A (formation of recirculations, acceleration zones), the modification of the orientation of the central obstacle induces an asymmetry in the velocity distribution in its immediate vicinity, and potentially an alteration in the size and intensity of the downstream recirculation zones. For example, the obstacle inclination

may favor a more pronounced acceleration on one side of the channel or a different redistribution of the turbulent kinetic energy, compared to the vertical obstacle case. This difference in the topology of the velocity fields between case A and case B is crucial to evaluate the relative thermal-hydraulic performances of each configuration.

Dynamic pressure field analysis is equally fundamental for characterizing the hydrodynamic behavior of the flow and estimating pressure losses within the channel. Moreover, Fig. 8 illustrates the dynamic pressure contours for both configurations, Case (A) and Case (B), at a Reynolds number of $Re = 10,000$. It is observed that in both scenarios, the fluid exhibits significantly low dynamic pressure zones, particularly at the channel inlet and upstream of the deflectors. Dynamic pressure values are also notably low within the wake and recirculation zones formed behind each obstacle, where the local fluid velocity is low or even negative. Similarly, the frontal and rear surfaces of the obstacles are characterized by significant pressure gradients. More specifically, for Case (A) (vertical central obstacle): The dynamic pressure increases sharply in the upper part of the channel, precisely in the area where the main flow is accelerated and constrained by the lower deflector and the central obstacle (as previously observed in Fig. 7). At the same time, a notable decrease in dynamic pressure is observed in the lower part of the same exchange, corresponding to a zone of relative fluid expansion and deceleration. Case (B) (inclined central obstacle) exhibits a distinct dynamic pressure distribution compared to Case A. The inclination of the central obstacle modifies the distribution of high- and low-pressure zones. Thus, unlike Case A, the main flow is directed differently, resulting in an increase in dynamic pressure in the lower part of the channel and a decrease in the upper part, thus reversing the trend observed in Case A. This variation highlights the direct impact of the orientation of the central obstacle on the flow dynamics and, by extension, on the distribution of pressure drops.

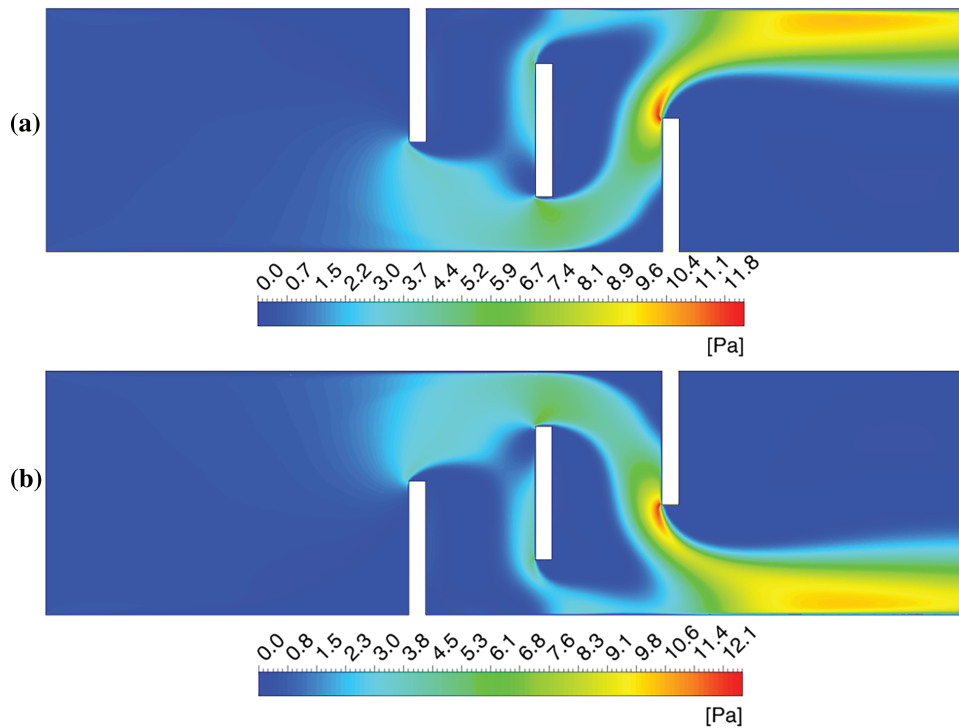


Figure 8: Dynamic pressure profiles with $Re = 10,000$: (a) Case A, (b) Case B.

In general, dynamic pressure values increase considerably in sections where the flow is highly accelerated (necking zones) and decrease within recirculation cells and expansion zones, which follows the

principles of energy conservation and the transformation of kinetic energy into pressure energy and *vice versa*. These variations are crucial for the evaluation of the overall system performance via the thermal performance factor (TPF), where pressure losses are a key parameter.

Turbulent kinetic energy (k) provides crucial information on the intensity of velocity fluctuations and, consequently, on mixing efficiency and the potential for heat transfer enhancement within the flow. Fig. 9 presents the turbulent kinetic energy fields for both configurations, Case (A) and Case (B), at a Reynolds number of $Re = 10,000$. In both cases, we observe significant generation and release of turbulent kinetic energy downstream of each baffle and the central obstacle. These areas of high turbulence are generally associated with flow separation and recirculation zones, where strong velocity and shear gradients are present. The increase in turbulent kinetic energy in these areas is consistent with the general understanding of flow disturbances caused by obstacles. Specifically, high k values are observed immediately downstream of the baffle edges and the central obstacle, as well as within the large recirculation cells. These maxima indicate that the turbulators are effective in breaking up laminar flow structures and inducing increased transverse mixing. A comparison between Case (A) and Case (B) in Fig. 9 reveals notable differences in the spatial distribution of turbulent kinetic energy, directly attributable to the orientation of the central obstacle. In Case (A), with the vertical central obstacle, areas of high turbulent kinetic energy are concentrated primarily in the immediate wake of each baffle and the central obstacle, with relatively symmetrical turbulence propagation within the channel. The region of maximum k after the second baffle is well defined and centered.

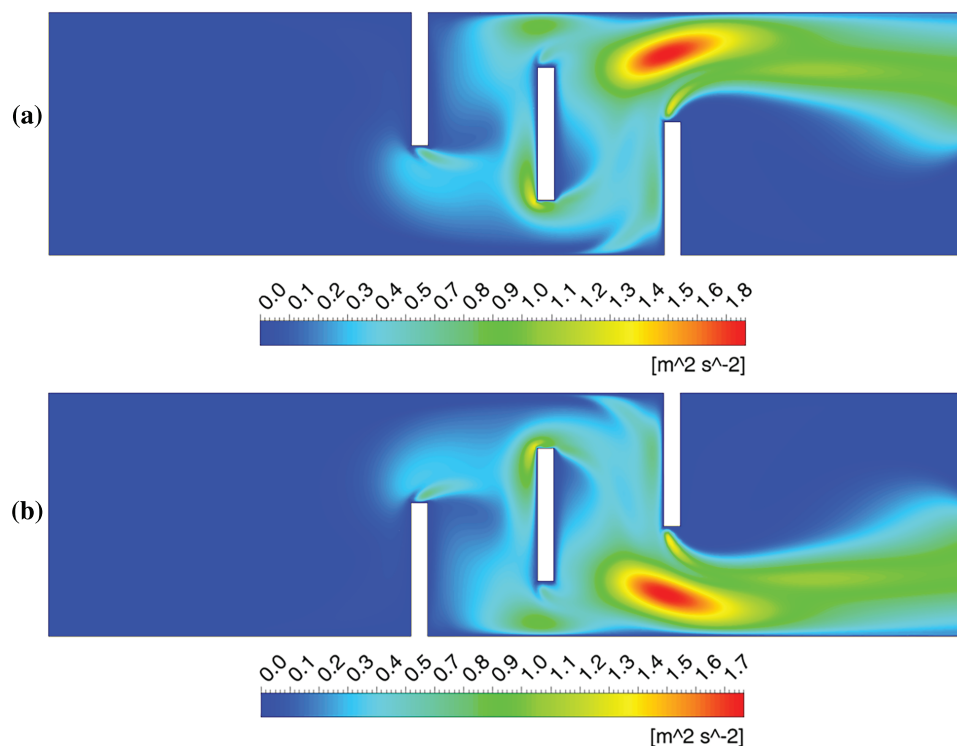


Figure 9: Field of turb-kinetic-energy with $Re = 10,000$: (a) Case A, (b) Case B.

In contrast, in Case (B), where the central obstacle is inclined, this inclination modifies the separation dynamics, leading to an asymmetric redistribution of turbulent kinetic energy. It is observed that the turbulence generated by the inclined central obstacle is preferentially directed toward one of the walls (the lower wall in Fig. 9), creating a more intense or more extensive shear zone along this wall. The high- k zone

after the second deflector in Case B also appears more laterally extensive and potentially less concentrated, suggesting a different diffusion of turbulence.

These variations in the intensity and spatial distribution of turbulent kinetic energy between Cases A and B are fundamental, as they directly influence the efficiency of fluid mixing and the disruption of thermal boundary layers. More intense or better distributed turbulence can significantly improve heat transfer but can also lead to different pressure drops between the two configurations. To provide a more detailed analysis of the flow dynamics along the channel, axial velocity profiles as a function of height are presented in Fig. 10 for various axial positions (x): (a) $x = 0.189$ m, (b) $x = 0.255$ m, (c) $x = 0.345$ m, (d) $x = 0.375$ m, and (e) $x = 0.525$ m. These profiles are compared across the three Reynolds numbers investigated ($Re = 10,000$, $Re = 20,000$, and $Re = 30,000$). In fact, the profiles of axial velocity at different positions are examined, firstly, and before the first deflector (Fig. 10a, $x = 0.189$ m), the axial velocity profiles already exhibit a slight perturbation due to the impending obstacle. A shift in the flow of the velocity direction is observed near the upper surface of the deflector. As anticipated, increasing the Reynolds number leads to higher overall flow velocities throughout the profile.

