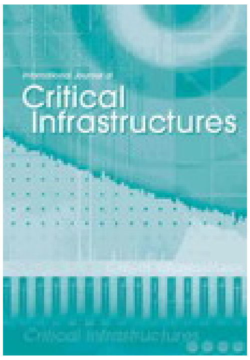


International Journal of Critical Infrastructures

ISSN online: 1741-8038 - ISSN print: 1475-3219
https://www.inderscience.com/ijcis

Investigation of the effect of inlet and outlet of the water flow on the productivity of a solar collector

Yi Cao

DOI: 10.1504/IJCIS.2028.10072417

Article History:

Received:	06 March 2025
Last revised:	06 May 2025
Accepted:	09 May 2025
Published online:	27 August 2025

Investigation of the effect of inlet and outlet of the water flow on the productivity of a solar collector

Yi Cao

Department of Equipment Engineering,
Shanxi Vocational University of Engineering Science and Technology,
Taiyuan City, Shanxi Province, China
Email: caoyime@163.com

Abstract: The paper makes it possible to harvest energy from solar radiation by using collectors containing fluid flow. Using this method, the radiant energy of the sun is absorbed by the fluid in the collector, and this energy is then conveyed to a thermal exchanger and used for a variety of purposes. Therefore, the influence of the inlet and outlet valve positions on the thermal productivity of the collector has been studied using computational fluid dynamics (CFD) in this study. The simulation of the problem was performed by COMSOL software by numerically solving the governing equations and establishing suitable boundary conditions. There are three different configurations investigated: input and output in the same direction, input and output perpendicular to each other, and input and output in the same direction and the opposite direction. Results indicate that the best thermal conduction rate occurs when the inlet and outlet are aligned (inlet and outlet are positioned opposite each other) and on the same side of the collector.

Keywords: solar collector; fluid flow; thermal performance; computational fluid dynamics; CFD; heat exchanger; boundary conditions; heat transfer; solar thermal systems.

Reference to this paper should be made as follows: Cao, Y. (2025) ‘Investigation of the effect of inlet and outlet of the water flow on the productivity of a solar collector’, *Int. J. Critical Infrastructures*, Vol. 21, No. 7, pp.1–25.

Biographical notes: Yi Cao graduated from Taiyuan University of Technology in 2006 with a Master’s degree in Municipal Engineering, and currently she is working at Shanxi Vocational University of Engineering Science and Technology. She has published three papers. Her research interest includes HVAC design.

1 Introduction

Solar energy is considered one of the most promising sources of renewable energy as it is extensively used to generate electricity, heat, and purify water. This has been propelled into the limelight as a solution considering the present-day chains of events under climate change and the depletion of fossil fuels, making solar energy one such sustainable, abundant, and clean alternative (Ameer and Jalil, 2024). Solar collectors are important apparatuses that absorb solar radiation and convert it into thermal energy for several applications, including heating water, desalination, industrial processes, and space heating (Anand et al., 2024). The performance of a solar collector is influenced by several factors, ranging from the design and material properties to the configuration of working fluid flow (Chandratreya, 2024). It is necessary to optimise the heat transfer mechanism in the collector to absorb maximum energy to enhance the performance of the entire system (Elaiyaraja and Boinapalli, 2024). Therefore, an important aspect to be taken into consideration concerning the performance of solar collectors is fluid flow dynamics (Femi et al., 2024). The best thermal mixing, the least heat loss by stagnant zones, and a more uniform temperature distribution across the collector may be achieved if these elements are placed scientifically (Uçkan and Yousif, 2021).

Solar collectors are designed differently for different applications, the most common modes being flat-plate collectors, evacuated tube collectors, and parabolic trough collectors (Gnanaguru et al., 2024). Flat-plate collectors are now being extensively used for domestic as well as industrial water heating because of their simple design and economical nature, while evacuated tube collectors hold the capacity to minimise heat losses for increased efficiency (Ha, 2024). Next, the parabolic trough collectors are employed in the solar thermal power plant but boast the new creation of concentrated solar energy upon a focal point to generate a higher temperature for efficient power generation (Ikwuagwu et al., 2024). Fluid dynamics and heat transfer mechanisms with any collector type are pivotal in ushering forth the overall efficiency (Kadam and Deming, 2024). Computational dynamics has established itself as an indispensable tool for the study of fluid motion and heat transfer phenomena occurring in solar collectors. The CFD simulation allows the researcher to examine the fluid flow behaviour under varying conditions, optimise the design parameters, and improve the performance of the system (Kumar et al., 2024). Finding the solution numerically to the governing equations of fluid dynamics and heat transfer allows one to find the configuration that enhances thermal conduction and maximal energy efficiency (Siddique et al., 2023). It also helps visualise the temperature distribution with velocity contours (Ajarostaghi et al., 2022).

