THE 3-D INVESTIGATION OF BAFFLE ROTATION EFFECTS ON HEAT TRANSFER PERFORMANCE IN COUNTERFLOW HEAT EXCHANGERS

by

Basma BAKIR [a]***, Imane ALIOUANE*** [a]***, Younes MENNI*** [a,b]***, Mustafa BAYRAM*** [c*]***, Omolayo M. IKUMAPAYI*** [d]***, Abiodun BAYODE*** [d]***, and Tin Tin TING*** [e,f]

[a] Department of Mechanical Engineering, Institute of Technology, University Center Salhi Ahmed Naama, Naama, Algeria
[b] College of Technical Engineering, National University of Science and Technology, Dhi Qar, Iraq
[c] Department of Computer Engineering, Biruni University, Istanbul, Turkey
[d] Department of Mechanical Engineering, Northwest University, Potchefstroom, South Africa
[e] Faculty of Data Science and Information Technology, INTI International University, Nilai, Malaysia
[f] School of Information Technology, UNITAR International University, Selangor, Malaysia

Original scientific paper
https://doi.org/10.2298/TSCI2504113B

This study numerically investigates the thermal and hydraulic performance of a 3-D counterflow heat exchanger equipped with internal baffles under laminar flow conditions. The finite element method is employed to simulate the effects of baffle rotation angle (θ = 0°, 30°, 45°, and 60°) and Reynolds number (Re = 500-2000). The heat exchanger consists of two adjacent rectangular channels separated by a thin aluminum wall, with hot and cold water flowing in opposite directions. Key performance indicators, velocity, pressure drop, temperature difference, ΔT, and thermal efficiency, are evaluated for both fluid streams. Results reveal that increasing the baffle rotation angle enhances thermal performance up to an optimal point (θ = 30°), beyond which further increases reduce heat transfer due to weaker flow disturbances. Conversely, pressure drop decreases steadily with higher baffle angles, with the lowest resistance observed at θ = 60°. The Reynolds number strongly affects both thermal and hydraulic behavior, with higher values resulting in lower ΔT and efficiency due to shorter residence times.

Key words: *baffle, rotation, counterflow, laminar flow, efficiency*

Introduction

Heat exchangers are essential in energy systems, widely used in applications such as power generation, thermal storage, and electronic cooling. Counterflow configurations are particularly valued for their high thermal efficiency and compact design [1, 2]. Recent advances aim to improve heat transfer while reducing pressure drop and material usage. Umer *et al*. [1] and Li *et al*. [3] emphasized the role of material selection and structural optimization in solid-gas and foam-filled exchangers, respectively.

* Corresponding author, e-mail: mustafabayram@biruni.edu.tr

Passive enhancement methods, including turbulators and modified geometries, have been extensively explored. Tavousi *et al*. [2] improved nanofluid-based exchangers using innovative turbulators, while Zhao [4, 5] introduced entropy-based methods for thermodynamic assessment. Geometric enhancements, such as twisted and conical turbulators proposed by Rinik *et al*. [6] and Haddadvand *et al.* [7], were shown to increase turbulence and thermal uniformity. Computational studies by Zhao *et al*. [8] and Chaquet and del Sastre [9] further examined complex thermal-fluid behavior.

Nanofluids have also emerged as effective additives for enhancing convective heat transfer. Kadhim *et al*. [10] and Alhulaifi [11] highlighted their benefits in double-pipe exchangers, while Ahmad and Alsalmah [12] combined AI with modelling to predict solar-assisted exchanger performance. Advances in materials and fabrication, such as binder jetting for Inconel exchangers [13] and empirical-experimental optimization of heat pipes [14], have also improved performance, supported by innovations in micro-channel and dimpled designs [15-17]. Moreover, contributions by Menni *et al*. [18-20], Ameur *et al*. [21], Djeffal *et a*l. [22], Tahrour *et al*. [23], Salmi *et al.* [24], and Chamkha *et al*. [25] have shown the effectiveness of baffle-induced turbulence and geometry optimization in enhancing heat transfer in air channels and finned-tube systems.

