CFD STUDY OF A DUAL-PASSAGE SOLAR COLLECTOR WITH LONGITUDINAL AND TRANSVERSE BAFFLES FOR ENHANCED THERMAL PERFORMANCE

by

***Mohammed Amine AMRAOUI*[a], *Mohammed Ayad ALKHAFAJI*[b], *Sherzod ABDULLAEV*[c,d], *Sajad A. ZEARAH*[e], *Ali AKGUL*[f,g*], *Rabab JARRAR*[i], *Hussein SHANAK*[i], *Jihad ASAD*[i], and *Younes MENNI*[j]**

[a] Faculty of Technology, Department of Mechanical Engineering,
University Djillali LIABES Sidi-Bel-Abbes, Sidi-Bel-Abbes, Algeria
[b] College of Engineering, National University of Science and Technology, Dhi Qar, Iraq
[c] Faculty of Chemical Engineering, New Uzbekistan University, Tashkent, Uzbekistan
[d] Department of Science and Innovation,
Tashkent State Pedagogical University named after Nizami, Tashkent, Uzbekistan
[e] Scientific Research Center, Al-Ayen University, Thi-Qar, Iraq
[f] Department of Mathematics, Art and Science Faculty, Siirt University, Siirt, Turkey
[g] Department of Electronics and Communication Engineering, Saveetha School of Engineering, SIMATS, Chennai, Tamilnadu, India
[i] Department of Physics, Faculty of Applied Science, Palestine Technical University-Kadoorie, Tulkarm, Palestine
[j] Department of Technology, University Center Salhi Ahmed Naama (Ctr. Univ. Naama), Naama, Algeria

Original scientific paper
https://doi.org/10.2298/TSCI2304133A

The focus of this research is to investigate the heat transfer performance of a solar flat plate collector by utilizing CFD simulation. To accomplish this, a 3-D model of the collector with an air inlet was created using ANSYS Workbench, and the grid was generated through ANSYS ICEM, ANSYS FLUENT, and ANSYS CFX were then used to obtain comprehensive results. The primary objective of this study is to enhance the efficiency of the solar collector by introducing two fluid-flow paths and comparing the results with those reported in existing literature. These findings will aid in the development of advanced solar collector designs and promote sustainable use of solar energy. Furthermore, the insights gained from this study may inspire further research in the renewable energy technology field. Overall, this research explores the potential of improving the performance of solar flat plate collectors and sheds light on how the use of CFD simulation can facilitate the development of innovative and sustainable energy solutions.

Key words: *renewable energy, solar energy, solar collectors, CFD, mesh*

Introduction

Solar flat plate collectors are devices designed to collect and convert solar radiation into thermal energy [1]. They consist of a dark-colored flat plate that is covered by a transparent cover, typically made of glass or plastic [2]. The plate absorbs the incoming solar radiation, and the absorbed energy is transferred to a fluid circulating through tubes or ducts in the

* Corresponding author, e-mail: aliakgul00727@gmail.com

collector [3]. After being heated, the fluid has the potential to serve various purposes such as providing space heating, domestic hot water, or driving an absorption cooling system [4].

Solar flat plate collectors are commonly used in both residential and commercial applications, and they are an effective means of reducing energy costs and GHG emissions. They are particularly well-suited for applications that require low temperature heat, such as domestic hot water or space heating [2].

Extensive studies have been conducted by researchers to explore the utilization of various impediments in solar flat plate collectors, with the aim of altering fluid-flow patterns and ultimately enhancing their performance. These impediments encompass an array of configurations, including fins [5-8], baffles [9], and other structural elements [10]. By introducing these elements, the natural flow of the fluid is disrupted [11], thereby augmenting the available surface area for efficient heat transfer [12]. Through the strategic incorporation of such obstructions, it becomes possible to regulate and optimize the fluid-flow, leading to a notable increase in thermal efficiency [13] and a substantial reduction in heat loss [14].