Downstream of the first deflector (Fig. 10b, $x = 0.255$ m): At this position, the flow has interacted with the first deflector. A significant acceleration of the fluid is observed in its primary left-to-right direction, particularly in the lower portion of the channel where the flow becomes channelized. Simultaneously, extensive recirculation zones, characterized by negative velocities, form downstream of the deflector. The extent and intensity of these recirculation zones visibly increase with the Reynolds number, indicating an amplification of mixing phenomena. Moreover, approaching the central obstacle (Fig. 10c, $x = 0.345$ m): A notable increase in flow velocity is observed along the lower wall just before the obstacle, attributable to a convergence effect. Conversely, a low-velocity zone, or even a recirculation region, appears immediately downstream of this central obstacle. At the top of the second deflector (Fig. 10d, $x = 0.375$ m): Further downstream, the flow velocity becomes higher than that observed at $x = 0.345$ m. The flow acceleration is strongly influenced by the presence of this second deflector, which induces a new disturbance and a re-acceleration of the dominant flow.

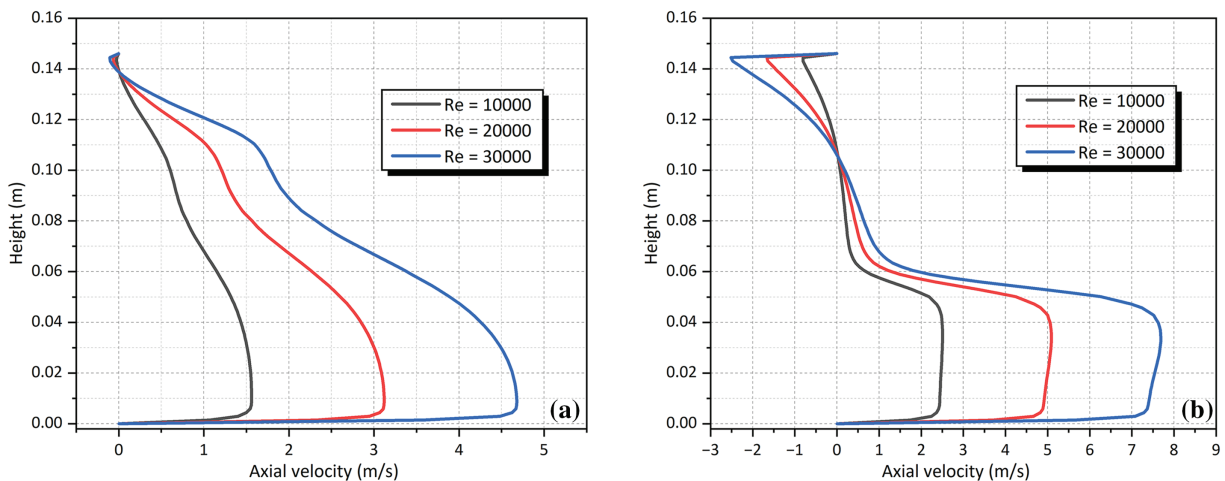


Figure 10: (Continued)

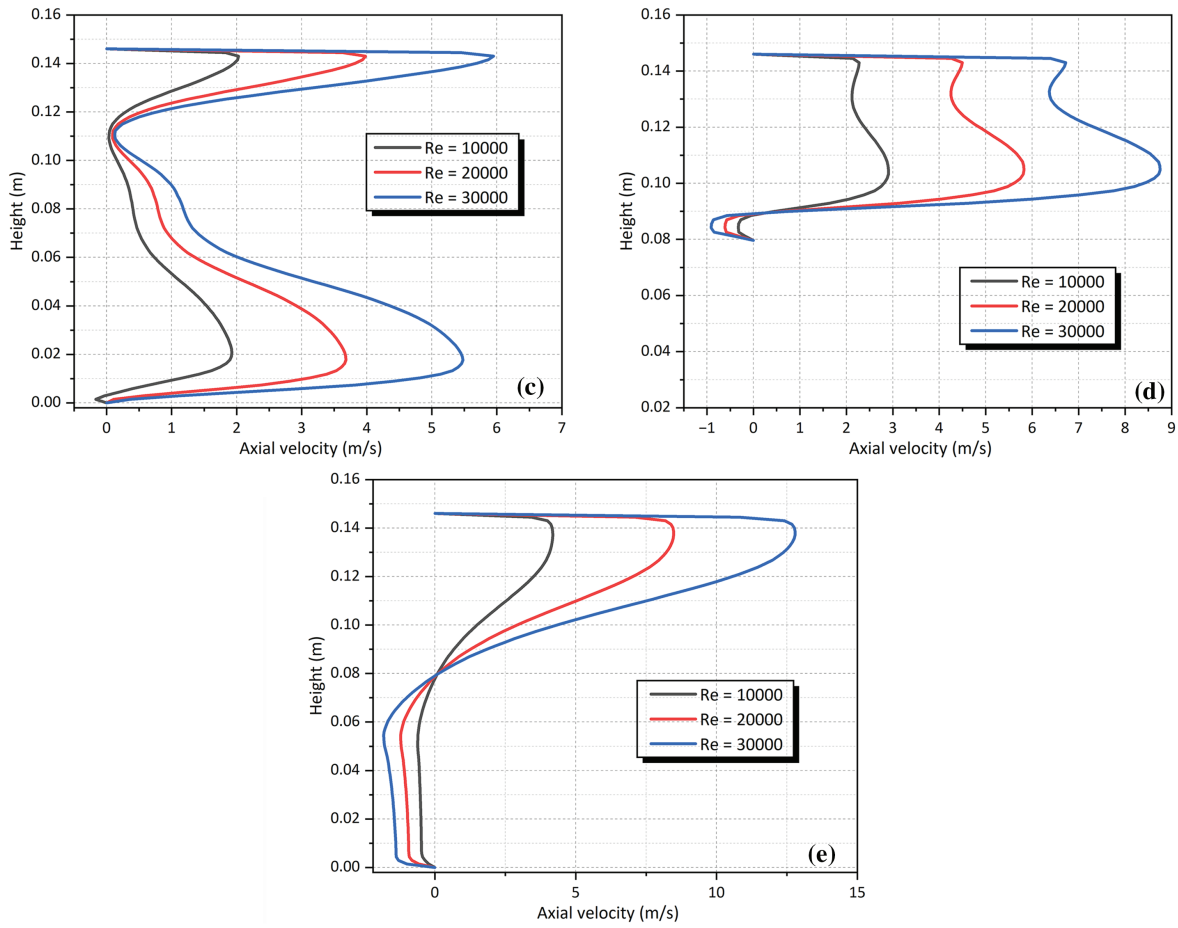


Figure 10: Axial velocity profiles for different Reynolds number values at: (a) $x = 0.189$ m, (b) $x = 0.255$ m, (c) $x = 0.345$ m, (d) $x = 0.375$ m and (e) $x = 0.525$ m.

Near the channel outlet (Fig. 10e, $x = 0.525$ m): At this final position, the flow velocity reaches its maximum value for all Reynolds numbers, exceeding five times the inlet velocity. This acceleration is accompanied by the formation of a large vortex near the second deflector, leading to significant variations in velocity values across the cross-section.

Overall, Fig. 10 confirms that the combination of baffles and the central obstacle induce an acceleration of the flow in its primary left-to-right direction through the channel. The changes in flow direction instigated by these obstacles are responsible for the highest velocity values, which consistently tend to occur near the upper channel wall in the accelerated regions.

In addition to the velocity fields, the impact of obstacles on the temperature distribution within the channel is also significant. The temperature profiles, presented in Fig. 11 for the same axial positions as previously (a–e), reveal notable variations. As anticipated, the largest temperature gradients are located near the obstacles and heated walls, reflecting the intense heat exchange in these regions. A key phenomenon observed is the significantly higher fluid temperature within the recirculation zones compared to the undisturbed flow regions. This is attributable to the increased fluid residence time in these low-velocity zones, allowing for greater heat accumulation. This observation is fully consistent with the results previously reported by Nasiruddin and Kamran Siddiqui [50], confirming the influence of recirculation on heat transfer efficiency.