This study addresses the need for compact, energy-efficient heat exchangers in applications such as electronic cooling and HVAC systems. Enhancing thermal performance in laminar flow requires effective geometric modifications that minimize pressure loss. Baffle rotation is a promising strategy, yet its impact in counterflow configurations remains underexplored. This work numerically investigates the effects of baffle rotation angle and Reynolds number, Re on the thermal and hydraulic performance of a 3-D counterflow heat exchanger, aiming to guide the design of efficient systems for low Reynolds number applications.

Numerical model and methodology

This study numerically analyzes the thermal and hydraulic performance of a 3-D counterflow heat exchanger with internal baffles, fig. 1. It features two adjacent rectangular channels (1 m long, 0.08 m internal diameter, and 1 mm wall thickness) separated by a thin aluminum wall (1 mm thickness) allowing heat transfer without fluid mixing.

To enhance heat transfer, stationary rectangular baffles are mounted on the inner surfaces of both channels. These baffles are oriented at rotation angles of 0°, 30°, 45°, and 60° relative to the flow direction, promoting secondary flows and boundary-layer disruption. The working fluid in both channels is water, with inlet temperatures of 373.15 K (hot fluid) and 293.15 K (cold fluid). The Reynolds number is varied from 500-2000, ensuring laminar flow throughout.

The physical model assumes steady-state, incompressible flow with constant thermo-physical properties. Conjugate heat transfer is modeled through the aluminum wall, with perfect thermal contact at fluid-solid interfaces. Boundary conditions include uniform inlet velocity based on the selected Reynolds number, zero-gradient outlet conditions, no-slip walls, and adiabatic external surfaces.

Based on the aforementioned assumptions, the governing equations describe fluid-flow and heat transfer in both the fluid channels and the separating solid wall.

Continuity equation:

$$\nabla \vec{\mathrm{u}} = 0 \tag{1}$$

Momentum equations:

$$\rho\left(\vec{\mathrm{u}}\nabla\right)\vec{\mathrm{u}} = -\nabla p + \mu\nabla^2\vec{\mathrm{u}} \tag{2}$$

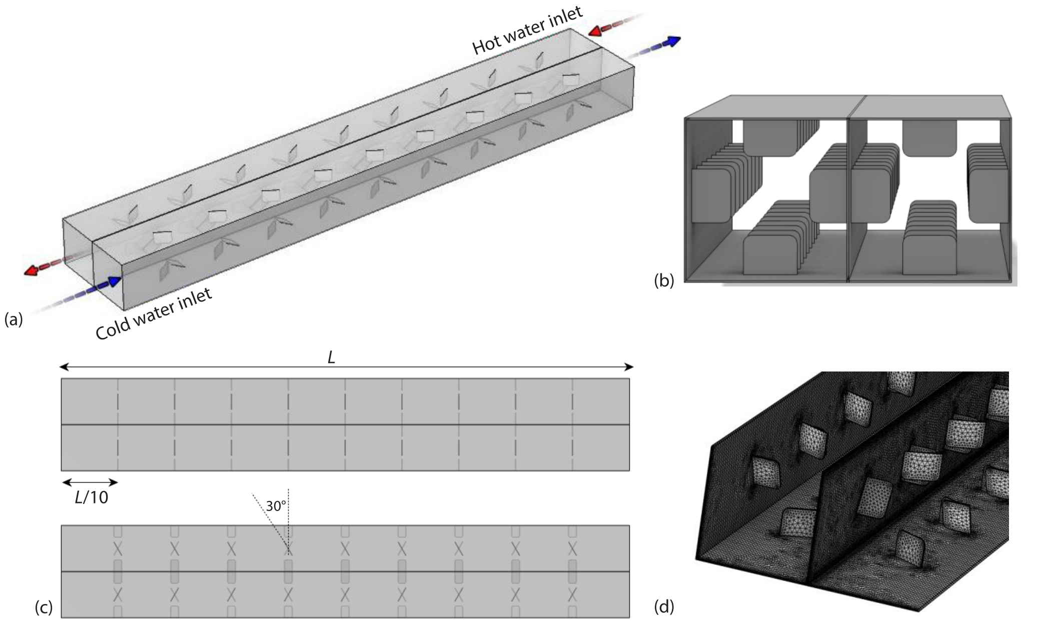

Figure 1. Schematic representation of the studied baffled counterflow heat exchanger; (a) side view, (b) front view, (c) dimensional specifications, and (d) computational mesh