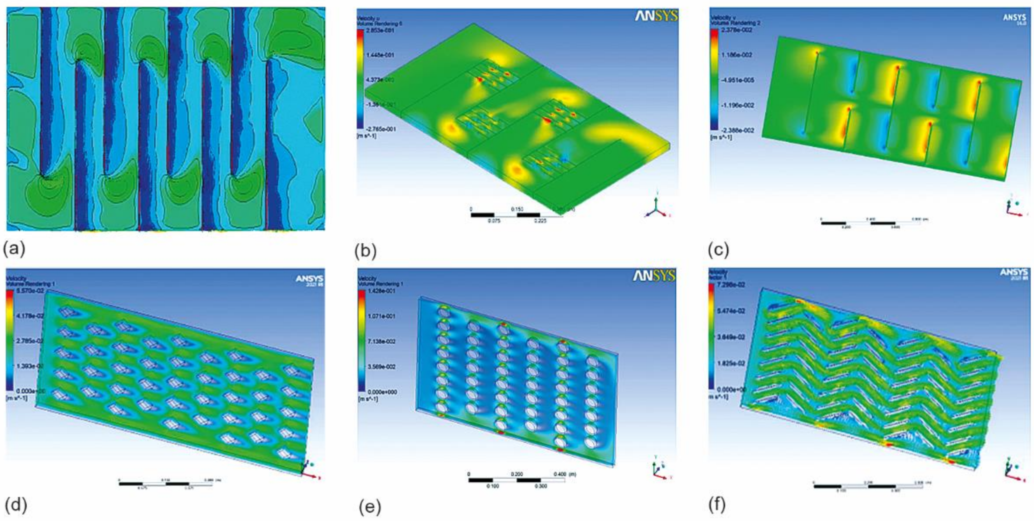

Figure 1. Solar collectors (SC) utilizing fin technology; (a) the SC featuring transverse baffles [15], (b) the SC featuring both transverse and longitudinal baffles [16], (c) baffled SC incorporating dual pathways [17], (d) the SC incorporating square-shaped baffles [18], (e) the SC with circular barriers [19], and (f) the SC featuring wing-shaped baffles [20]

Extensive research findings have demonstrated the substantial performance enhancements achieved by incorporating obstacles in solar flat plate collectors [15-20], particularly in scenarios where these collectors are exposed to fluctuating weather conditions, as depicted in fig. 1. The ongoing exploration of collector design optimization and the careful selection of suitable obstacles represent active areas of research. Continued progress in this field holds great promise for significantly expanding the utilization of solar energy in heating and cooling applications [21]. These advancements have the potential to revolutionize the energy landscape by harnessing the abundant and sustainable power of the sun to meet our heating and cooling needs.

Furthermore, independent studies have also focused on investigating fluid properties for various other applications [22-24]. These studies delve into understanding the unique characteristics of fluids and their behavior in diverse contexts.

This study aims to enhance the performance of a SC by evaluating its thermal efficiency through a qualitative analysis utilizing numerical and quantitative visualization methods. The main objective is to introduce two fluid-flow paths to the collector in order to improve its heat transfer capabilities. The results of this research provide insights into the optimal design of SC, which can lead to the development of more sustainable and efficient solar energy technologies. These outcomes have significant implications for the transition towards renewable energy sources and can contribute to the reduction of GHG emissions.

Utilizing mathematical modeling and numerical simulation to investigate phenomena

In the current study, the optimization of a SC was achieved through the introduction of obstacles and the creation of a two-passage model for air. The arrangement of baffles positioned perpendicular to the insulation in 80% of the SC width is illustrated in fig. 2. These baffles, which have the same height as the passage, are arranged in a labyrinthine pattern and are five in number. The SC itself is rectangular in shape.

