African Journal of Advanced Pure and Applied Sciences (AJAPAS)

Online ISSN: 2957-644X

Volume 3, Issue 3, 2024, Page No: 555-563

Website: *https://aaasjournals.com/index.php/ajapas/index*

ISI 2022-2023: 0.557 SJIFactor 2023: 5.689 معامل التأثير العربي 2023: (1.55)

Special issue: First Libyan Conference on Technology and Innovation (LCTI-2024), Benghazi, Libya

Improvement of the Thermal Performance of Air Collector by Using Inclined Baffles: A Three-Dimensional Analysis

Khaled Saad Alareby [1]*, Saleh Etaig [2], Nabil Elsharif [2]

[1] Department of Renewable Energy Engineering, Faculty of Engineering, Libyan Academic, Benghazi, Libya.

[2] Mechanical Engineering Department, University of Benghazi, Benghazi, Libya

تحسين الأداء الحراري لمجمع الهواء باستخدام الحواجز المائلة : تحليل ثلاثي الأبعاد

خالد سعد العريبي[1]*، صالح اعتيق[2]، نبيل الشريف[2]

[1] قسم تقنيات الطاقة، مدرسة الهندسة، الاكاديمية الليبية، بنغازي، ليبيا

[2] قسم الهندسة الميكانيكية، جامعة بنغازي، بنغازي، ليبيا

*Corresponding author: ksaldon77@gmail.com

Received: March 01, 2024	Accepted: May 05, 2024	Published: May 30, 2024

Abstract

In response to the pressing global concerns surrounding energy consumption, particularly in the context of escalating global warming and climate change. This research is dedicated to enhancing the performance of solar air collectors. Employing advanced computational fluid dynamics (CFD) modeling, the study focuses on optimizing the design efficiency by scrutinizing the impact of varying inlet air velocities on crucial thermal parameters. The investigation utilizes ANSYS Fluent to simulate a 3-D collector, providing a comprehensive understanding of its physical dynamics and accurate solutions. Through a thorough analysis of different operating conditions, including overall thermal efficiency and total air pressure drop, the research aims to map the nuanced performance of the solar air collector. The outcomes highlight the 90° baffle angle as a standout performer, showcasing thermal efficiency exceeding 95%. However, this configuration is accompanied by the highest pressure drop among the considered designs. In contrast, the 60° baffle angle emerges as a well-balanced option, offering optimal overall pressure drop and an acceptable range of thermal efficiency. The interplay of these variables is further explored through detailed simulations, providing a holistic view of the collector's behavior under various scenarios. This research not only contributes to the theoretical understanding of solar air collectors but also addresses practical considerations in design optimization. The findings underscore the importance of balancing thermal efficiency with operational considerations, guiding the selection of baffle angles for enhanced performance. As a result, this study not only advances the knowledge base in sustainable energy technologies but also provides actionable insights for the design and deployment of efficient solar air collectors.

Keywords: CFD, Solar Collector, Thermal Efficiency, Pressure Drop, Heat Transfer

الملخص

استجابةً للمخاوف العالمية الملحة بشأن استهلاك الطاقة، لا سيما في سياق تصاعد ظاهرة الاحتباس الحراري وتغير المناخ العالمي، يخصص هذا البحث لتحسين أداء سخانات الهواء الشمسية. باستخدام نمذجة ديناميكا الموائع الحسابية (CFD) المتقدمة، تركز الدراسة على تحسين كفاءة التصميم من خلال تدقيق تأثير سرعات دخول الهواء المختلفة على المعلمات الحرارية الأساسية. يستخدم البحث برنامج ANSYS Fluent لمحاكاة سخان هواء ثلاثي الأبعاد، مما يوفر فهمًا شاملاً لديناميكياته الفيزيائية وحلولاً دقيقة. من خلال تحليل شامل لظروف التشغيل المختلفة، بما في ذلك الكفاءة الحرارية الكلية وفقدان ضغط الهواء الكلي، يهدف البحث إلى رسم خريطة لأداء سخان الهواء الشمسي الدقيق. تسلط النتائج الضوء على زاوية الحاجز 90 درجة باعتبارها الأفضل أداءً، حيث تظهر كفاءة حرارية تتجاوز 95٪. ومع ذلك، يصاحب هذا التكوين أعلى انخفاض للضغط بين التصميمات المدروسة. على النقيض من ذلك، تبرز زاوية الحاجز 60 درجة كخيار متوازن، حيث توفر انخفاضًا مثاليًا في الضغط الكلي ونطاقًا مقبولاً للكفاءة الحرارية. يتم استكشاف التفاعل بين هذه المتغيرات بشكل أكبر من خلال عمليات محاكاة تفصيلية، مما يوفر رؤية كلية لسلوك السخان في سيناريوهات مختلفة. لا يساهم هذا البحث فقط في الفهم النظري لسخانات الهواء الشمسية، ولكنه يعالج أيضًا الاعتبارات العملية في تحسين التصميم. تؤكد النتائج على أهمية موازنة الكفاءة الحرارية مع الاعتبارات التشغيلية، مما يوجه اختيار زوايا الحاجز لتحسين الأداء. نتيجة لذلك، لا تقدم هذه الدراسة فقط قاعدة معرفية متقدمة في تقنيات الطاقة المستدامة ولكنها توفر أيضًا رؤى عملية لتصميم ونشر سخانات هواء شمسية فعالة.