Energy equation in the fluid domain:

$$\rho C_p \left(\vec{u}\nabla T\right) = k\nabla^2 T \tag{3}$$

Energy equation in the solid wall:

$$k_w \nabla^2 T_w = 0 \tag{4}$$

Conjugate heat transfer conditions:

– Temperature continuity

$$T_f = T_w \tag{5}$$

– Heat flux continuity

$$-k_f \left(\frac{\partial T}{\partial n}\right)_f = -k_w \left(\frac{\partial T}{\partial n}\right)_w \tag{6}$$

where $\vec{u} = (u, v, w)$ is the velocity components in the *x*-, *y*-, and *z*-directions. The flow equations involve fluid density, ρ, dynamic viscosity, μ, pressure, p, and convective acceleration, $\vec{u}\,\nabla$, resolved in all three directions. The thermal model includes fluid properties such as specific heat, C_p, thermal conductivity, k, and temperature, T. In the solid wall, T_w represents the temperature field, and k_w is the thermal conductivity of aluminum.

Thermal efficiency:

$$\varepsilon = \left(\frac{T_{c,o} - T_{c,i}}{T_{h,i} - T_{c,i}}\right) \times 100\% \tag{7}$$

where $T_{c,o}$ and $T_{c,i}$ are the outlet and inlet temperatures of the cold fluid, respectively, and $T_{h,i}$ is the inlet temperature of the hot fluid [26].

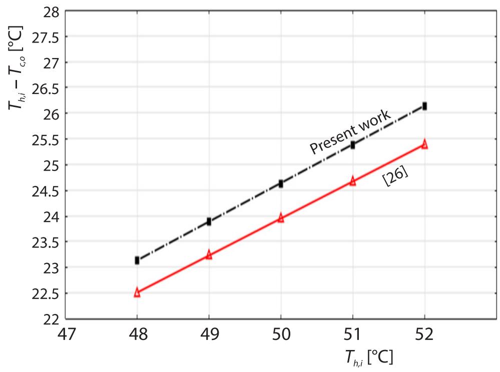

Figure 2. Model validation under same geometrical and operational conditions

Numerical simulations are performed using the finite element method implemented in COMSOL Multiphysics under steady-state conditions. The computational domain is discretized using an unstructured tetrahedral mesh consisting of 1404894 elements and 261143 nodes, with refined regions near baffles, walls, and thermal gradients. The average element quality is 0.575, and the minimum element quality is 0.0393. A segregated solver is used with convergence criteria set to 10^{-6} for all residuals, ensuring stability and accuracy. The numerical model was validated against Laskowski [26] using a counterflow heat exchanger with coaxial channels under matching thermal and flow conditions. As shown in fig. 2, the temperature distribution closely agrees with the reference results, confirming the model's accuracy and reliability.

Findings and analysis

Flow field visualization

Figure 3 presents the velocity contours in the counterflow heat exchanger at a low Reynolds number (Re = 500) for four different baffle rotation angles: $\theta = 0°$, 30°, 45°, and 60°. At $\theta = 0°$, the baffles are oriented perpendicular to the flow, generating strong flow obstruction and high velocity jets between the baffles. This configuration leads to pronounced acceleration in the flow channels and the formation of re-circulation zones downstream of each baffle. As the baffle rotation angle increases from 0° to 60°, the flow experiences less disruption due to the increasingly streamlined orientation of the baffles. Consequently, the peak velocity magnitude gradually decreases across the flow domain. At $\theta = 60°$, the baffles are more aligned with the main flow direction, minimizing resistance and resulting in a smoother, lower-velocity profile. This progressive reduction in velocity with increasing baffle angle indicates a trade-off between flow mixing intensity and pressure drop, which directly influences the heat exchanger's thermal and hydraulic performance.