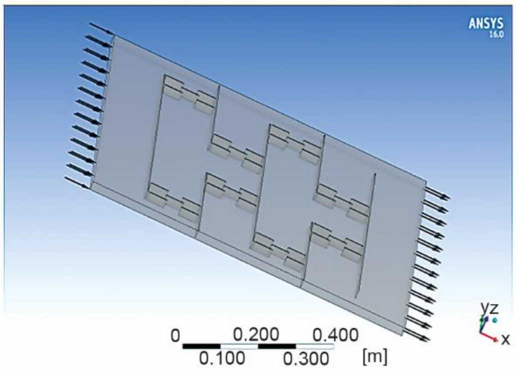

Figure 2. The SC geometry investigated in the current study

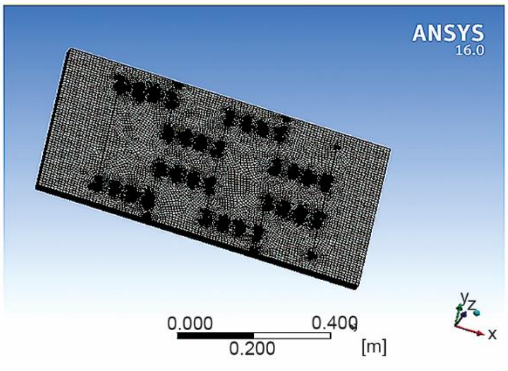

Figure 3. Mesh system utilized for numerical analysis

The SC under investigation in this study has a length of 900 mm and a width of 500 mm, with a flow path height of 25 mm. The baffles, which are rectangular in shape, have two different sizes, with the smaller ones measuring 50 × 25 mm. These smaller baffles are placed in the compartments between the larger baffles, forming a labyrinth pattern with a spacing of 150 mm between the larger baffles. There is a distance of 30 mm between the smaller baffles, and their height measures 25 mm. Previous studies, such as those conducted by Moummi [25], have informed the adoption of these particular dimensions.

Notably, the design adopted is different from that used in previous studies, which represents a new approach for optimizing SC.

It is possible to formulate the conservation of mass principle, also known as the continuity equation, in the following manner [16]:

$$\nabla(\rho\vec{\mathrm{v}}) = 0 \tag{1}$$

The principle of momentum conservation can be defined:

$$\nabla(\rho\vec{\mathrm{v}}\,\vec{\mathrm{v}}) = -\nabla p + \nabla\left(\overline{\overline{\tau}}\right) + \vec{\rho \mathrm{g}} \tag{2}$$

Additionally, the equation governing energy conservation is:

$$\nabla[\vec{\mathrm{v}}(\rho E + p)] = \nabla\left[k_{\mathrm{eff}}\nabla T - \sum_j h_j \overrightarrow{\mathrm{J}_j} + \left(\overline{\overline{\tau}}_{\mathrm{eff}}\vec{\mathrm{v}}\right)\right] \tag{3}$$

where the gravitational body force is represented by the vector $\rho\vec{\mathrm{g}}$, the stress tensor is represented by τ, ρ – the fluid density, E – the total energy, p – the pressure, and k_{eff} – the effective conductivity.

Due to the turbulent nature of the flow, the k-ε model [26] has been chosen. Furthermore, the no-slip situation has been enforced on all walls. The transport equations below can be used to derive the kinetic energy of turbulence, k, as well as the associated rate of dissipation of turbulence, ε:

$$\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_{\mathrm{t}}}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho_\varepsilon \tag{4}$$

$$\frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_{\mathrm{t}}}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{5}$$

The turbulent viscosity is denoted by μ_{t}, while the model constants are represented by $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ, σ_k, and σ_ε. The following boundary conditions have been defined for the present simulation [16]:

– inlet: U_{in} = 0.02216 m/s (which corresponds to a rate of flow of 79.79 m^3/hm^2); and T_{in} = 300 K;
– absorber: T_{abs} = 380 K (which is equivalent to 800 W/m^2 of solar flux);
– insulation: T_{iso} = 340 K; and
– exit: $P_{\mathrm{exi}} = P_{\mathrm{atm}}$.

A mesh for the domain was generated using the ANSYS software. Generally, a fine mesh is utilized in regions where a more detailed analysis is required, such as baffles, while a coarse mesh is used in regions with minor changes in the domain geometry, fig. 3.

Table 1 displays the comparison between the results of our earlier study [16] and the experimental findings of Moummi [25]. The objective of this assessment was to gauge the precision of the numerical model [16] in predicting the output temperature for different flow rates. The analysis demonstrated that the simulated [16] and experimental [25] output temperatures differ by about 4 °C. However, the comparison revealed that the numerical [16] and experimental [25] models have a satisfactory level of agreement. This indicates that the previous numerical model [16], which is also employed in this study, can be deemed reliable.

Table 1. Model validation through comparison with experimental data

Flow [m^3/h^{-1}m^{-2}]	79.79	74.01	64.76	42.79	23.71
CFD output temperature [16] [°C]	53	54	59	65	69
Exp. output temperature [25] [°C]	50	51	56	63	68

Findings and discussions

In fig. 4, the temperature distribution within the SC is depicted. It is noteworthy that the heat transfer within the collector exhibits significant values from the initial baffle all the way to the outlet. As a result, the temperature of the air in the vicinity of the surface is noted to be greater than the temperature of the air located farther away from the surface, up to a height equivalent to one third of the duct. This finding indicates that the SC is highly efficient at transferring heat, especially in the region adjacent to the surface. However, as we move away from the surface, the temperature decreases gradually, highlighting the importance of proper air circulation within the collector to ensure optimal heat transfer.