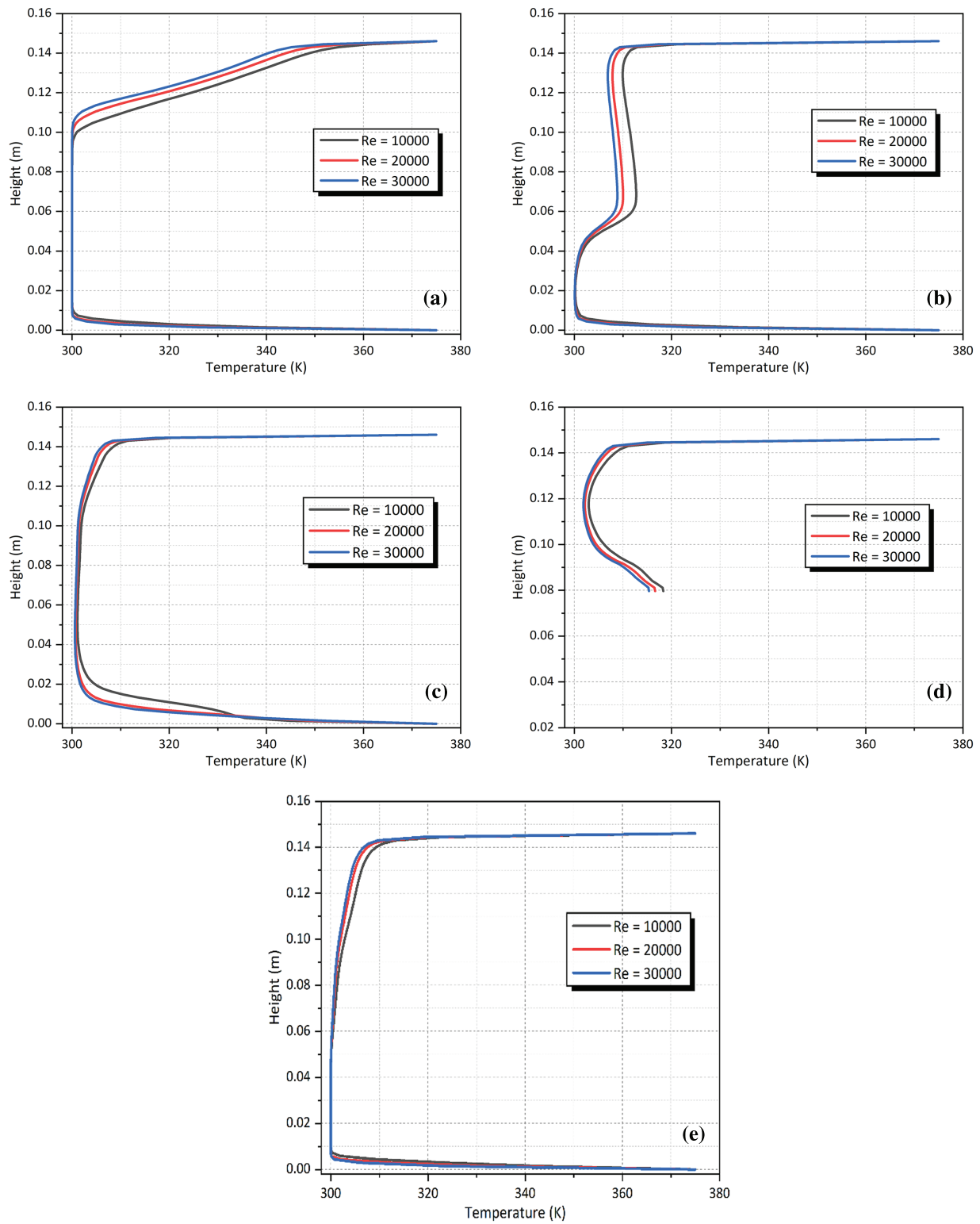


Figure 11: Temperature profiles for different Reynolds number values at: (a) $x = 0.189$ m, (b) $x = 0.255$ m, (c) $x = 0.345$ m, (d) $x = 0.375$ m and (e) $x = 0.525$ m.

Furthermore, a general trend of decreasing mean fluid temperature is observed with increasing Reynolds number, for each axial position considered. This behavior is explained by the intensification of turbulent mixing at higher Reynolds numbers, resulting in a better distribution of thermal energy across the channel section and a reduced contact time of the fluid with the hot walls, thus facilitating its cooling or more homogeneous heat distribution. To evaluate the thermal efficiency of the studied configurations, we examined the evolution of the average Nusselt number, Nu , as a function of Reynolds number. Fig. 12 illustrates this evolution comparatively for Case (A) and Case (B), over a Reynolds number range from $Re = 10,000$ to $Re = 30,000$.

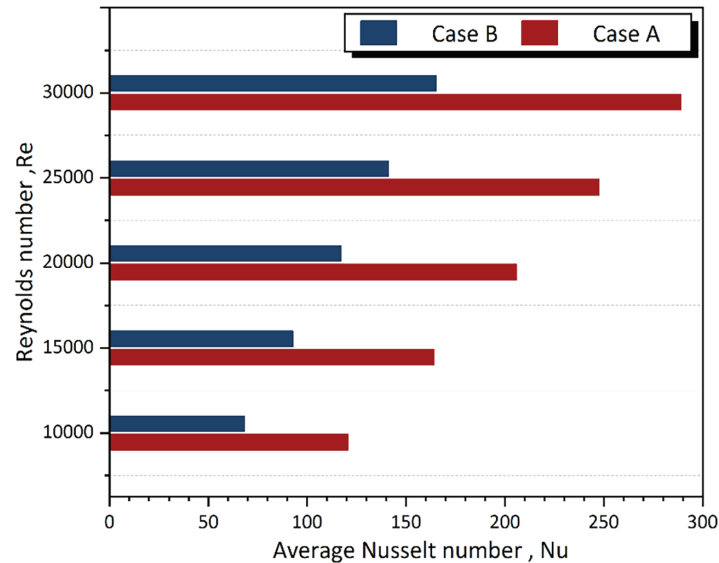


Figure 12: Variations of average Nu number with Re number for cases A and B.

Analysis of Fig. 12 significantly reveals that the installation of baffles in the Case (A) configuration leads to a systematically greater increase in the average Nusselt number compared to the Case (B) configuration. As clearly shown in the figure, and consistent with expectations based on the turbulator arrangement, the Nu of Case (A) is significantly higher than that of Case (B) for all Reynolds numbers examined.

This superior thermal performance of Case (A) can be attributed to the flow characteristics previously discussed. The vertical central obstacle in Case (A) appears to generate a more effective turbulence and recirculation topology to disrupt the thermal boundary layer along the exchanger walls, compared to the inclined obstacle in Case (B). The distribution of the velocity fields (Fig. 7) and turbulent kinetic energy (Fig. 9) suggests that Case (A) creates more intense mixing or recirculation zones that are more conducive to enhancing heat transfer through the channel. Furthermore, Fig. 12 indicates that for both cases, Nu tends to increase with Reynolds number, which is consistent with the intensification of turbulence and convective heat transfer at higher flow velocities. The performance gap between Case (A) and Case (B) also appears to be maintained throughout the Reynolds number range studied. This Nu analysis constitutes an essential component of the overall evaluation of the system performance, which will be completed by the examination of the friction factor to determine the Thermal Performance Factor (TPF).

A comprehensive performance assessment of a heat exchanger is not limited to improving heat transfer; it must also consider the pressure drop penalty. Fig. 13 illustrates the variation of the average friction coefficient (f) as a function of Reynolds number for both configurations (Case A and Case B), over the same Re range of 10,000 to 30,000. As expected for turbulent flows in ducts equipped with turbulators, the figure

shows that the average friction coefficient f , increases with increasing Reynolds number for both models. This increase is due to the intensification of the interaction between the fluid and obstacles at higher velocities, which generates increased shear forces and friction losses.

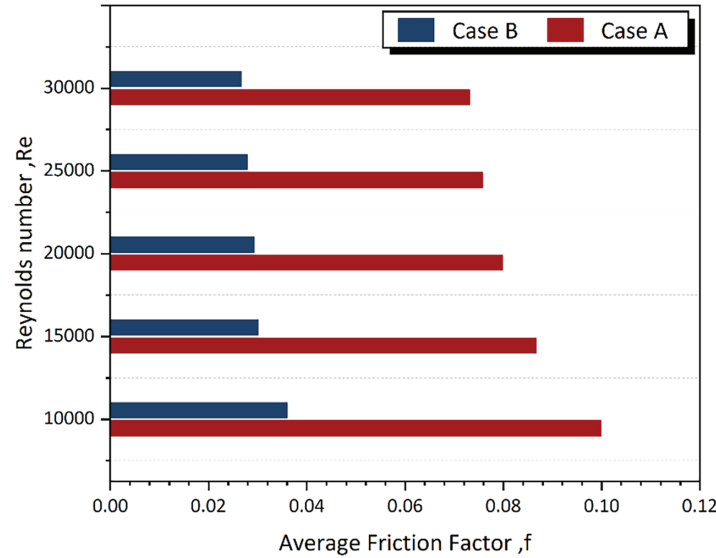


Figure 13: Variations of average friction coefficient with Re number for cases A and B.

It is particularly important to note that the average increase in the average friction coefficient in Case (A) is significantly more pronounced than in Case (B). This suggests that the configuration in Case (A), while more efficient in terms of heat transfer (as discussed for Fig. 12), also results in a substantial increase in pressure drops across the entire study domain. This difference may be correlated with the geometry of the central obstacle: the vertical obstacle in Case (A) likely creates more direct flow resistance and more intense or resistive recirculation zones than the inclined obstacle in Case (B), which could allow for a smoother flow around it, thus reducing frictional forces.

Understanding the evolution of f is essential for calculating the Thermal Performance Factor (TPF), which will determine whether the improvement in heat transfer sufficiently offsets the increase in pressure drops to justify adopting one configuration over the other.

The final evaluation of the efficiency of a heat exchanger configuration integrated with turbulators is summarized by the Thermal Performance Factor (TPF), as defined by Eq. (19). The TPF combines the heat transfer improvement (via the normalized Nusselt number) and the pressure drop penalty (via the normalized friction coefficient). Fig. 14 illustrates the variations of the TPF as a function of Reynolds number for the two configurations studied, Case (A) and Case (B). The analysis of Fig. 14 reveals several crucial points: Generally, the TPF increases with increasing number of Reynolds for both cases. This indicates that the overall efficiency of the turbulators improves at higher flow velocities, where increased turbulence and mixing more significantly compensate for increased pressure drops. Consistent with the observations on the average Nusselt number (Fig. 12) and the average friction coefficient (Fig. 13), the TPF value for the Case (A) geometry is consistently higher than that of Case (B) across the entire Reynolds number range examined. For Case A, the TPF ranges from approximately 2.51 to 2.59, while for Case B, it is between 2.02 to 2.07. This means that despite higher pressure drops (Fig. 13), the heat transfer improvement in Case A is significant enough to provide superior overall performance. The TPF values greater than unity ($TPF > 1$) for both configurations

and all Reynolds numbers confirm that the integration of turbulators is beneficial, as it provides a thermal gain that exceeds the energy penalty compared to a smooth channel.