However, environmental and economic factors were some of the most important reasons for deploying solar collector technology on a wider range (Madhuranthakam, 2024). Thus, while cost plays a major role in manufacturing, installing, and maintaining efficient systems, solar thermal technology will eventually lower operational costs than their fossil fuel counterparts (Manikandan et al., 2024). Nevertheless, the initial cost is usually high; governments and organisations worldwide encourage solar energy using subsidies and tax incentives for research grants, thus making such technology more accessible to residential consumers as well as industries (Mary et al., 2024). Solar collector technology forms one of the main tools addressing major environmental benefits, including zero greenhouse gas emissions, low fossil fuel dependency, and an overall sustainable energy generation source (Munshi, 2024; Suraj et al., 2024). It is, therefore, a more significant technology in globally reducing carbon emissions (Mahadewi, 2024). Solar collectors can be best optimised to achieve higher temperatures by streamlining complex interplays considering micro-mechanisms such as fluid flow dynamics, heat transfer processes, and their configuration in the whole system (Mehta et al., 2023). Some aspects that increase the efficiency of solar thermal technology include the positioning of inlet and outlet ports, the incorporation of energy storage solutions, the selection of advanced materials, and the inclusion of blended systems (Nomula et al., 2023; Victor Ikwuagwu et al., 2023). Researchers focus on sustainable and cost-effective solar collectors through the use of computational simulations coupled with experimental validations and innovative design approaches (Mathew et al., 2024; Zanardo, 2024). Advances in designs for solar collectors, CFD modelling, and experimental validations have resulted in a platform that will eventually provide even more exciting opportunities for experimentation (Zachar et al., 2003).

The fluid inlet and outlet configurations do not solely determine the thermal performance of a solar collector (Pandey et al., 2024). Still, it is also strongly influenced by the collector's design geometry, the thermal conductivity of its materials, surface emissivity, and operating conditions (Panyaram, 2024; Srilakshmi et al., 2024). These factors collectively determine the heat transfer efficiency and should be considered in predictive modelling to obtain a more accurate and generalised assessment of collector performance (Pierre et al., 2024; Singh, 2024a). The optimisation of solar collector design, particularly in terms of the inlet and outlet configurations, offers considerable advantages for both domestic and industrial heating systems (Raj et al., 2024; Tin et al., 2024). The findings from this study can be applied to improve the efficiency of solar thermal collectors, enhancing their overall thermal performance (Ramya et al., 2023; Singh, 2024b). In domestic heating systems, these optimisations can lead to the development of more efficient solar water heaters, ensuring better heat absorption and thermal retention (Saxena et al., 2023). This improvement can reduce energy consumption and lower operational costs for homeowners relying on solar thermal systems for hot water production (Shenbagavalli et al., 2024). For industrial heating systems, optimised collector designs could lead to more efficient large-scale solar collectors, which are often used in various processes such as water heating in factories, greenhouses, and food production (Senapati et al., 2023; Srinivasan and Arulvendhan, 2024). By reducing the pressure drop and improving the flow dynamics, these designs can lead to a more reliable and cost-effective use of solar energy in industrial applications, promoting sustainability and reducing dependence on non-renewable energy sources.