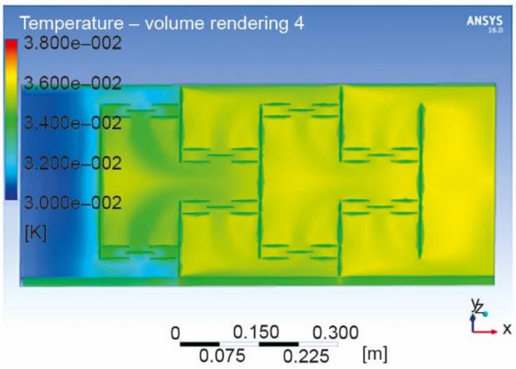

Figure 4. Temperature distribution within the collector

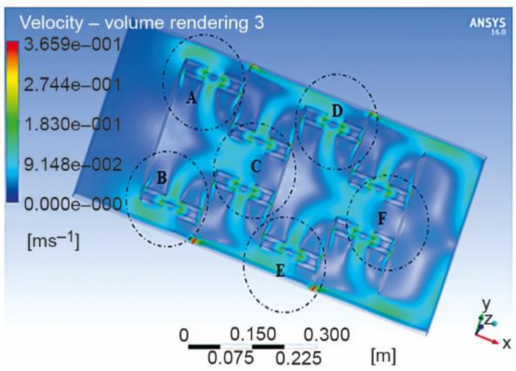

Figure 5. Velocity distribution within the collector

In fig. 5, the speed distribution within the SC is presented. In order to enhance the velocity of the air-flow inside the collector, the air-flow was partitioned into two separate paths. Longitudinal baffles were installed to lengthen the path of the air-flow, resulting in an increase in dead zones. To address this issue, transverse baffles were added to the design, which not only eliminated dead zones but also increased turbulence. In addition, small spaces were created between the longitudinal baffles and the lateral sensor walls, as well as between the transverse baffles and the longitudinal baffle, to ensure the elimination of dead zones. The efficiency of the SC has been enhanced as a result of these changes, which have resulted in an increase in the speed and uniformity of air-flow while simultaneously reducing the negative impact of inactive regions.

Figures 6(a)-6(f) displays the configuration of the current lines surrounding different small transverse obstacles. The illustrations demonstrate that the current lines tend to cluster over these obstacles, thereby expediting the air-flow. However, it is also observed that the presence of transverse baffles can promote the emergence of unwanted recirculation zones, which can have adverse effects on the overall behavior of the SC. Therefore, while the small transverse baffles have a positive impact on the flow velocity, it is crucial to carefully design the placement and size of the baffles to prevent the formation of recirculation zones that could hinder the efficiency of the SC.

By incorporating small transverse baffles, it is possible to augment the speed of flow. As illustrated in fig. 7(a), the *u* component, which flows along the primary direction, experiences a decrease in speed in regions distant from the baffles. Passing the air through