الكلمات المفتاحية: CFD، المجمع الشمسي، الكفاءة الحرارية، انخفاض الضغط، انتقال الحرارة.

1- Introduction

The simplicity and efficiency with which solar energy can be harnessed make it an increasingly attractive prospect for a myriad of applications. The pivotal aspect lies in the utilization of solar collectors that seamlessly convert sunlight into thermal energy. This conversion process, devoid of any harmful byproducts or adverse effects, opens up a world of possibilities for leveraging solar power in a sustainable and responsible manner.

Beyond the immediate benefits of reducing environmental degradation, the widespread adoption of solar energy can usher in a transformative era marked by energy independence and resilience. The journey towards a future powered by solar energy involves not just technological advancements but also a collective commitment to embracing cleaner alternatives for a healthier planet.

As we continue to explore and refine the potential of solar energy, a sustainable and harmonious coexistence between our energy needs and environmental well-being beckons. The path to a solar-powered future is illuminated not just by the radiant rays of the sun but also by the collective efforts of individuals, communities, and industries dedicated to fostering a greener and more sustainable world [1].

Several studies have been conducted to investigate the use of inclined baffles in improving the thermal performance of solar air collectors. In particular, a three-dimensional analysis using computational fluid dynamics (CFD) simulations has been used to provide an accurate and comprehensive understanding of the flow and temperature distribution inside the collector. For example, the use of inclined baffles could increase the heat transfer rate exceeding 17% compared to a collector without baffles [2]. Jurinak and Abdel-Khalik's study [20] provides valuable insights into the optimization of phase-change energy storage materials for solar air-heating systems. Their research aimed to determine the optimum physical properties of phase-change energy storage materials (PCES) and assess the system performance over an entire heating season for different space heating loads.

The incorporation of V-shaped ribs represents a noteworthy advancement in optimizing heat transfer distribution across the span of Solar Air Collectors (SAC). This innovative modification serves to enhance the convective heat transfer process by fostering the development of additional flow regions characterized by rotating secondary flow. The V-shaped ribs act as disruptors within the airflow, inducing a controlled turbulence that, in turn, facilitates a more efficient distribution of heat along the collector's surface [6].

The expanded surface area facilitates increased contact between the air and the absorber, leading to longer interaction times. The prolonged contact duration, coupled with the enhanced heat transfer area, contributes to an overall improvement in the heat transfer rate within the collector. The corrugated surface design, therefore, emerges as a sophisticated solution that balances the trade-off between increased pressure drop and the augmented heat transfer performance in solar air collectors [7].

The combined effect of increased turbulence and reduced thermal boundary layer contributes to improved thermal and hydraulic performance within the system. This dual enhancement is crucial for optimizing the overall efficiency of the packed bed solar thermal system, making it a promising configuration for harnessing solar energy effectively [8].

In the quest for optimizing the performance of solar air heaters, researchers have explored various techniques and configurations. Dutta and Dutta [9] delved into the examination of eight distinctive types of baffles in rectangular channels. Their study emphasized the strategic placement of baffles in areas with high heat flux to maximize heat transfer effectiveness. This underscores the importance of thoughtful baffle positioning as a key factor in enhancing overall thermal performance.