2 Literature review

Solar energy is now a major provider of sustainable and clean energy for replacing fossil fuels, whose use is expanding worldwide (Conceicao et al., 2022; Sikiru et al., 2022; Verduci et al., 2022). Growing concerns about environmental pollution and the emergence of oil shortage crises during the last few years have made many countries, especially the developed industrial ones, turn to sustainable energy providers. Due to this, the investigation of factors that help in the efficiency of solar collectors becomes quite essential, especially regarding the mechanics of water flow in solar collectors (Aggarwal et al., 2023; Alshukri et al., 2022; Shoeibi et al., 2022). The increase in the global demand for freshwater sources and the depletion of freshwater resources have intensified research into sustainable water purification technologies. Among the various methods, the high acceptability of solar-powered desalination and distillation systems arises mainly from their environmental sustainability and the ability to serve off-grid applications. The thermal performances of the different system components, especially solar stills and condensers, essentially govern the system's efficiency. Hoang et al. (2021) have carried out experimental work on a novel solar distillation system that uses inclined metal tubes as both collectors and condensers to speed up both evaporation and condensation processes. The high thermal conductivity of metal tubes improved heat absorption and transfer, subsequently accelerating the distillation process. The performance of the case study was evaluated concerning water yield, thermal efficiency, and energy conversion efficiency based on varying operational conditions. Outcomes derived from this study are useful in the development of low-cost and efficient technologies for solar distillation, which could be implemented for decentralised water purification systems in water-scarce regions.

Several investigations, including Alobaid et al. (2018) and Mehdipour et al. (2020), have emphasised that inlet and outlet configurations can seriously affect the general productivity of solar collectors. The flow rate, temperature, spatial distribution, and systems configuration interact in a very complicated way to significantly affect the collector efficiency in harnessing solar energy. The rush for fresh water in the world, especially in arid and remote areas, has necessitated a great deal of research on solar desalination technologies. Simple and relatively cost-effective solar stills are the most used among several methods available. However, the real challenge lies in their low productivity and, hence, the necessity of improvements in thermal efficiency and energy storage mechanisms. Dawood et al. (2020) carried out an experimental study on enhancing the productivity of solar stills by coupling them with parabolic trough collectors (PTCs) and incorporating phase change material (PCM) in the receiver's evacuated tubes as well as the still. With the aid of all said PTCs, considerably more heat can be fed into the system, while PCMs improve thermal storage, giving rise to extended water evaporation periods. The overall performance of the developed system was evaluated based on water yield, thermal efficiency, and energy utilisation at different operational conditions. The results presented here will lead to more advanced high-performance solar desalination devices by showing just how beneficial PTC and PCM integration would be. One can foresee this solution being an effective method for improving freshwater production under conditions in which water scarcity is caused by not having readily available conventional water sources. On the other hand, a vital role is played by how the water enters and leaves the collectors, which eventually decides their efficiency. It has also been shown that the geometrical changes at the inlets and outlets significantly influence the mechanism of thermal conduction, hence affecting the productivity of such systems (Gunjo et al., 2017; Zhang et al., 2016; Hussein, 2007).

Eltawil and Omara (2014) had an extraordinary idea about an enhanced solar still system that was incorporated with solar photovoltaic (PV) panels, a flat plate collector (FPC), and hot-air circulation for optimised thermal performance and distillate yield. The solar PV system supplied power for auxiliary heating and automation operations, and the FPC aided in heat absorption and transfer. Besides, the hot air contributed to speeding up evaporation and thus water production. The study performance evaluation was based on water yield, thermal efficiency, and energy utilisation. The results pave the way for the development of better-performing solar desalination systems since they show that the PV panel-FPC-hot air circulation coupling works. Such hybrid setups might prove to be a perfect solution for enhancing freshwater generation in areas with water scarcity and very little access to conventional water sources. The work by Khodadadi et al. (2015) details an empirical investigation of container thermal conduction in solar water heaters; they concentrate on a flat-plate solar collector. In this research effort, a mapping of the thermal spread inside the container is attempted. In this experiment, 29 temperature sensors are installed inside the container in a network. Besides these, two sensors are placed inside the inlet and outlet water passages, respectively.