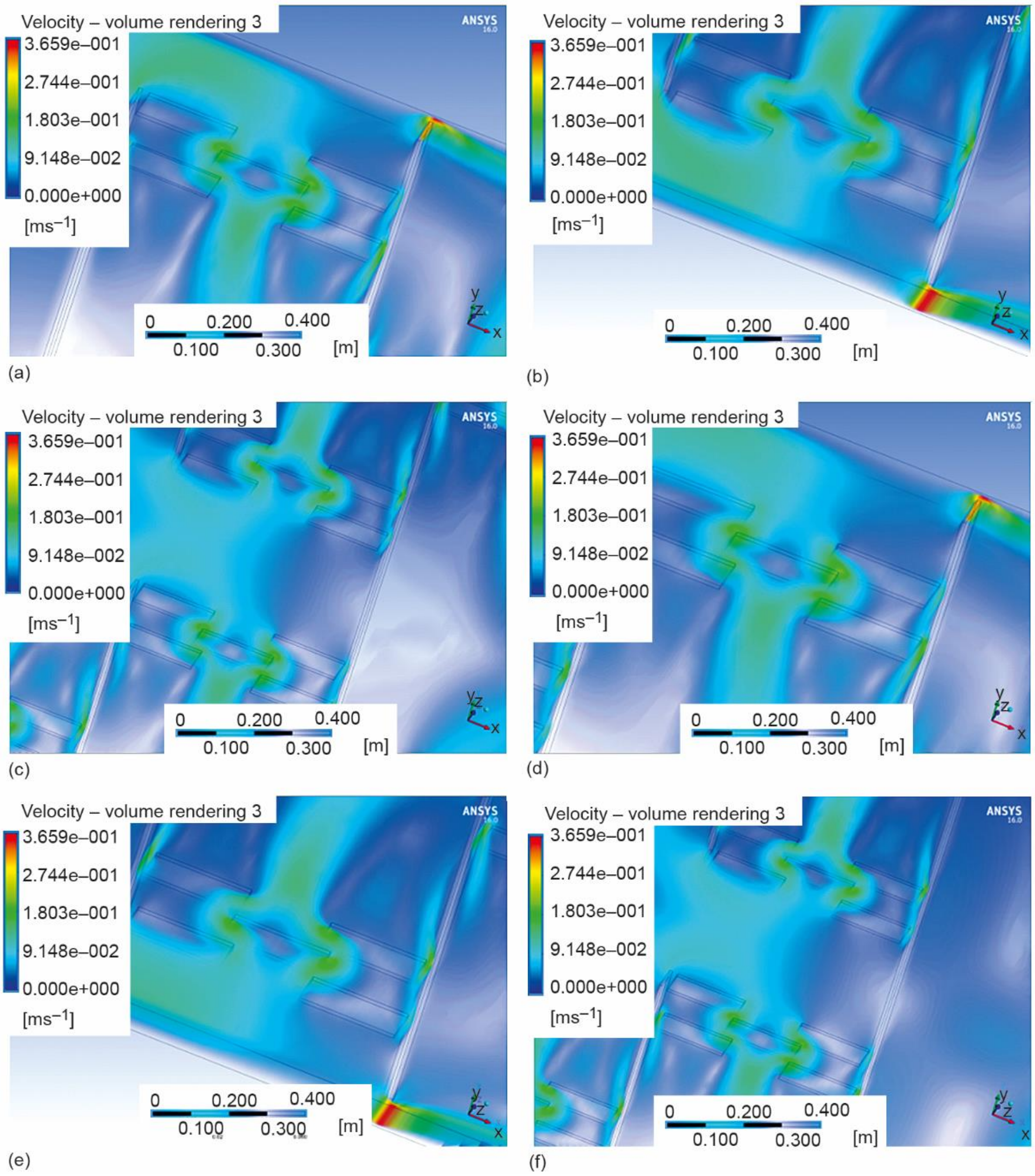

Figure 6. Velocity distribution in various stations; (a) station A, (b) station B, (c) station C, (d) station D, (e) station E, and (f) station F

these small baffles results in a velocity increase of more than ten times the entry speed. The direction of the velocity is positive when aligned with the flow, and negative when opposed to it. Figure 7(b) highlights the significance of the *v* component around the small transverse baffles. The *w* component, as shown in fig. 7(c), remains almost negligible, except in the vicinity of the baffles where it assumes critical values.