Han et al. [10], on the other hand, focused on investigating the impact of rib shape, angle of attack, and the pitch-to-rib height ratio (P/e) on friction factor and heat transfer characteristics in a rectangular duct with two side roughened walls. Their findings highlighted that the optimum heat transfer and friction factor values were achieved at a relative roughness pitch of 10, with the ribs oriented at a 45-degree angle. This information provides valuable insights into the design considerations for ribbed surfaces, offering a basis for improving the efficiency of solar air heaters.

Therefore, the use of inclined baffles has emerged as an effective technique for improving the thermal performance of solar air collectors. This research aims to provide a comprehensive understanding of the improvement of the thermal performance of air collectors using inclined baffles, based on numerical simulation. The three-dimensional analysis using CFD simulations tools approved to provide a valuable result in optimizing the design and performance of the collector.

2- Model Description

A comprehensive numerical model based on Computational Fluid Dynamics (CFD) has been meticulously developed to analyze and simulate the performance of a flat plate solar collector. Computational Fluid Dynamics, commonly referred to as CFD, constitutes a powerful tool for studying complex systems involving fluid flow, heat transfer, and associated phenomena through computer-based simulations.

The physical model of the flat plate solar collector serves as the foundation for constructing the CFD model, allowing for a detailed examination of the collector's behavior under various operating conditions. This CFD model is designed to capture the intricate interplay of fluid dynamics and heat transfer within the collector, providing valuable insights into its thermal performance.

2.1. Geometry creation

The solar air heater geometry is designed as a straightforward rectangular channel, featuring key dimensions of a length of 1000 mm, a width of 500 mm, and an air flow duct height of 50 mm, as illustrated in Figure 1. The collector plate, made of steel with a thickness of 2 mm, was selected for its thermal properties. To enhance heat transfer and investigate different flow patterns, inclined baffles were strategically positioned. These baffles are located sequentially on the bottom plate, each with a thickness of 5 mm, and are equally distributed in 10 parallel passes, forming a rectangular shape.

The study explores various inclinations of the baffles, namely 30°, 45°, 60°, and 90°, to assess their impact on the overall performance of the solar air heater. Both long and short configurations of the baffles were investigated, with the lengths differing by 100 mm. This comparative analysis aims to discern the influence of baffle length on heat transfer and fluid dynamics within the solar air heater. Overall, the geometric parameters and material selection contribute to a structured investigation into the thermal efficiency and effectiveness of the solar air heater under diverse baffle configurations.

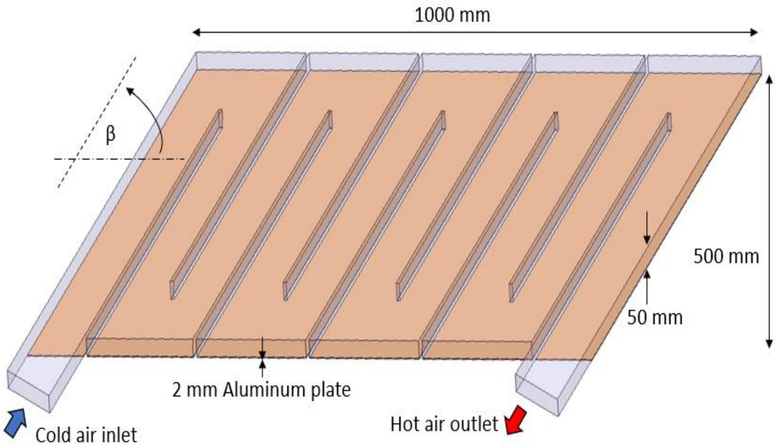


Figure 1. Schematic view of the air duct with the plate.

2.2. CFD model

The Computational Fluid Dynamics (CFD) model employed in this study is derived by solving the Navier-Stokes equations and the energy equation using the finite volume method. The simulations were conducted using the commercial software package ANSYS FLUENT 18.1. Several assumptions were made to facilitate the computational analysis, including:

- 3D Fluid Flow and Heat Transfer Simulation.
- Steady-State Conditions.
- Constant Thermo-physical Properties.
- No-Slip Condition at the Wall.