Drop hydrodynamic pressure loss

Table 1 summarizes the pressure drop behavior of the counterflow heat exchanger under varying baffle angles ($\theta = 0°$, 30°, 45°, and 60°) and Reynolds numbers. For all configurations, pressure drop increases significantly with Reynolds number due to higher fluid velocities and associated wall shear stress. For example, at $\theta = 0°$, the hot channel pressure drop rises from 0.039 Pa (Re = 500) to 0.574 Pa (Re = 2000), while the cold channel increases from 0.048 Pa to 0.644 Pa, representing increases of 1372% and 1242%, respectively.

Baffle rotation angle strongly influences pressure loss. As θ increases from 0° to 60°, pressure drop consistently decreases at all Reynolds number values due to reduced flow obstruction. At Re = 2000, the hot channel pressure drop falls by 80.7% (from 0.574-0.111 Pa), and the cold channel by 79.5% (from 0.644-0.132 Pa), indicating improved hydraulic performance at higher angles.

Cold channels exhibit slightly higher pressure drops than hot ones across all cases, mainly due to the higher viscosity of cold water (293.15 K) compared to hot water (373.15 K), leading to greater viscous resistance under identical flow and geometric conditions.

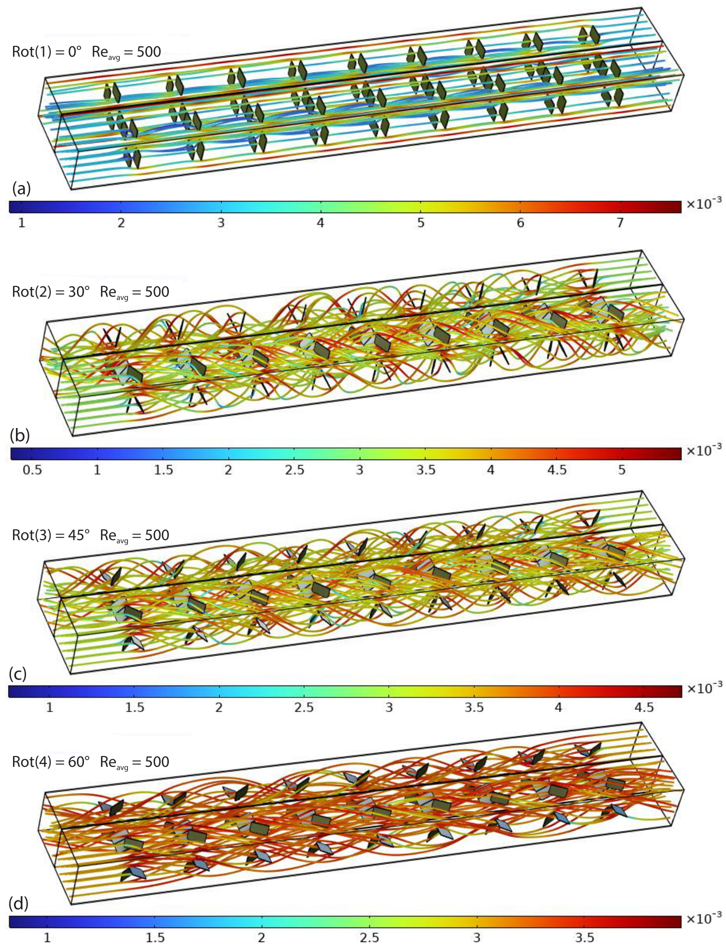

Figure 3. Velocity contours in the counterflow heat exchanger at Re = 500 for baffle angles θ; (a) 0°, (b) 30°, (c) 45°, and (d) 60°

Inlet-outlet temperature difference

As shown in tab. 2, the temperature difference, ΔT, between inlet and outlet decreases with increasing Reynolds number for all baffle angles due to shorter residence time and reduced heat transfer opportunity. At $\theta = 0°$, the hot fluid ΔT drops from 15.450 K at Re = 500 to 11.479 K at Re = 2000, while the cold fluid ΔT decreases from 14.783-11.109 K. This consistent trend across all configurations indicates that lower Reynolds numbers favor higher thermal gradients under constant inlet conditions.