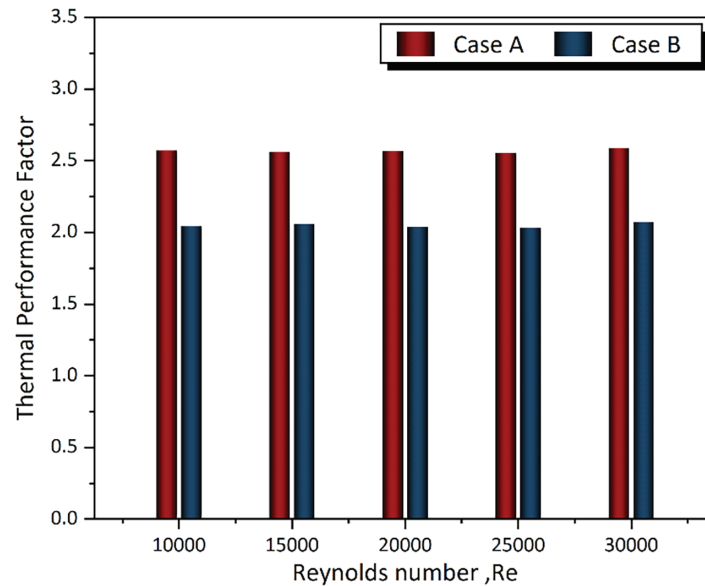


Figure 14: Variations of TPF with Re number for cases A and B.

This superiority of Case (A) in terms of TPF is explained by the complex interaction between flow and obstacle geometry. Although Case (A) generates higher pressure drops, the effectiveness of its vertical central obstacle in creating more intense turbulence and recirculations more favorable to heat exchanges (as seen in Figs. 7 and 9) prevails over the associated penalty. Case (B), with its inclined obstacle, offers lower pressure drops but a less pronounced heat transfer improvement, resulting in an overall lower TPF. These results are essential for the design and optimization of heat exchangers, highlighting the importance of an optimal trade-off between heat transfer and pressure drop.

For $Re = 10,000$, Fig. 15 presents the velocity field distribution for four different configurations characterized by varying baffle spacings. This visualization is essential for understanding the geometric impact on flow dynamics.

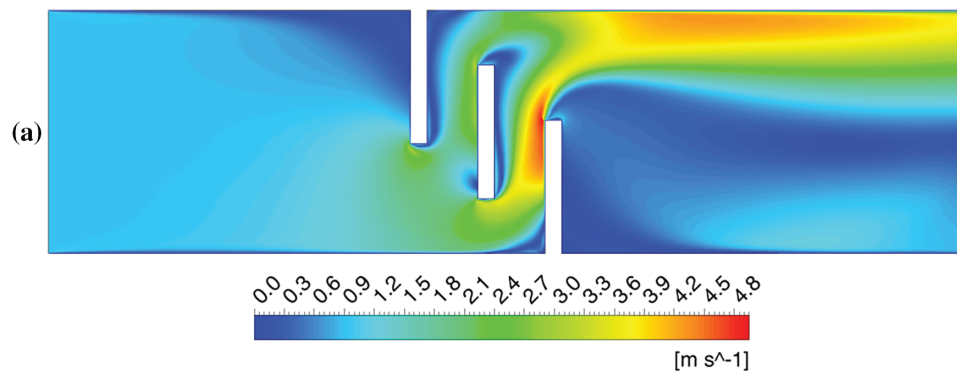


Figure 15: (Continued)

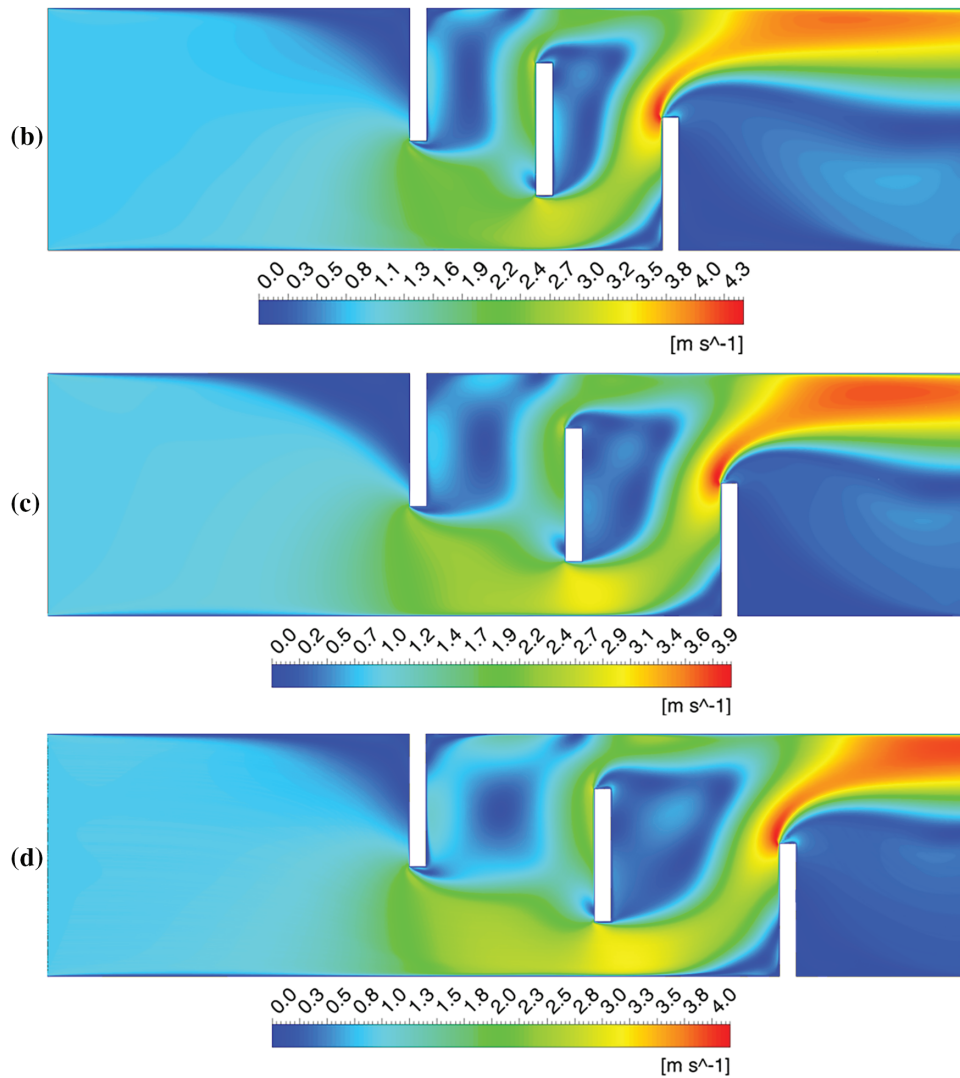


Figure 15: Distribution of velocity fields with $Re = 10,000$ as a function of baffle spacing: (a) Spacing d_1 , (b) Spacing d_2 , (c) Spacing d_3 , (d) Spacing d_4 .

It is observed that the presence of baffles systematically induces the formation of vortices and significantly alters the channel flow patterns. The median obstacle, in particular, impedes the main fluid trajectory, leading to a significant modification of the recirculation cells formed upstream and downstream of the baffles. Analysis of the different configurations reveals that the size and intensity of these vortices are directly influenced by the spacing between the baffles.

More specifically, it appears that the shorter the distance between the baffles, the more pronounced the flow disturbances and the greater the changes in the vortex structures. This means that the intensity of mixing and shear increases inversely with the inter-baffle spacing. As spacing decreases and the baffles move closer together, the fluid flow area between them is reduced, resulting in significant local acceleration of the flow. Consequently, maximum velocity values are generally observed in areas of closer spacing, particularly near the upper surface of the channel, close to its outlet, where flow contraction is greatest.

These changes in velocity fields and the generation of turbulence are directly related to the potential improvement in heat transfer, but also to the associated pressure drops, as high velocities and intense disturbances increase friction and energy dissipation losses.

To assess the impact of different geometric configurations on heat transfer efficiency, Fig. 16 illustrates the evolution of the average Nusselt number (Nu) as a function of Reynolds number, over a range from $Re = 10,000$ to $Re = 30,000$. Four baffle spacing configurations are examined: $S = D/2$, $S = D$, $S = 5D/4$, and $S = 3D/2$. The graph reveals several important trends. Firstly, for all spacing configurations, the average Nusselt number increases in direct proportion to the Reynolds number. This observation is consistent with the intensification of convective phenomena and turbulence at higher flow velocities. Secondly, the comparative analysis of spacings clearly shows that heat transfer efficiency is inversely proportional to the distance between obstacles: the heat transfer rate improves significantly as the spacing decreases.

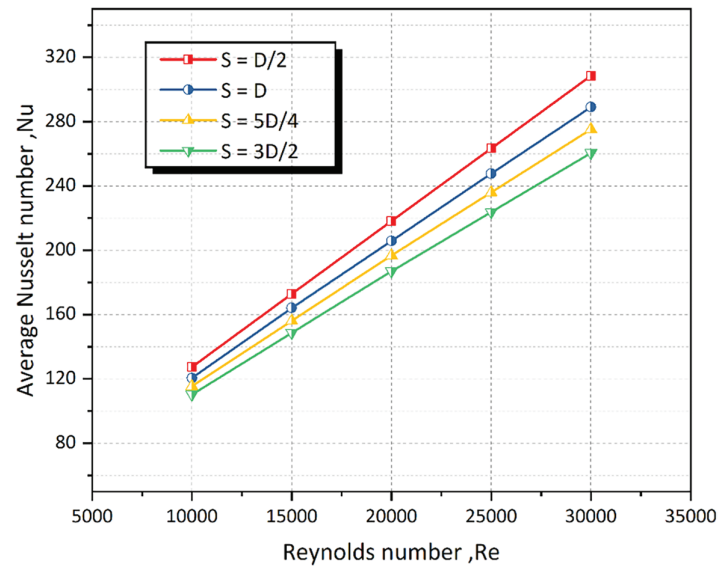


Figure 16: Variation of the average Nusselt number (Nu) as a function of Re number for different baffle spacing configurations ($S/D = 0.5; 1.0; 1.25; 1.5$).