2.2.1. Governing equation

In CFD methods there are numerical solutions of mass, energy conversion and momentum. The solution of these equations can be analyzed through the help of numerical methods and algorithms. The governing equations are summarized as follows [3]:

Continuity equation:

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

Momentum equation:

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

Energy equation:

$$\frac{\partial}{\partial x_j}[u_i(\rho E + P)] = \frac{\partial}{\partial x_j}\left[\left(k_f + \frac{C_p \mu_t}{Pr_t}\right)\frac{\partial T}{\partial x_j}\right] \quad (3)$$

where

$$\tau_{ij} = \mu_{eff}\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right] \quad (4)$$

and

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (5)$$

where u_j, u_i are representing the velocity vectors, ρ is the density of the used fluids, E identifies the total energy, C_p and k_f are the specific heat and the effective thermal conductivity, respectively. δ_{ij} is the identity matrix, μ_t is the turbulent viscosity, Pr_t is the turbulent Prandtl number, k is the turbulent kinetic energy, and ε is the dissipation rate. μ_{eff} is the effective molecular dynamic viscosity, τ_{ij} describes the stress tensor [4].

The K-Epsilon viscous model is used for turbulent flow only, therefore, it is necessary to calculate the Reynolds number to account for any turbulent flow present within the duct. The flow is laminar when Re < 2300, transient when 2300 < Re < 4000 and turbulent Re > 4000 [5]. The Reynolds number can be calculated as follows:

$$Re = \frac{\rho v D_h}{\mu} \quad (6)$$

where Dh is representing the Hydraulic diameter, ρ is the air density, v is the mean velocity. The average Nusselt number for solar air collector is defined as:

$$Nu = hD_h/k_f \quad (7)$$

where h is the average convective heat transfer coefficient. The friction factor calculated by using the pressure drop at the test section duct, defined as:

$$f = \frac{(\Delta P/l)D}{2\rho v^2} \quad (8)$$

The hydraulic diameter for the domain is:

$$D_h = 4A/P \quad (9)$$

where A is representing the area of the duct (500mm X 50mm), P is the Perimeter of the air duct (2*(500mm+50mm)). The useful heat transfer to the air stream can be written in the form of the inlet and the outlet temperatures as followed:

$$Q = \frac{\dot{m}C_p\,(T_o - T_i)}{WL} \quad (10)$$

The efficiency of the solar air collector can be described as follow:

$$\eta = \frac{\dot{m}C_p\,(T_o - T_i)}{WL\,.I} \quad (11)$$

2.3. The mesh

Creating an accurate computational grid, also known as meshing, is a critical step when utilizing Computational Fluid Dynamics (CFD) tools, as it significantly influences the precision of the solution. This technique is not only pertinent to CFD but is also integral in software-based simulations such as Finite Element Analysis (FEA). The computational grid divides the simulation domain into discrete elements, and the size and shape of these elements play a crucial role in determining the accuracy of the simulation results. Each element represents a discrete solution for the differential equations applied within the domain.

In this study, the commercial mesh generator ANSYS (version 18.1) was employed for creating the geometry of the solar air heater (SAH) and generating the computational domain meshes. The mesh, or grid, for the baffled cases is depicted in Figure 2. This grid is a representation of the discretized computational domain, where each element corresponds to a portion of the domain, facilitating the numerical solution of the governing equations.

The quality of the mesh is vital for obtaining reliable results, and ANSYS provides advanced capabilities for generating structured or unstructured meshes based on the geometry and simulation requirements. The visual representation of the mesh in Figure 2 offers insights into the density and distribution of grid points, showcasing the meticulous process of mesh generation carried out in preparation for the subsequent CFD simulations.

Figure 2. All domain mesh mesh.

2.4. Boundary conditions

Once the geometry and mesh of the studied physical domain are established, the next step involves specifying the geometrical zones on which boundary conditions will be applied.

The flow within the channel is confined by two horizontal walls – the upper "glass" and the lower "absorber." The absorber wall is subjected to a specified heat flux and is thermally isolated from the bottom. The interior domain is categorized as "fluid." The flow is modeled using the realizable k-ε turbulent model, with boundary conditions set as follows:

- Air Flow Inlet Velocity: 3.983 m/s
- Temperature at the Inlet of the Flat Air Solar Collector: 300 K
- Initial Temperature of the Absorber: 300 K
- Outlet Pressure: Zero Gage Pressure

These boundary conditions are instrumental in defining the characteristics of the flow and heat transfer within the solar air heater.