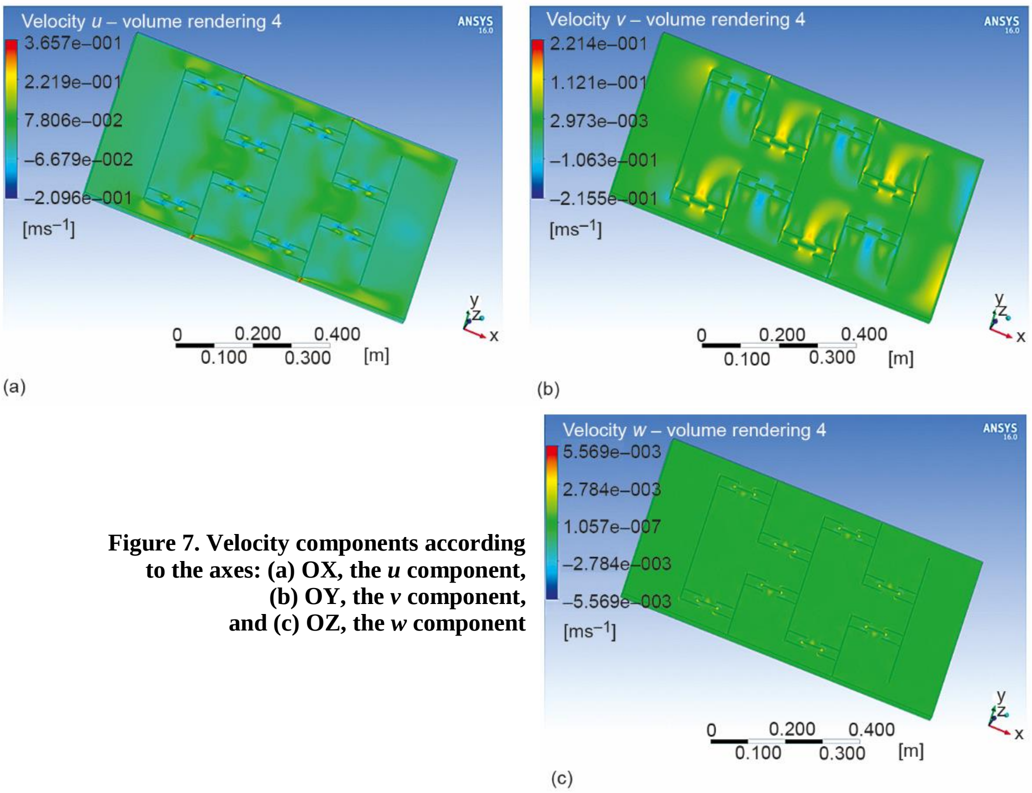

Figure 7. Velocity components according to the axes: (a) OX, the *u* component, (b) OY, the *v* component, and (c) OZ, the *w* component

In fig. 8, the assessment of the *k* energy is displayed, revealing that the fluid-flow possesses *k* throughout its trajectory. The figure highlights a significant surge in the *k* around the small transverse baffles. This finding indicates that the presence of small baffles leads to an intensification of the *k*, which could potentially impact the overall fluid-flow behavior. The observed distribution of *k* in the system emphasizes the need for careful consideration of the impact of turbulence on the system performance.

Figure 9 depicts the distribution rate of turbulence, represented by the symbol ε, in the system under consideration. The figure indicates that the turbulent eddy dissipation exhibits a sharp increase, reaching up to 6.12×10^{-2}, around the small transverse baffles. Additionally, it is noteworthy that the spaces between the longitudinal baffle and the side walls harbor significant values of turbulent eddy dissipation, as observed in the figure. The finding of high turbulent eddy dissipation in these regions emphasizes the potential impact of the system geometry on fluid-flow behavior and highlights the importance of designing the system with careful consideration of the potential turbulence effects.

Figure 10 illustrates the temperature distribution at the exit of the SC, showcasing significant improvements in thermal transfer efficiency in our recent model compared to our previous study [16]. The results reveal that the average temperature has risen impressively to reach 359 K, indicating a substantial increase in thermal energy output from the collector. Such improvements in thermal transfer efficiency have far-reaching implications for the overall performance of the SC system, enabling higher energy conversion efficiencies and more sustainable energy practices. The observed enhancements in thermal transfer efficiency high-light the effectiveness of our SC model and its potential to contribute to the development of more efficient and sustainable energy systems.