Specifically, the highest average Nusselt number is consistently observed for the $S = D/2$ configuration, while the lowest is recorded for $S = 3D/2$ across the entire Reynolds number range. This variable performance is directly related to the hydrodynamic characteristics of the flow. A narrow spacing ($S = D/2$) between obstacles further constrains the flow, resulting in more intense local acceleration and increased turbulence and shear generation (as visualized in Fig. 15). This phenomenon promotes more efficient mixing between the core fluid and the thermal boundary layer adjacent to the hot channel surface, thus improving heat transfer.

Conversely, for wider spacings, such as $S = 3D/2$, the recirculation flows generated by the obstacles are less intense and less conducive to significant mixing or sufficient disruption of the thermal boundary layer. This configuration, therefore, proves less effective in improving heat transfer, as it fails to promote as vigorous a heat exchange between the channel surface and the fluid core. In conclusion, within the Reynolds number range examined, the use of obstacles with a spacing of $S = D/2$ produces the highest heat transfer rate (Nu) compared to other configurations.

Fig. 17 shows the evolution of the average friction coefficient (f) as a function of Reynolds number (Re), for the same four longitudinal spacing (S) configurations between baffles as previously. This analysis

is crucial for quantifying the pressure drop penalty associated with the improved heat transfer. It is clearly demonstrated that, for all the spacing configurations studied, the average friction coefficient decreases as the Reynolds number increases, which is characteristic of turbulent flows in rough or obstructed channels. More importantly, the comparative analysis of the spacings reveals a trend opposite to that of heat transfer: the friction coefficient decreases as the longitudinal spacing between the baffles increases, and this determines the Reynolds numbers examined.

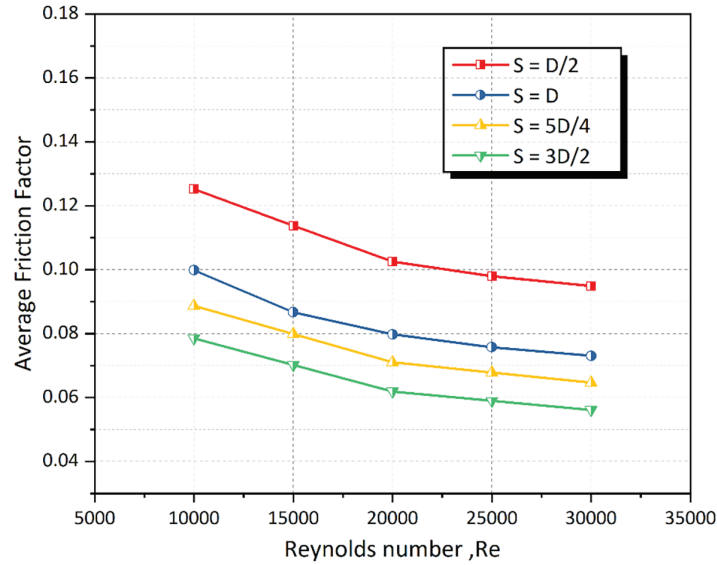


Figure 17: Variation of the average friction coefficient (f) as a function of Reynolds number for different baffle spacing configurations ($S/D = 0.5; 1.0; 1.25; 1.5$).

More specifically, the highest friction factor is observed when the longitudinal spacing is the smallest ($S = D/2$), reaching maximum values particularly at low Reynolds numbers. Conversely, the friction coefficient is minimal for the largest spacing ($S = 3D/2$). This relationship is explained by the nature of the fluid-obstacle interaction. A shorter separation distance between obstacles ($S = D/2$) results in a more severe constriction of the flow area, a more pronounced acceleration of the fluid, and a more frequent and intense generation of separations and recirculation. These phenomena increase not only the shear forces at the walls, but also, more importantly, the form drag due to the obstacles, which results in greater pressure losses. Conversely, a larger spacing allows the flow to develop more freely, notably the intensity of the disturbances and, consequently, friction losses.

These observations on the friction coefficient are essential and must be compared with the heat transfer gains for an overall assessment of the performance of each configuration via the Thermal Performance Factor (TPF).

The overall performance assessment of heat exchangers equipped with baffles, taking into account both the heat transfer improvement and the pressure drop penalty, is summarized by the Thermal Performance Factor (TPF). Fig. 18 illustrates the evolution of the TPF as a function of Reynolds number for the four baffle spacing configurations studied.

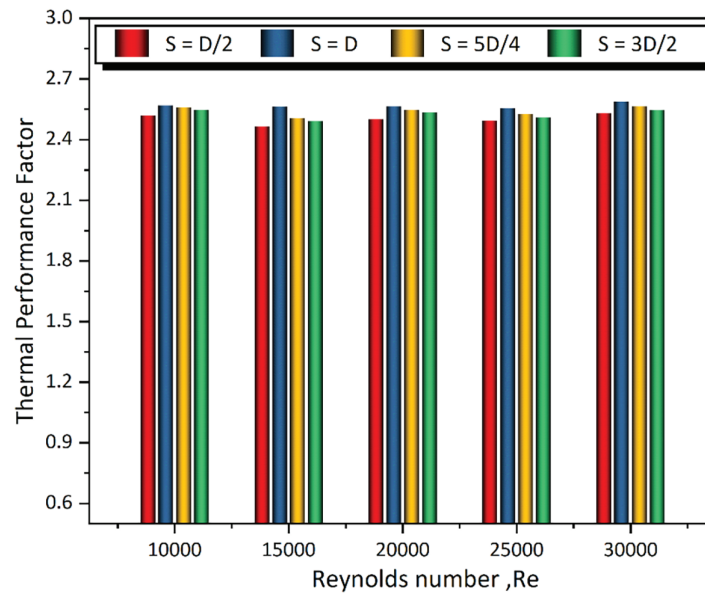


Figure 18: Variation of Thermal Performance Factor (TPF) as a function of Reynolds number for different baffle spacing configurations ($S/D = 0.5; 1.0; 1.25; 1.5$).

This figure is crucial because it allows for a direct comparison of the net efficiency of each design. It can be observed that, for all configurations and Reynolds number ranges, the TPF remains above unity ($TPF > 1$). This confirms that the integration of baffles is overall beneficial, as the heat transfer gain outweighs the increase in pressure drops. However, the comparative analysis of the spacings reveals a particularly interesting trend:

The highest Thermal Performance Factor is obtained with the configuration where the spacing is $S = D$. This result, although the highest average Nusselt number was observed for $S = D/2$, indicates that the trade-off between improved heat transfer and pressure penalty is optimal for this spacing. The $S = D$ configuration manages to generate sufficient turbulence to significantly improve heat transfer, without inducing excessively high-pressure drops. Conversely, the lowest TPF is observed for the $S = D/2$ spacing. Despite its excellent heat transfer performance, the penalty associated with its friction coefficient is disproportionate and degrades its overall performance.

These results highlight the critical importance of optimizing baffle spacing. Too close a spacing, although very efficient for heat transfer, results in prohibitive pressure losses that reduce the overall energy performance of the system. Choosing optimal spacing (here, $S = D$) is therefore essential for designing baffle heat exchangers that are both thermally efficient and economically viable.

Fig. 19 presents the velocity fields (including streamlines and velocity contours) for five distinct values of the baffle inclination angle (θ), all at a constant $Re = 10,000$. This visualization allows for analysis of the geometric influence of obstacle orientation on the flow structure. For all angular configurations, the persistence of recirculation zones (vortices) upstream and downstream of the obstacles is systematically observed. These zones, essential for improving heat transfer, are directly modulated by the baffle inclination angle. The figure highlights that the obstacle angle significantly influences the size, shape, intensity, and position of the vortices.