Varying the baffle angle shows that thermal performance improves notably up to $\theta = 30°$. At Re = 500, ΔT increases from 15.450-25.408 K for the hot stream and from 14.783 K to 24.625 K for the cold stream. This improvement is due to enhanced secondary flow and thermal boundary-layer disruption. However, beyond 30°, ΔT begins to decline, as higher angles reduce turbulence intensity by aligning more with the main flow, which weakens vortex formation and wall mixing.

Table 1. Pressure drop in hot and cold fluid streams *vs.* Reynolds number and baffle angle

Baffle rotation angle, θ [°]	Reynolds number	Δp [Pa] – Hot fluid	Δp [Pa] – Cold fluid
0	500	0.039	0.048
	1000	0.149	0.174
	1500	0.327	0.373
	2000	0.574	0.644
30	500	0.032	0.038
	1000	0.123	0.140
	1500	0.270	0.304
	2000	0.473	0.529
45	500	0.015	0.020
	1000	0.060	0.071
	1500	0.132	0.152
	2000	0.230	0.264
60	500	0.007	0.010
	1000	0.029	0.036
	1500	0.064	0.077
	2000	0.111	0.132

Table 2. Effect of baffle angle and Reynolds number on ΔT in hot and cold channels

Baffle rotation angle, θ [°]	Reynolds number	ΔT [K] – Hot fluid	ΔT [K] – Cold fluid
0	500	15.450	14.783
	1000	13.130	12.619
	1500	12.083	11.663
	2000	11.479	11.109
30	500	25.408	24.625
	1000	24.441	23.621
	1500	23.943	23.138
	2000	23.632	22.847
45	500	23.562	22.787
	1000	22.528	21.765
	1500	22.019	21.270
	2000	21.699	20.966
60	500	20.882	20.197
	1000	19.669	19.010
	1500	19.123	18.483
	2000	18.789	18.163

In all cases, the hot stream exhibits slightly higher ΔT than the cold stream, despite symmetric geometry and boundary conditions. This asymmetry arises from the higher inlet temperature of the hot fluid, which creates a stronger thermal gradient and enhances heat loss near the inlet. Overall, the 30° baffle configuration offers the best thermal performance, striking a balance between efficient heat transfer and manageable pressure drop.

Thermal efficiency trends

Table 3 shows that thermal efficiency decreases with increasing Reynolds number across all baffle angles. This decline is primarily due to shorter residence time at higher flow rates, which limits heat transfer between streams despite greater convective potential. For instance, at $\theta = 30°$, efficiency drops from 32.252% at Re = 500 to 29.889% at Re = 2000 (a 7.3% decrease), while at $\theta = 0°$, it falls more sharply from 19.345-14.519% (a 24.9% drop). These results indicate that lower Reynolds numbers are more favorable for maximizing thermal performance in laminar counterflow conditions.

The baffle rotation angle strongly influences thermal efficiency. At a fixed Reynolds number, increasing θ from 0° to 30° significantly improves performance due to enhanced flow mixing and thermal boundary-layer disruption. For example, at Re = 500, efficiency increases from 19.345% at 0° to 32.252% at 30°, a 66.7% gain. However, beyond 30°, further increases in baffle angle reduce efficiency slightly. This is attributed to reduced secondary flow generation as baffles become more aligned with the main flow, weakening convective mixing and overall heat transfer.