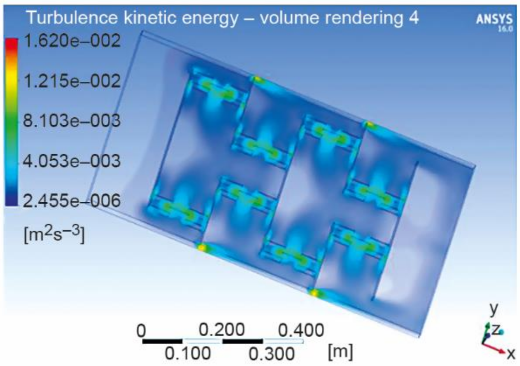

Figure 8. Distribution of *k* energy

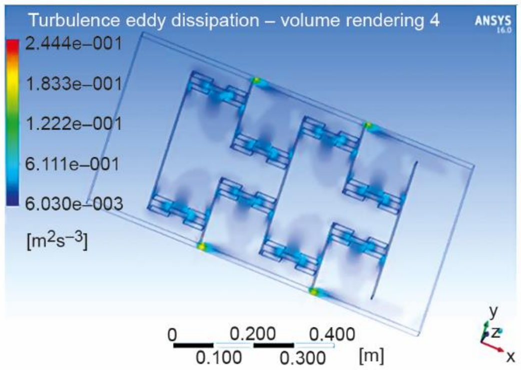

Figure 9. Distribution of *ε* rate

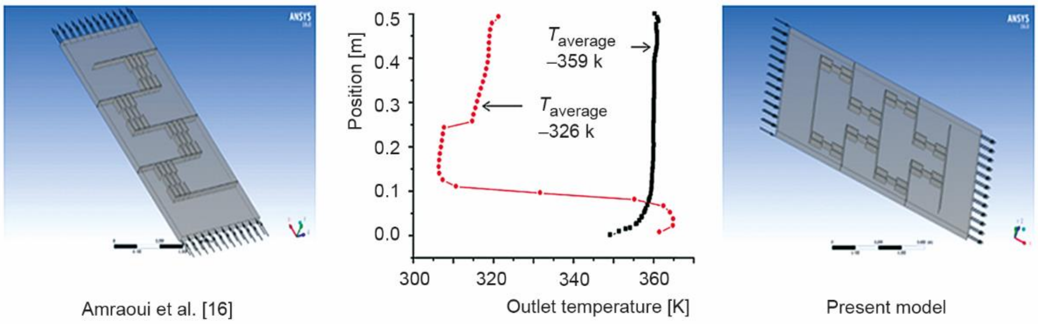

Figure 10. Temperature profiles at the collector exit

Conclusions

Improving the heat transfer between the heat transfer fluid and absorber in flat plate SC can be achieved through various methods. One approach is to incorporate longitudinal and transverse baffles and create double paths for the flow. In this study, such a SC was developed and its dynamic and thermal behavior was evaluated. The results indicate that the proposed model exhibited a significant improvement in the hydrothermal structure of the collector.

Future studies in this domain could explore further modifications to the SC design, such as the use of alternative materials for the absorber and fins, as well as exploring different methods of creating multiple flow paths. Additionally, alternative heat transfer fluids could be considered to optimize SC performance and energy conversion efficiency. The development of more efficient SC has the potential to contribute significantly to sustainable energy practices, and ongoing research in this field is critical for continued progress.

Acknowledgment

The authors Jihad Asad, Rabab Jarrar, and Hussein Shanak would like to thank Palestine Technical University-Kadoorie for supporting this work financially.

References

[1] Rebhi, R., *et al.*, Forced-Convection Heat Transfer in Solar Collectors and Heat Exchangers: A Review, *Journal of Advanced Research in Applied Sciences and Engineering Technology*, *26* (2022), 3, pp. 1-15

[2] Menni, Y., *et al.*, A Review of Solar Energy Collectors: Models and Applications, *Journal of Applied and Computational Mechanics*, *4* (2018), 4, pp. 375-401

[3] Menni, Y., *et al.*, Advances of Nanofluids in Solar Collectors – A Review of Numerical Studies, *Math Model Eng Probl*, *6* (2019), 3, pp. 415-27

[4] Menni, Y., *et al.*, Nanofluid-flow in Complex Geometries – A Review, *Journal of Nanofluids*, *8* (2019), 5, pp. 893-916

[5] Menni, Y., *et al.*, Computational Thermal Analysis of Turbulent Forced-Convection Flow in an Air Channel with a Flat Rectangular Fin and Downstream V-Shaped Baffle, *Heat Transfer Research*, *50* (2019), 18, pp. 1781-1818

[6] Moummi, N., *et al.*, Energy Analysis of a Solar Air Collector with Rows of Fins, *Renewable energy*, *29* (2004), 13, pp. 2053-2064

[7] Bazri, S., *et al.*, A Review of Numerical Studies on Solar Collectors Integrated with Latent Heat Storage Systems Employing Fins or Nanoparticles, *Renewable Energy*, *118* (2018), Apr., pp. 761-778

[8] Panchal, H., *et al.*, Performance Evaluation of Using Evacuated Tubes Solar Collector, Perforated Fins, and Pebbles in a Solar Still-Experimental Study And Co_2 Mitigation Analysis, *Environmental Science and Pollution Research*, *30* (2023) 5, pp. 11769-11784