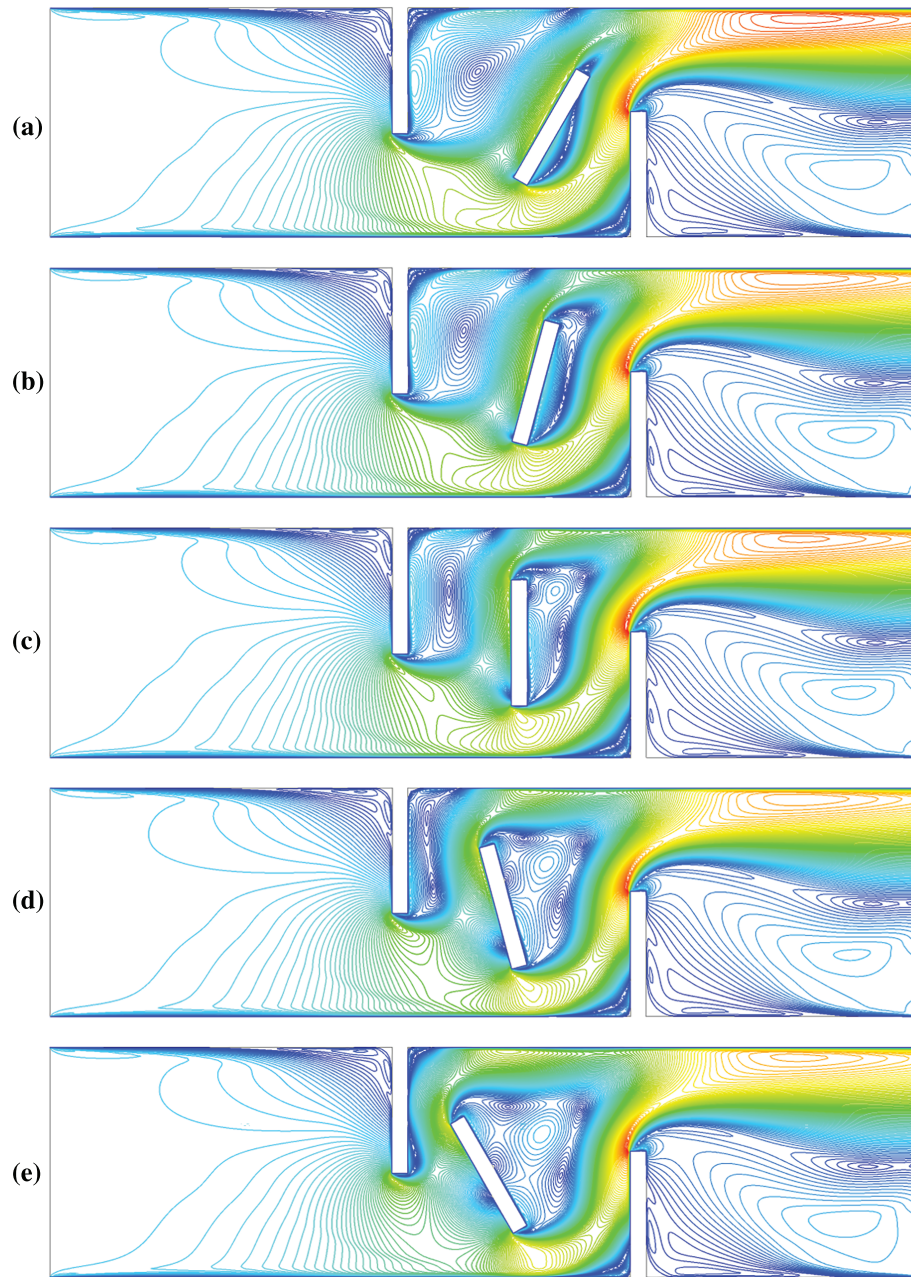


Figure 19: Velocity fields (contours and streamlines) along the channel for $Re = 10,000$ at different baffle inclination angles: (a) $\theta = 60^\circ$, (b) $\theta = 75^\circ$, (c) $\theta = 90^\circ$, (d) $\theta = 105^\circ$, and (e) $\theta = 120^\circ$.

For angles where the baffle directly opposes the flow (e.g., near $\theta = 90^\circ$), the recirculation zones are generally larger and more intense upstream, with a more pronounced deviation from the main flow. As the inclination angle varies from the perpendicular position (moving away from 90° to 60° or 120°), the flow can become more complex, potentially resulting in the formation of secondary vortices or changes in the stability of the primary recirculation zones.

These variations in vortex structure alter not only fluid mixing but also the extent of regions where the fluid is significantly slowed or accelerated, thus affecting velocity gradients and wall shear forces. These

detailed observations of velocity fields are fundamental to understanding the subsequent variations of heat transfer and pressure losses as a function of the baffle inclination angle, providing a physical basis for thermal-hydraulic performance.

Fig. 20 illustrates the effect of the baffle inclination angle (θ) on the average Nusselt number (Nu) at the upper channel wall, for a Reynolds number range from 10,000 to 30,000. The results are presented for five distinct angles: $\theta = 60^\circ$, 75° , 90° , 105° , and 120° . It is observed that, for each angular configuration, the average Nusselt number increases significantly with increasing number of Reynolds. This trend, already noted in previous analyses, confirms that increasing flow velocity intensifies forced convection and turbulence, thus promoting heat transfer. Regarding the influence of the inclination angle, the figure reveals a notable dependence. The average Nusselt number tends to increase with the inclination angle starting from $\theta = 60^\circ$. The highest Nusselt values are achieved for angles $\theta = 90^\circ$, 105° , and 120° , which exhibit very similar and superior performance at shallower angles. This improvement is attributable to more efficient interaction between the obstacle and the flow, leading to optimal generation and intensification of vortices and recirculation zones (as detailed in Fig. 19). These vortex structures increase fluid mixing and effectively disrupt the thermal boundary layer at the hot wall surface, optimizing heat transfer. Conversely, shallower angles (e.g., $\theta = 60^\circ$ and 75°) offer less obstruction to the main flow, resulting in less intense flow disturbance and, consequently, less efficient heat transfer.

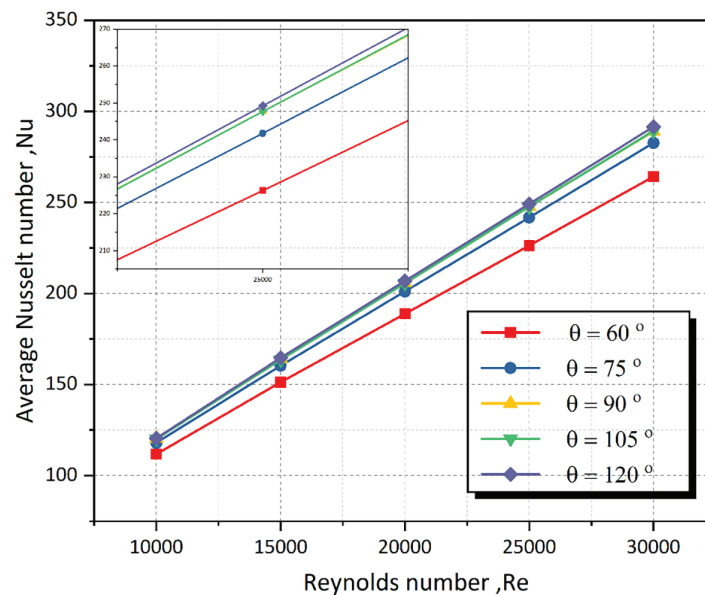


Figure 20: Effect of baffle inclination angle on the average Nusselt number (Nu) at the upper channel wall for different Reynolds numbers.

To assess the pressure drop penalty associated with different baffle orientations, Fig. 21 shows the evolution of the average friction coefficient (f) as a function of Reynolds number, for the five baffle inclination angles ($\theta = 60^\circ$, 75° , 90° , 105° , 120°). The graph shows that, for all the angular configurations studied, the average friction coefficient progressively decreases as the Reynolds number increases. This trend is typical of turbulent flows in channels with disturbing elements, where increasing inertia at higher Reynolds numbers tends to reduce the relative impact of viscous forces. Analyzing the influence of the inclination angle, it is clear that the 60° angle leads to the lowest friction coefficient over the entire Reynolds number range. Conversely,

the highest friction coefficient is observed for the 120° angle, followed closely by 105° . Generally, the friction coefficient tends to increase as the inclination angle increases from 60° to 120° .

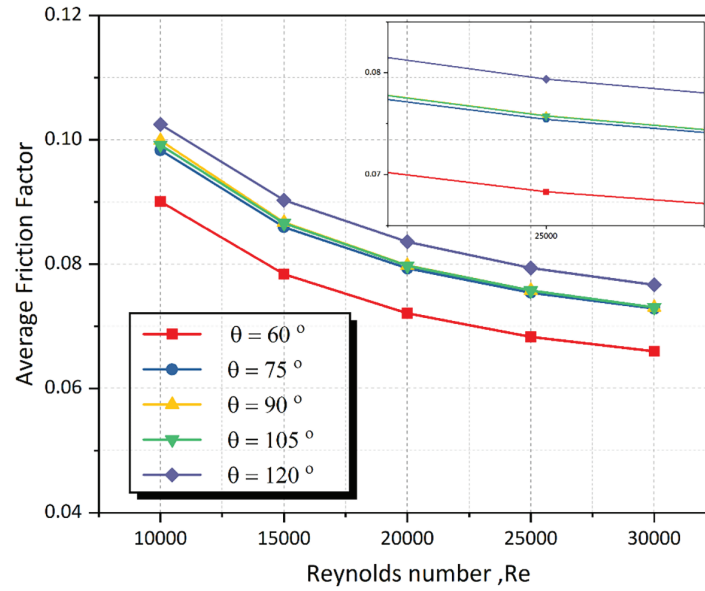


Figure 21: Effect of baffle inclination angle on the average friction coefficient (f) at the upper channel wall for different Reynolds numbers.

This observation is explained by changes in the flow structure. An angle of 60° provides less direct obstruction to the flow, reducing form drag and energy dissipation losses. In contrast, larger angles, particularly those approaching perpendicularity (such as 90°) or opposing it more (105° , 120°), generate larger recirculation zones and higher shear intensities (visible in Fig. 19), which translates into increased pressure drops. These friction results are fundamental for the calculation of the Thermal Performance Factor (TPF), because they represent the energy “penalty” associated with the heat transfer gains observed in Fig. 20.

Fig. 22 illustrates the influence of the baffle inclination angle (θ) on the Thermal Performance Factor (TPF), a key metric for assessing the trade-off between improved heat transfer and pressure penalty. The results are presented as a function of Reynolds number for the five angles studied: 60° , 75° , 90° , 105° , and 120° . Analysis of this figure reveals several crucial points. First, for all angular configurations, the TPF is consistently greater than unity ($TPF > 1$), indicating that the use of baffles is beneficial in terms of overall thermal-hydraulic performance across the entire Reynolds number range examined. This means that the increased heat transfer always offsets the increased pressure drops. Second, a significant dependence of the TPF on the baffle inclination angle is observed. The TPF generally tends to increase with the inclination angle up to a certain point. The angles of $\theta = 90^\circ$, 105° , and 120° display very close and highest TPF values, suggesting a range of optimal angles. Specifically, it appears that the highest performance is achieved for the angle of $\theta = 120^\circ$ or $\theta = 105^\circ$, which maintain a TPF slightly higher than or equivalent to the other angles across most Reynolds numbers. This trend is explained by the synergy between heat transfer gains and pressure drops. Although angles close to 90° generate excellent heat transfer, slightly obtuse angles (105° , 120°) can maintain a substantial improvement in heat transfer while exhibiting a proportionally less penalizing increase in pressure drops. This leads to a more favorable benefit/cost ratio. Choosing an optimal angle is therefore essential to maximize the energy efficiency of the system.