3. Results And Discussion

The numerical results of the flat plate solar collector that employs air as its working medium. The primary benchmarks for assessing the investigated configuration of baffle angles were derived from the collector's outlet air temperature, pressure drop, and overall efficiency, encompassing various thermodynamic parameters. Additionally, the impact of the baffle's length was assessed.

3.1. The effects of the baffles on the outlet temperature

As described in Figure 3, the reduction in baffle length led to a decrease in the outlet temperature, ranging from 8% at the maximum velocity to 2.15% when the air velocity was increased to the maximum, as compared to using the long baffle structure. This percentage holds true for the 60° angle, while no substantial difference is observed at 45°. The rationale behind this can be attributed to the shorter air passage through the collector, from the inlet to the exit, resulting in fewer significant interactions between the air and the plate at the corners. This, in turn, diminishes the surface area available for heat transfer.

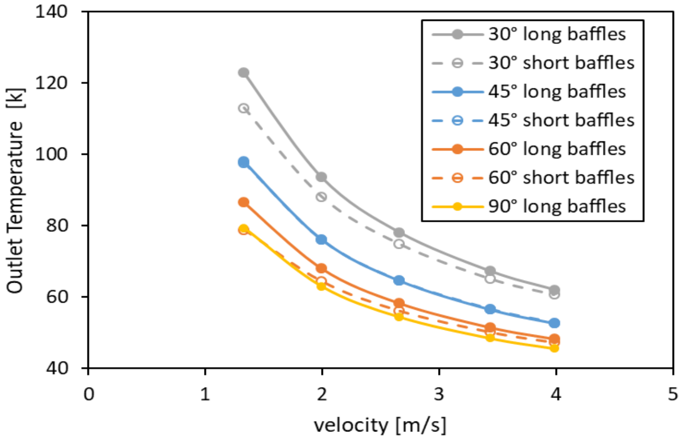
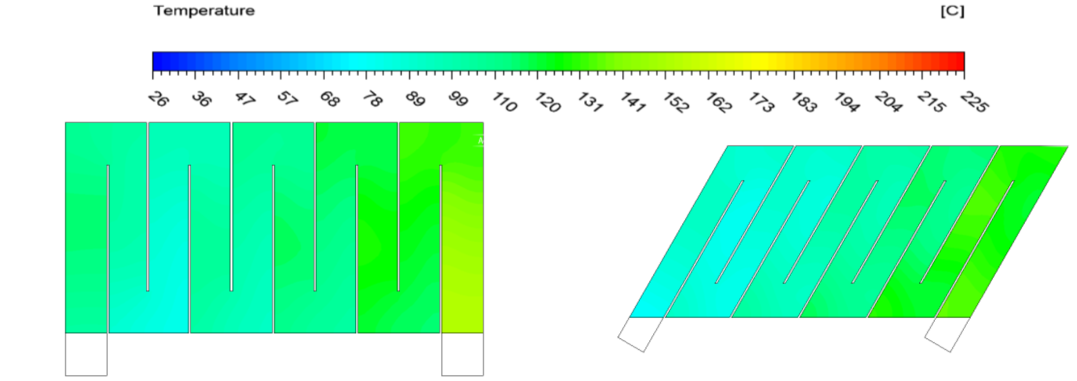
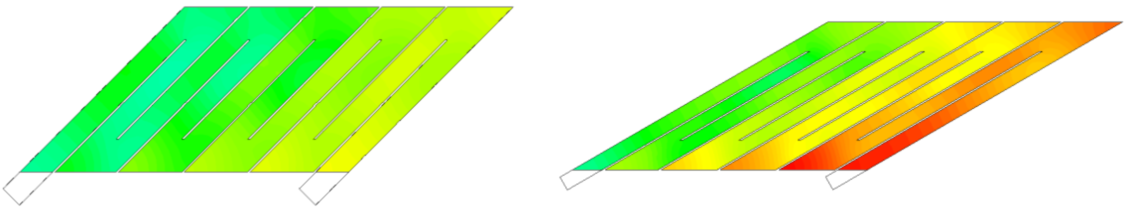


Figure 3. Air outlet temperature at different inlet velocities for long and short baffles.