[9] Menni, Y., *et al.*, Baffle Orientation and Geometry Effects on Turbulent Heat Transfer of a Constant Property Incompressible Fluid-flow inside a Rectangular Channel, *International Journal of Numerical Methods for Heat & Fluid-flow*, *30* (2020) 6, pp. 3027-3052

[10] Menni, Y., *et al.*, Combination of Baffling Technique and High-Thermal Conductivity Fluids to Enhance the Overall Performances of Solar Channels, *Engineering with Computers*, *38* (2022), Sept., pp. 607-628

[11] Menni, Y., *et al.*, Numerical Calculations of the Thermal-Aerodynamic Characteristics in a Solar Duct with Multiple V-Baffles, *Engineering Applications of Computational Fluid Mechanics*, *14* (2020), 1, 1173-1197

[12] Ameur, H., Menni, Y., Laminar Cooling of Shear Thinning Fluids in Horizontal and Baffled Tubes: Effect of Perforation in Baffles, *Thermal Science and Engineering Progress*, *14* (2019), Dec., 100430

[13] Ameur, H., *et al.*, Numerical Investigation of The Performance of Perforated Baffles in a Plate-Fin Heat Exchanger, *Thermal Science*, *25* (2021), 5B, pp. 3629-3641

[14] 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), Aug., 100534

[15] Ammar, M., *et al.*, Numerical Analysis of a Flat Plate Solar Collector with Baffles in The Air Duct, *International Journal of Scientific Research & Engineering Technology (IJSET)*, *9* (2019), 2, pp. 1-5

[16] Amraoui, M. A., *et al.*, Three-Dimensional Analysis of Air-flow in a Flat Plate Solar Collector, *Periodica polytechnica mechanical engineering*, *62* (2018), 2, pp. 126-135

[17] Amraoui, M. A., Three-Dimensional Numerical Simulation of a Flat Plate Solar Collector with Double Paths, *International Journal of Heat and Technology*, *39* (2021) 4, pp. 1087-1096.

[18] Amraoui, M. A., and Benosman, F., Numerical Modeling of a Flat Air Solar Collector Fitted with Obstacles, *E3S Web of Conferences, 321* (2021), Nov., 04016

[19] Amraoui, M. A., Numerical Study of an Air-flow in a Flat Plate Air Solar Collector with Circular Obstacles, *Artificial Intelligence and Renewables Towards an Energy Transition, 4* (2021), Dec., pp. 839-846

[20] Abdi, G., *et al.*, 3-D Evaluation of a Thermal and Hydraulic Winged Solar Collector, *Instrumentation, Mesures, Métrologies*, *21* (2022), 2, pp. 35-41

[21] Menni, Y., *et al.*, Enhancement of Convective Heat Transfer in Smooth Air Channels With Wall-Mounted Obstacles in the Flow Path: A Review, *Journal of Thermal Analysis and Calorimetry*, *135* (2019), Apr., pp. 1951-1976

[22] Ulkir, O., *et al.*, Production of Piezoelectric Cantilever Using MEMS-Based Layered Manufacturing Technology, *Optik*, *273* (2023), Feb., 170472

[23] Ulkir, O., *et al.*, Fabrication and Experimental Study of Micro-gripper with Electrothermal Actuation by Stereolithography Method, *Journal of Materials Engineering and Performance*, *31* (2022), Apr., pp. 8148-8159

[24] Vural, E., Ozer, S., Thermal Analysis of a Piston Coated with SiC and $MgOZrO_2$ Thermal Barrier Materials, *International Journal of Scientific and Technological Research*, *1* (2015), 7, pp. 43-51
[25] Moummi, A., *Etude globale et locale du rôle de la géométrie dans l'optimisation des capteurs solaires plans à air* , Ph. D. thesis, Global and Local study of the role of geometry in the optimization of flat air solar collectors (in French language), Polytechnic University Hauts-de-France, Valenciennes, France, (1994), (Doctoral dissertation, Valenciennes)
[26] Launder, B. E., Spalding, D. B., The Numerical Computation of Turbulent Flow, *Computer Methods in Applied Mechanics and Engineering*, *3* (1974), 2, pp. 269-289

Paper submitted: March 30, 2023
Paper revised: April 12, 2023
Paper accepted: April 25, 2023