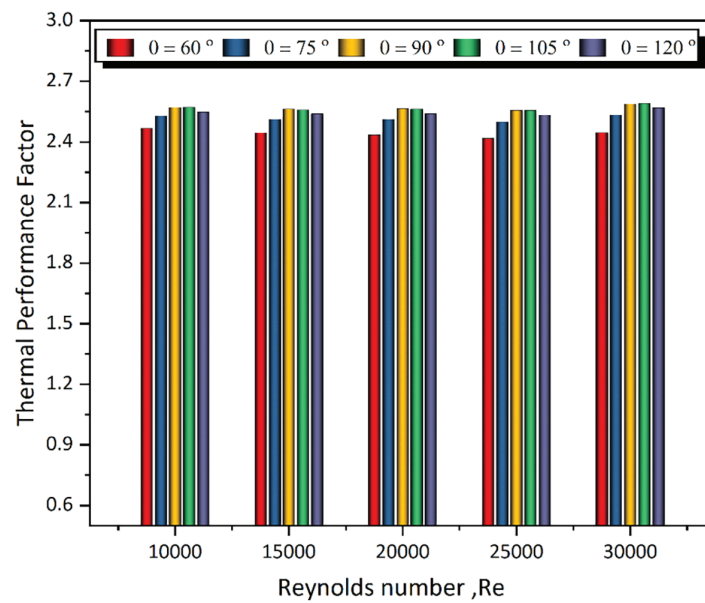


Figure 22: Effect of baffle inclination angle on *TPF* at the upper channel wall for different Reynolds numbers.

5 Conclusions

In conclusion, this in-depth numerical study analyzed the thermodynamic behavior of a turbulent air flow with forced convection in a rectangular heat exchanger channel, for Reynolds numbers ranging from 10,000 to 30,000. The innovative geometric configuration, incorporating two opposing vertical baffles and a central rectangular obstacle, was rigorously modeled using the finite volume approach and the standard $k-\epsilon$ turbulence model, after satisfactory validation against literature data.

The main conclusions of this investigation are as follows:

- The presence of the baffles and the central obstacle induce significant disturbances in the flow, manifesting as the formation of intense recirculation zones downstream of each obstacle. These zones, characterized by negative axial velocities and increased fluid residence time, actively contribute to improving mixing.
- Analysis of the axial velocity profiles revealed that the most pronounced disturbance and the most significant flow acceleration occurred downstream of the central obstacle and as it approached the second baffle, indicating a localized intensification of hydrodynamic phenomena.
- It was demonstrated that this turbulator configuration, particularly the addition of the central rectangular obstacle, significantly increases flow turbulence and promotes increased fluid mixing. This is essential for maximizing contact between the hot and cold zones.
- All of these hydrodynamic changes induced by the baffles and the central obstacle directly lead to a substantial improvement in heat exchange within the channel. The recirculation and turbulence zones act as effective mechanisms for breaking the thermal boundary layer and homogenizing the fluid temperature.
- Overall, the results confirm the significant and positive impact of baffles combined with a central obstacle on the heat exchange rate of the system, offering promising prospects for optimizing the design of compact and efficient heat exchangers.
- The *TPF* value for the geometry of Case A is higher than that of Case B.
- The angle $\theta = 90^\circ$ is more efficient than the other angle.

Generally, the study presented a proof-of-concept hydrodynamic and thermal strategy in evaporator tubes, and although modern fabrication techniques may allow such features, their practical application and cost-effectiveness should be evaluated in future work.

Acknowledgement: The present study was conducted by the National School of Applied Sciences of Al Hoceima (ENSAH), the National School of Applied Sciences of Marrakech (ENSAM), and the Private University of Fes (UPF). The numerical calculations in this paper have been done on the supercomputing system in the Big Data Center at the laboratory LSIA, Department of Civil, Energy, and Environmental Engineering of Abdelmalek Essaâdi University, Morocco.

Funding Statement: The authors received no specific funding for this study.

Author Contributions: Omar Ghoulam: Writing—original draft, Validation, Software. Hind Talbi: Methodology, Writing, Investigation. Kamal Amghar: Writing—original draft, Conceptualization, Software, Supervision. Hamza Faraji: Writing—review & editing, Methodology. Saloua Senhaji: Writing—review & editing, Methodology. Ismael Driouch: Methodology, Investigation, Supervision. All authors reviewed and approved the final version of the manuscript.

Availability of Data and Materials: No data and Materials were used for the research described in the article.

Ethics Approval: Not applicable.

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

Nomenclature

D_h	Hydraulic diameter	[m]
k	Turbulent kinetic energy	$[m^2/s^2]$
P	Pressure	[Pa]
ΔP	Pressure drop	[Pa]
P_{atm}	Atmospheric pressure	[Pa]
TEF	Thermal performance enhancement factor	
Re	Reynolds number	
S_ϕ	Limit of source for the general variable	
T	Temperature	[K]
T_w	Wall temperature	[K]
U_0	Inlet velocity	[m/s]
u, v	Air velocity in the x, y direction	[m/s]
f/f_0	Normalized friction factor	
Nu/Nu_0	Normalize Nusselt number	
λ	Thermal conductivity of fluid	$[W/m \cdot K]$
μ	Molecular viscosity	$[kg/m \cdot s]$
Γ	Diffusion coefficient	
ρ	Density of the air	$[kg/m]$
h	Heat transfer coefficient	$[W/m^2 \cdot K]$
ε	Dissipation rate of turbulence energy	$[m^2/s]$

Subscripts

atm	Atmospheric	$[1 \text{ atm} = 105 \text{ Pa}]$
x, y	Cartesian coordinate	[m]
in, out	Inlet, outlet of the test section	
w	Wall	

References

1. Lei YG, He YL, Chu P, Li R. Design and optimization of heat exchangers with helical baffles. *Chem Eng Sci*. 2008;63(17):4386–95. doi:10.1016/j.ces.2008.05.044.
2. Menasria F, Zedairia M, Moummi A. Numerical study of thermohydraulic performance of solar air heater duct equipped with novel continuous rectangular baffles with high aspect ratio. *Energy*. 2017;133:593–608. doi:10.1016/j.energy.2017.05.002.
3. Gururatana S, Skullong S. Experimental investigation of heat transfer in a tube heat exchanger with airfoil-shaped insert. *Case Stud Therm Eng*. 2019;14(6):100462. doi:10.1016/j.csite.2019.100462.
4. Tang X, Zhu D. Experimental and numerical study on heat transfer enhancement of a rectangular channel with discontinuous crossed ribs and grooves. *Chin J Chem Eng*. 2012;20(2):220–30. doi:10.1016/S1004-9541(12)60382-6.
5. Ghoulam O, Alla M, Rharrabi Y, Amghar K, Driouch I, Bouali H. Numerical study of heat transfer phenomenon for thermal management along the heat exchanger. In: *Proceedings of the 4th International Conference on Electronic Engineering and Renewable Energy Systems—Volume 2*. Singapore, Singapore: Springer Nature; 2025. p. 511–9. doi:10.1007/978-981-97-9975-6_48.
6. Wang F, Zhang J, Wang S. Investigation on flow and heat transfer characteristics in rectangular channel with drop-shaped pin fins. *Propuls Power Res*. 2012;1(1):64–70. doi:10.1016/j.jprr.2012.10.003.
7. Patankar SV, Liu CH, Sparrow EM. Fully developed flow and heat transfer in ducts having streamwise-periodic variations of cross-sectional area. *J Heat Transf*. 1977;99(2):180–6. doi:10.1115/1.3450666.
8. Chandra PR, Alexander CR, Han JC. Heat transfer and friction behaviors in rectangular channels with varying number of ribbed walls. *Int J Heat Mass Transf*. 2003;46(3):481–95. doi:10.1016/S0017-9310(02)00297-1.
9. Chand S, Chand P, Kumar Ghritlahre H. Thermal performance enhancement of solar air heater using louvered fins collector. *Sol Energy*. 2022;239(8):10–24. doi:10.1016/j.solener.2022.04.046.
10. Mohit MK, Gupta R. Numerical investigation of the performance of rectangular micro-channel equipped with micro-pin-fin. *Case Stud Therm Eng*. 2022;32:101884. doi:10.1016/j.csite.2022.101884.
11. Boonloi A, Jedsadaratanachai W. CFD analysis on heat transfer characteristics and fluid flow structure in a square duct with modified wavy baffles. *Case Stud Therm Eng*. 2022;29:101660. doi:10.1016/j.csite.2021.101660.
12. Sriromreun P, Thianpong C, Promvonge P. Experimental and numerical study on heat transfer enhancement in a channel with Z-shaped baffles. *Int Commun Heat Mass Transf*. 2012;39(7):945–52. doi:10.1016/j.icheatmasstransfer.2012.05.016.
13. Handoyo EA, Ichsani D, Prabowo, Sutardi. Numerical studies on the effect of delta-shaped obstacles' spacing on the heat transfer and pressure drop in v-corrugated channel of solar air heater. *Sol Energy*. 2016;131(5/6):47–60. doi:10.1016/j.solener.2016.02.031.
14. Peng D, Zhang X, Dong H, Lv K. Performance study of a novel solar air collector. *Appl Therm Eng*. 2010;30(16):2594–601. doi:10.1016/j.applthermaleng.2010.07.010.
15. Kumar A, Kim MH. Convective heat transfer enhancement in solar air channels. *Appl Therm Eng*. 2015;89:239–61. doi:10.1016/j.applthermaleng.2015.06.015.
16. Promvonge P, Sripattanapipat S, Tamna S, Kwankaomeng S, Thianpong C. Numerical investigation of laminar heat transfer in a square channel with 45° inclined baffles. *Int Commun Heat Mass Transf*. 2010;37(2):170–7. doi:10.1016/j.icheatmasstransfer.2009.09.010.
17. Mokhtari M, Gerdroodbary MB, Yeganeh R, Fallah K. Numerical study of mixed convection heat transfer of various fin arrangements in a horizontal channel. *Eng Sci Technol Int J*. 2017;20(3):1106–14. doi:10.1016/j.jestch.2016.12.007.
18. Mousavi SS, Hooman K. Heat and fluid flow in entrance region of a channel with staggered baffles. *Energy Convers Manag*. 2006;47(15–16):2011–9. doi:10.1016/j.enconman.2005.12.018.
19. Yongsiri K, Eiamsa-ard P, Wongcharee K, Eiamsa-ard S. Augmented heat transfer in a turbulent channel flow with inclined detached-ribs. *Case Stud Therm Eng*. 2014;3(10):1–10. doi:10.1016/j.csite.2013.12.003.
20. Karwa R, Maheshwari BK. Heat transfer and friction in an asymmetrically heated rectangular duct with half and fully perforated baffles at different pitches. *Int Commun Heat Mass Transf*. 2009;36(3):264–8. doi:10.1016/j.icheatmasstransfer.2008.11.005.