Despite this decline beyond 30°, baffle angles of 45°, and 60° still outperform the baseline 0° case. At Re = 2000, efficiency drops from 29.889% (30°) to 27.417% (45°), and 23.743% (60°), showing diminishing returns at higher angles. These findings confirm that the 30° baffle orientation offers the optimal balance between heat transfer enhancement and hydraulic performance, achieving the highest thermal efficiency across the studied range.

Table 3. Variation of thermal efficiency with flow and baffle configuration

Baffle rotation angle, θ [°]	Reynolds number	Efficiency, ε [%]
0	500	19.345
	1000	16.501
	1500	15.245
	2000	14.519
30	500	32.252
	1000	30.917
	1500	30.275
	2000	29.889
45	500	29.839
	1000	28.477
	1500	27.820
	2000	27.417
60	500	26.437
	1000	24.865
	1500	24.166
	2000	23.743

Conclusions

This study examined the thermal and hydraulic performance of a 3-D counterflow heat exchanger with internal baffles at various rotation angles (0° to 60°) and Reynolds numbers (500-2000). Results showed that increasing the Reynolds number leads to higher pressure drops and lower thermal efficiency due to reduced residence time. In contrast, baffle rotation significantly influenced flow and heat transfer. A 30° rotation angle provided the best balance, achieving the highest thermal efficiency while maintaining moderate pressure loss.

Higher baffle angles (45° and 60°) reduced pressure drop but weakened thermal performance, indicating a trade-off between mixing and hydraulic resistance. The findings suggest that a 30° baffle angle is optimal for laminar flow applications. Future work should consider transient effects, varying fluid properties, and experimental validation enhance the model's applicability.

References

[1] Umer. M., *et al*., Heat Transfer Investigation in a Solid-Gas Counterflow Heat Exchanger for Thermochemical Energy Storage Applications, *Applied Thermal Engineering*, *275* (2025), 126743
[2] Tavousi, E., *et al*., Effect of Novel Turbulators on the Hydrothermal Performance of Counterflow Double Tube Heat Exchanger Using Nanofluids, *International Journal of Heat and Fluid-Flow*, *107* (2024), 109427
[3] Li, Z. H., *et al.*, Multi-Objective Optimization of the Foam Filled Counterflow Double-Pipe Heat Exchanger under High Temperature Condition, *International Journal of Heat and Mass Transfer*, *223* (2024), 125230
[4] Zhao, B., Entropy Transfer Efficiency-Effectiveness Method for Heat Exchangers – Part 2: Temperature-Conductance-Entropy Load Diagram and Temperature-Heat Load Diagram with Thermal Resistance Limits, *Energy*, *314* (2025), 134130
[5] Zhao. B., Entropy Transfer Efficiency-Effectiveness Method for Heat Exchangers – Part 1: Local Entropy Generation Number and Operation Performance Limits, *Energy*, *304* (2024), 132133
[6] Rinik, R. A., *et al.*, Enhancement of Heat Transfer Using Elliptical Twisted Inner Pipe with Convergent Conical Ring Turbulator for Turbulent Flow in Double Pipe Heat Exchanger, *International Journal of Thermal Sciences*, *210* (2025), 109558
[7] Haddadvand, R., *et al.*, Numerical Study on the Influence of Flow Direction and Fluid Type on Heat Transfer and Pressure Drop in a Two-Tube Spiral Heat Exchanger with Innovative Conical Turbulators, *Case Studies in Thermal Engineering*, *61* (2024), 104933
[8] Zhao. T., *et al.*, Coupled Dynamic Analysis of an Underground Sewage Pipe-Line Circumferential Spiral-Wound Heat Exchanger, *Results in Engineering*, *26* (2025), 105641
[9] Chaquet. J. M., del Sastre, P. G., Temperature Fields Calculation in Heat Exchangers Using the Finite Element Method, *Finite Elements in Analysis and Design*, *249* (2025), 104385
[10] Kadhim. S. A., *et al.*, Feasibility Review of Using Copper Oxide Nanofluid to Improve Heat Transfer in the Double-Tube Heat Exchanger, *Results in Engineering*, *24* (2024), 103227
[11] Alhulaifi. A. S., Numerical Characterization of the Performance of Counter-Flow Double-Pipe Heat Exchanger Using Nanofluids for Laminar Flow Regime, *Results in Engineering*, *25* (2025), 104061
[12] Ahmad, S., Alsalmah, H. A., Numerical and Artificial Intelligent Analysis of Double Pipe Heat Exchanger Exposed to Solar Irradiation for Sustainable Energy Solutions, *Results in Engineering*, *27* (2025), 105994
[13] Du, W., *et al.*, Development of a High-Temperature Inconel 625 Heat Exchanger by Model Design and Binder Jetting Additive Manufacturing, *Materials & Design*, *251* (2025), 113333
[14] Kang, S., *et al*., Empirical Modelling and Experimental Validation of Gas-To-Liquid Heat Pipe Heat Exchanger with Baffles, *Energy*, *303* (2024), 131972
[15] Fan, Y., *et al*., Experimental Study on Boiling Heat Transfer and Bubble Characteristics in Counterflow Minichannel Heat Sink with Vapor Transport Structures, *Applied Thermal Engineering*, *278* (2025), 127204
[16] Rao, Y., *et al*., Experimental and Numerical Study of Thermal Performance of an Additively Manufactured Micro-Channel Heat Exchanger, *Thermal Science and Engineering Progress*, *62* (2025), 103572
[17] Xu, G., *et al*., Experimental and Numerical Investigation of the Frictional Performance of the Single-Phase Flow through a Dimple Plate Heat Exchanger, *International Journal of Thermal Sciences*, *217* (2025), 110081