Four long baffles inclined with different angles (90°, 60°, 45°, and 30°) are examined at various inlet air velocities. Figure 4 shows the temperature contour of the absorber plate at different angles illustrated at 1.327m/s. It can be observed that the highest inclined angles of the baffles (30°) help to provide the maximum plate temperature range, which increases toward the outlet. This can be justified by the lower mass flow rate passing through the collector and absorbing the available heat flux. The maximum plate temperature was recorded at 225°C for the 30° baffle compared with 175°C in the case of the baffle with 45° and the minimum temperature of 122°C when the 60° baffle is utilized.



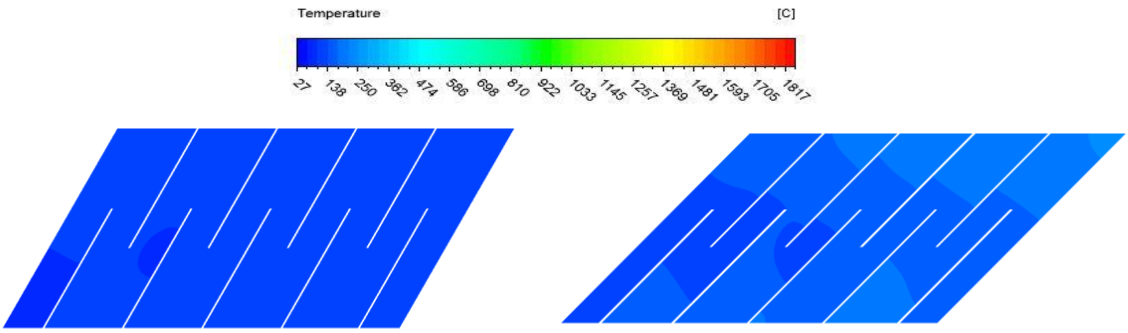
90° baffles at v = 1.327m/s   60° baffles at v = 1.327m/s



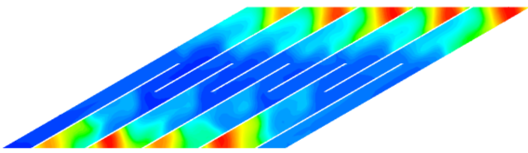
45° baffles at v = 1.327m/s   30° baffles at v = 1.327m/s

Figure 4 Temperature contour of the absorber plate via different long baffle angles at 1.327m/s

Three different angles of the short baffles, 60°, 45°, and 30°, were compared at the same velocity tested for the long baffles. Figure 5 shows the temperature contour of the plate exposed at the same range of the inlet air velocity. It can be seen that the short baffles disagreed with the temperature distribution shown for the long one, which provided higher temperature spots at the baffle corners. This is because of the short passage that the air followed, and less contact for the air with the plate corner reached. Therefore, the corners could not be cooled compared with the long structure tested in the previous section. The result still nominates the angle with a higher inclination for the higher plate temperature; for example, the 30-degree baffle offers a higher plate temperature compared with the others.



60° baffles at v = 1.327m/s   45° baffles at v = 1.327m/s



30° baffles at v = 1.327m/s

Figure 5 Temperature contour of the absorber plate via different short baffle angles at 1.327m/s

3.2. The effects of the pressure drop

Figure 6 shows the Pressure drop through SAH with baffles inclined at angle (90°, 60°, 45°, and 30°) at different velocities. It can be seen that the long baffles Pressure drop increase with increase the velocity and the baffles inclined at angle 45° shows the minimum Pressure drop among all cases, while the short baffles provided the highest pressure drop compared with the long once.