21. Kamali R, Binesh AR. The importance of rib shape effects on the local heat transfer and flow friction characteristics of square ducts with ribbed internal surfaces. *Int Commun Heat Mass Transf.* 2008;35(8):1032–40. doi:10.1016/j.icheatmasstransfer.2008.04.012.
22. Yu C, Zhang H, Zeng M, Wang R, Gao B. Numerical study on turbulent heat transfer performance of a new compound parallel flow shell and tube heat exchanger with longitudinal vortex generator. *Appl Therm Eng.* 2020;164:114449. doi:10.1016/j.applthermaleng.2019.114449.
23. Nakhchi ME, Esfahani JA, Kim KC. Numerical study of turbulent flow inside heat exchangers using perforated louvered strip inserts. *Int J Heat Mass Transf.* 2020;148:119143. doi:10.1016/j.ijheatmasstransfer.2019.119143.
24. Lv JY, Liu ZC, Liu W. Active design for the tube insert of center-connected deflectors based on the principle of exergy destruction minimization. *Int J Heat Mass Transf.* 2020;150(7):119260. doi:10.1016/j.ijheatmasstransfer.2019.119260.
25. Nidhul K, Kumar S, Yadav AK, Anish S. Enhanced thermo-hydraulic performance in a V-ribbed triangular duct solar air heater: CFD and exergy analysis. *Energy.* 2020;200(2):117448. doi:10.1016/j.energy.2020.117448.
26. Bazdidi-Tehrani F, Naderi-Abadi M. Numerical analysis of laminar heat transfer in entrance region of a horizontal channel with transverse fins. *Int Commun Heat Mass Transf.* 2004;31(2):211–20. doi:10.1016/S0735-1933(03)00226-4.
27. Tsay YL, Chang TS, Cheng JC. Heat transfer enhancement of backward-facing step flow in a channel by using baffle installation on the channel wall. *Acta Mech.* 2005;174(1):63–76. doi:10.1007/s00707-004-0147-5.
28. Roetzel W, Lee DW. Effect of baffle/shell leakage flow on heat transfer in shell-and-tube heat exchangers. *Exp Therm Fluid Sci.* 1994;8(1):10–20. doi:10.1016/0894-1777(94)90068-x.
29. Ko KH, Anand NK. Use of porous baffles to enhance heat transfer in a rectangular channel. *Int J Heat Mass Transf.* 2003;46(22):4191–9. doi:10.1016/S0017-9310(03)00251-5.
30. Karwa R, Maheshwari BK, Karwa N. Experimental study of heat transfer enhancement in an asymmetrically heated rectangular duct with perforated baffles. *Int Commun Heat Mass Transf.* 2005;32(1–2):275–84. doi:10.1016/j.icheatmasstransfer.2004.10.002.
31. Liu J, Xie G, Simon TW. Turbulent flow and heat transfer enhancement in rectangular channels with novel cylindrical grooves. *Int J Heat Mass Transf.* 2015;81:563–77. doi:10.1016/j.ijheatmasstransfer.2014.10.021.
32. Ghoulam O, Talbi H, Amghar K, Amrani AI, Charef A, Driouch I. Heat transfer improvement in turbulent flow using detached obstacles in heat exchanger duct. *Int J Thermofluids.* 2025;27:101225. doi:10.1016/j.ijft.2025.101225.
33. Amghar K, Louhibi MA, Salhi N, Salhi M. Numerical simulation of forced convection turbulent in a channel with transverse baffles. *J Mater Environ Sci.* 2017;8:1417–27.
34. Kitayama S, Miyamoto K, Izutsu R, Tabuchi S, Yamada S. Numerical optimization of baffle configuration in header of heat exchanger using sequential approximate optimization. *Simul Model Pract Theory.* 2022;115:102429. doi:10.1016/j.simpat.2021.102429.
35. Ismael MA. Forced convection in partially compliant channel with two alternated baffles. *Int J Heat Mass Transf.* 2019;142:118455. doi:10.1016/j.ijheatmasstransfer.2019.118455.
36. Mo SH, Li YQ, Yuan WH. Optimizing thermal performance of a premixed hydrogen/air fueled micro-combustor with baffles. *J Cent South Univ.* 2023;30(12):4285–98. doi:10.1007/s11771-023-5505-3.
37. Khadang A, Nazari M, Maddah H, Ahmadi MH, Sharifpur M. Experimental study and neural network-based prediction of thermal performance of applying baffles and nanofluid in the double-pipe heat exchangers. *J Therm Anal Calorim.* 2024;149(9):4239–59. doi:10.1007/s10973-024-12969-0.
38. Amghar K, Ameer H, Bouali H, Salhi N. Numerical study of thermal and dynamic structure of a laminar flow in solar collector. *Mater Today Proc.* 2021;45:7501–6. doi:10.1016/j.matpr.2021.02.258.
39. Bi YZ, Wang DP, Fu XL, Lin YX, Sun XP, Jiang ZY. Optimal array layout of cylindrical baffles to reduce energy of rock avalanche. *J Mt Sci.* 2022;19(2):493–512. doi:10.1007/s11629-021-6916-y.
40. Yaseen DT, Ismael MA. Structural mechanics of flexible baffle used in enhancing heat transfer of power law fluids in channel-trapezoidal cavity. *Exp Tech.* 2023;47(1):37–46. doi:10.1007/s40799-022-00554-9.

41. Mazdak S, Ali Sheikhzadeh G, Fattahi A. Numerical analysis of a heat exchanger with curved segmental baffle and Cassini oval cross-section tubes in various bundle arrangements. *J Therm Anal Calorim.* 2023;148(16):8459–76. doi:10.1007/s10973-023-12062-y.
42. Maouedj R, Menni Y, Inc M, Chu Y-M, Ameer H, Lorenzini G. Simulating the turbulent hydrothermal behavior of oil/MWCNT nanofluid in a solar channel heat exchanger equipped with vortex generators. *Comput Model Eng Sci.* 2021;126(3):855–89. doi:10.32604/cmesci.2021.014524.
43. Ahmad H, Alam MN, Omri M. New computational results for a prototype of an excitable system. *Results Phys.* 2021;28(9):104666. doi:10.1016/j.rinp.2021.104666.
44. Saravanakumar PT, Somasundaram D, Matheswaran MM. Exergetic investigation and optimization of arc shaped rib roughened solar air heater integrated with fins and baffles. *Appl Therm Eng.* 2020;175(1):115316. doi:10.1016/j.applthermaleng.2020.115316.
45. Peiravi MM, Alinejad J. Hybrid conduction, convection and radiation heat transfer simulation in a channel with rectangular cylinder. *J Therm Anal Calorim.* 2020;140(6):2733–47. doi:10.1007/s10973-019-09010-0.
46. Demartini LC, Vielmo HA, Möller SV. Numeric and experimental analysis of the turbulent flow through a channel with baffle plates. *J Braz Soc Mech Sci Eng.* 2004;26(2):153–9. doi:10.1590/s1678-58782004000200006.
47. Menni Y, Ghazvini M, Ameer H, Kim M, Ahmadi MH, Sharifpur M. Combination of baffling technique and high-thermal conductivity fluids to enhance the overall performances of solar channels. *Eng Comput.* 2022;38(1):607–28. doi:10.1007/s00366-020-01165-x.
48. Parneix S, Durbin PA, Behnia M. Computation of 3-D turbulent boundary layers using the V2F model. *Flow Turbul Combust.* 1998;60(1):19–46. doi:10.1023/A:1009986925097.
49. Behnia M, Parneix S, Shabany Y, Durbin PA. Numerical study of turbulent heat transfer in confined and unconfined impinging jets. *Int J Heat Fluid Flow.* 1999;20(1):1–9. doi:10.1016/S0142-727X(98)10040-1.
50. Nasiruddin, Kamran Siddiqui MH. Heat transfer augmentation in a heat exchanger tube using a baffle. *Int J Heat Fluid Flow.* 2007;28(2):318–28. doi:10.1016/j.ijheatfluidflow.2006.03.020.
51. Arora O, Cosials KF, Vaghetto R, Hassan YA. Pressure drop and friction factor study for an airfoil-fin printed circuit heat exchanger using experimental and numerical techniques. *Int J Heat Fluid Flow.* 2023;101(7):109137. doi:10.1016/j.ijheatfluidflow.2023.109137.
52. Sheikhzad Saravani M, Mohaddes Deylami H, Naghashadegan M. Investigating the effect of physical parameters of a flexible vortex generator on the flow field and heat transfer inside a microchannel. *Therm Sci Eng Prog.* 2024;54:102824. doi:10.1016/j.tsep.2024.102824.