[18] Menni, Y., *et al*., Heat Transfer in Air-Flow Past a Bottom Channel Wall-Attached Diamond-Shaped Baffle-Using a CFD Technique, *Periodica Polytechnica Mechanical Engineerin*g, *63* (2019), 2, pp. 100-112
[19] Menni, Y., Azzi, A., Design and Performance Evaluation of Air Solar Channels with Diverse Baffle Structures, *Computational Thermal Sciences*, *10* (2018), 3, pp. 225-249
[20] Menni, Y., *et al*., Effects of Two-Equation Turbulence Models on the Convective Instability in Finned Channel Heat Exchangers, *Case Studies in Thermal Engineering*, *31* (2022), 101824
[21] Ameur, H., *et al.*, Enhancement of the Cooling of Shear-Thinning Fluids in Channel Heat Exchangers by Using the V-Baffling Technique, *Thermal Science and Engineering Progress*, *18* (2020), 100534
[22] Djeffal, F., *et al.*, Numerical Investigation of Thermal-Flow Characteristics in Heat Exchanger with Various Tube Shapes, *Applied Sciences*, *11* (2021), 20, 9477
[23] Tahrour, F., *et al*., The 3-D Numerical Study and Comparison of Thermal-Flow Performance of Various Annular Finned-Tube Designs, *Journal of Ocean Engineering and Science*, *8* (2023), 3, pp. 294-307
[24] Salmi, M., *et al*., The CFD-Based simulation and Analysis of Hydrothermal Aspects in Solar Channel Heat Exchangers with Various Designed Vortex Generators, *Computer Modelling in Engineering & Sciences*, *126* (2021), 1, pp. 147-173
[25] Chamkha, A. J., *et al.*, Thermal-Aerodynamic Performance Measurement of Air Heat Transfer Fluid Mechanics over S-Shaped Fins in Shell-and-Tube Heat Exchangers, *Journal of Applied and Computational Mechanics*, *7* (2021), 4, pp. 1931-1943
[26] Laskowski, R., The black Box Model of a Double-Tube Counter-Flow Heat Exchanger, *Heat and Mass Transfer*, *51* (2015), Dec., pp. 1111-1119

Paper submitted: March 2, 2025
Paper revised: May 11, 2025
Paper accepted: June 13, 20255