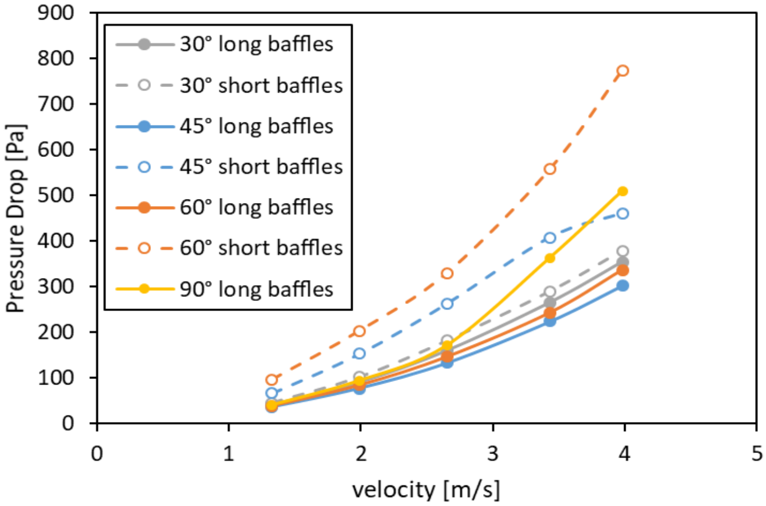
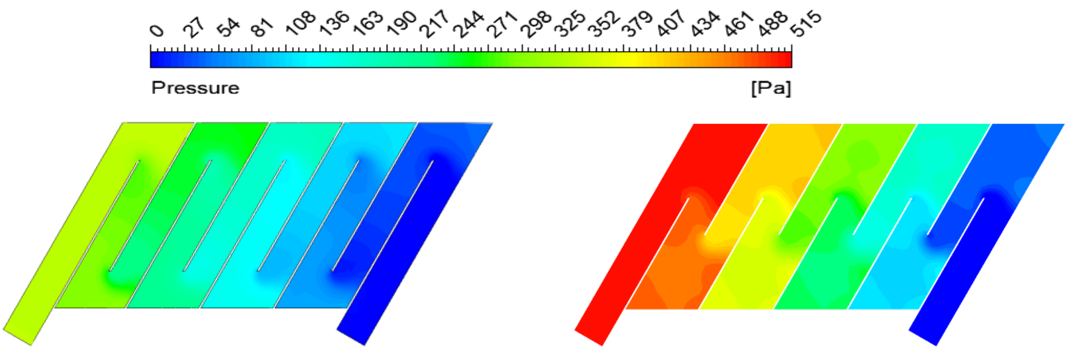


Figure 6 Pressure drop at different inlet velocities for long and short baffles.

In the realm of mechanical equipment dealing with fluids, the alteration in pressure assumes a pivotal role, representing the pressure drop in the context of energy loss as defined by the Bernoulli equation. Figures 7 illustrate the pressure contours of the air within the collector at 3.983m/s for 60° collector. The results were obtained with a fixed outlet pressure at atmospheric value, and the inlet pressure was recorded for each instance. Observing the figures, it becomes evident that the short baffles provided higher pressure drop compared with the long one



60° long baffles 60° short baffles

Figure 7 Pressure distribution at 3.983m/s for long and short baffles

3.3. The collector thermal efficiency

The primary parameter for evaluating the suitability of any collector is thermal efficiency. This term characterizes the maximum amount of heat that can be absorbed from the input heat to the collector, primarily the heat flux directed to the absorber plate. Figure 8 illustrates the variation in overall collector efficiency with inlet velocity for various inclined baffle angles. In term of the long baffles; The results show that overall collector efficiency increases with velocity due to the increase in mass flux passing through the collector as the velocity rises. In summary, the collector with a 90° baffle angle demonstrated the highest thermal efficiency, surpassing 95%, followed by the 60° baffles. In contrast, the lowest baffle angle of 30° could not achieve more than 90%, particularly at the maximum air velocity. Furthermore, the trend in the results suggests that the tested velocity falls within the optimal range for simulation, as an increase in velocity would yield only a marginal change that does not align with the design requirements of this collector.

For the short baffles, surprisingly, the 45° baffle angle provided the highest efficiency in the short structure, whereas the 30° angle in this configuration could not achieve more than 72.6% to 87% efficiency at the tested air velocities. Consequently, the short structure with these baffle angles is not recommended for such a design.

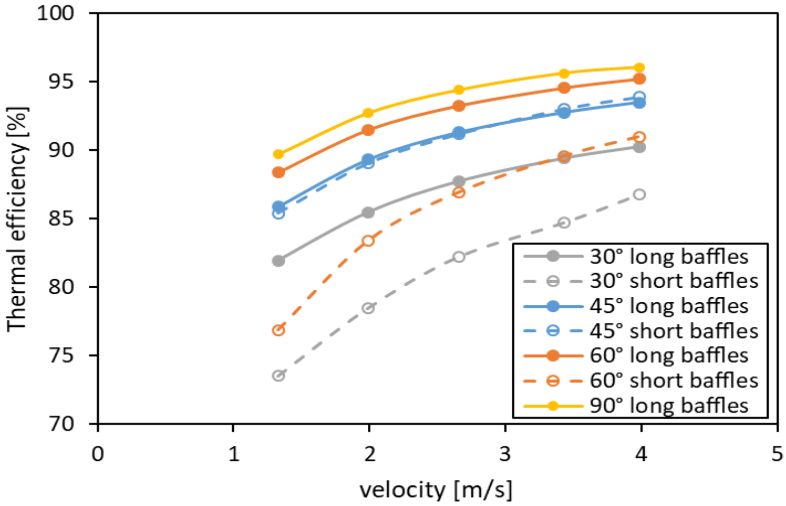


Figure 8 Thermal efficiency at different inlet velocities for long and short baffles.

4. Conclusions

This research investigates the assessment of various solar air collectors, focusing on the influence of structural geometry, particularly the angle of baffle inclination, the impact of inlet air velocity, and the effect of utilizing long and short baffles. Computational Fluid Mechanics was employed for 3D modelling to comprehensively analyse the collector's performance across varying operational conditions using ANSYS Fluent. The primary conclusions drawn from the research are as follows:

1. Baffles inclined at a 30° angle exhibited the highest outlet temperature among the considered angles. This outcome is attributed to the reduction in air mass flux flowing through the collector.
2. The thermal efficiency analysis revealed that the 90° baffles achieved the highest thermal efficiency, followed by the 60° baffles, both exceeding 90%.
3. Evaluation of pressure drop suggested that the 90° baffles were not recommended due to an exaggerated pressure difference. Consequently, these angles were excluded from the comparison with the short baffles.
4. Across various parameters, including output air temperature, plate temperature distribution, pressure drop, and thermal efficiency, all long baffle cases were deemed optimized when compared to short baffles. This conclusion was further supported by visualizations of parameter contours and the main distribution of air through the collector.

References

1. Maka A O M, Alabid J M, Solar energy technology and its roles in sustainable development, Clean Energy, Volume 6, Issue 3, June 2022, Pages 476–483, https://doi.org/10.1093/ce/zkac023
2. Zhao, L., Yang, X., Zhang, Y., Huang, L., & Ma, J. (2020). Improvement of thermal performance of solar air collectors by using inclined baffles: A three-dimensional analysis. Solar Energy, 204, 101-112. https://doi.org/10.1016/j.solener.2020.04.028
3. Qader, B. S., Supeni, E. E., Ariffin, M. K. A., & Talib, A. A. (2019). RSM approach for modeling and optimization of designing parameters for inclined fins of solar air heater. Renewable Energy, 136, 48-68.
4. CFD EXPERTS Simulate the Future', 2021.
5. Dunne, N. A., Liu, P., Elbarghthi, A. F., Yang, Y., Dvorak, V., & Wen, C. (2023). Performance evaluation of a solar photovoltaic-thermal (PV/T) air collector system. Energy Conversion and Management, 10(20), 100466.
6. Singh Patel S, Lanjewar A. Experimental and numerical investigation of solar air heater with novel V-rib geometry. J Energy Storage 2019;21:750–64. https://doi. org/10.1016/j.est.2019.01.016.
7. El Ferouali H, Zoukit A, Salhi I, Ei Kilali T, Doubabi S, Abdenouri N. Thermal efficiency and exergy enhancement of solar air heaters, comparative study and experimental investigation. J Renew Sustain Energy 2018;10(4):043709. https:// doi.org/10.1063/1.5039306

8. Zhu J, Jia H, Cheng X, Huang X, Liu X, Guo J. The design and performance evaluation of a high-efficient flexible solar air heater based on transparent spacer fabric composite. Sol Energy Mater Sol Cells 2019;201:110089. https://doi.org/ 10.1016/j.solmat.2019.110089.
9. Dutta, P.; Dutta, S. Effect of baffle size, perforation, and orientation on internal heat transfer enhancement. Int. J. Heat Mass Transf. 1998, 41, 3005–3013
10. Ozgen F, Esen M, Esen H. Experimental investigation of thermal performance of a double- low solar air heater having aluminium cans. Renewable Energy. 2009 Nov 1;34(11):2391-8