The purpose of this experiment is to investigate and obtain insight regarding the positioning of inlet and outlet passages that affect the thermal spread inside the container. This investigation tries to find the optimal position of these passages that will yield a maximum temperature difference within the container. If this is done with a thorough analysis, this study will have an enormous effect on the efficacy and productivity of the solar water heater system. Assari et al. (2018) have done an integrated numerical and empirical investigation of the influence of changing inlet and outlet locations along a horizontal cylindrical container tank. The work has been focused on the optimisation of the thermal zoning process taking place in the tank. For carrying out the analysis, they kept one of the two positions constant, the inlet or the outlet, and changed the position of the other one. For this analysis, a model with an energy balance basis was developed. In each variation of this work, the empirical data were compared with the simulated temperature profile. Their results showed that the peak deviation of the simulated temperature profile from the empirically measured value was up to 7%. This probably provides the deviation of predicted thermal behaviour inside the tank from that observed temperature and thereby models prediction accuracy to some order of magnitude. These studies embark on various configurations and inlet conditions that influence thermal zoning within container tanks.

Kabeel et al. (2020) studied the effect of mass flow rate on the enhancement of freshwater in an inclined photovoltaic (PV) panel basin solar still. The inclined PV panel aims to generate electrical power as well as provide additional thermal energy to the major solar distillation process. The study was aimed at optimising water evaporation, condensation, and the overall distillate output by varying the mass flow rate under investigation. The performance of the system was analysed in terms of important parameters like water yield, thermal efficiency, and energy utilisation. Having mass flow rate optimisation proves its effect in this study in developing a more efficient solar desalination system. The research findings provide needed knowledge for improving hybrid solar still systems' operational performance, which holds good promise for freshwater production in water-scarce regions. A greater influence on the thermal zoning by changing the area near the plate was found. Alizadeh (1999) empirically and numerically investigated thermal zoning in a horizontal tank for thee different cases. In case I, the temperature was uniform throughout the tank and was higher than that of the cold water to be introduced at the inlet. In the second case, stratification was witnessed in a tank having an inlet cold water warmth equal to the bottom water warmth. The third case refers to the stratified tank whose inlet cold water temperature is below that of the bottom water temperature. Indeed, this resulted in a minor improvement in thermal zoning within the tank.

Kenjo et al. (2007) experimented by using a solar domestic hot water system with a mantle tank to investigate thermal zoning. They conducted three tests in which different inlets were used at higher, middle, and lower levels in the mantle. The first test, which had the inlet at a higher level, gave much better thermal zoning inside the tank. They also mentioned that there was a 6% deviation between the empirical and numerical values. All these, put together, highlight various factors that could affect thermal zoning inside the container tanks under different inlet conditions, configurations, and the placement of obstructions or devices inside the tanks. Such results bring out the role of the inlet conditions and the structural elements in achieving effective thermal zoning for energy efficiency in systems containing thermally stratified container tanks (Chandra and Matuska, 2019; Dzikevics et al., 2020; Li et al., 2021). Because the dynamics of water flow are so intimately connected with the productivity of solar collectors, this research aims to focus on that relationship with greater clarity. By borrowing from methodologies of past studies and adding in some innovative experimentation, this research hopes to provide some insight into optimal inlet and outlet configurations. Careful analysis of such dynamics is intended to help unlock pathways for a major stride forward in efficiency and viability regarding solar collectors as sustainable energy providers.

Shoeibi et al. (2023) performed an experimental study into the impact of PV/T waste heat on solar still water production and electricity generation using heat pipes and thermoelectric generators. The heat pipes allowed effective heat transfer from the PV/T system to the solar still, while the TEGs utilised temperature differences to create additional electricity production. Important performance parameters such as water yield, thermal efficiency, electrical output, and overall energy utilisation were assayed. Similarly, an environmental assessment was done for the proposed system in terms of sustainability benefits. The results, thus, advance hydra-efficient hybrid solar desalination systems through the successful application of waste heat from a PV/T system. This study can further improve the possibility of achieving higher yields in freshwater production with the simultaneous generation of renewable electric power, presenting a chance to meet water-energy scarcity in arid regions. The search for alternative sources of energy has redoubled efforts to explore solar technology, and it is here that solar collectors have become important instruments in the conversion of this limitless energy resource. However, even as strides continue to be made in solar research, one key task remains: the maximisation of the efficiency of these solar collectors. One such vital area under much examination concerns the complex interaction that exists between water flow dynamics and the efficiency of these collectors themselves.

Recent studies have highlighted that not only the design and material of the collector but also the internal flow dynamics – especially the position of the inlet and outlet – play a vital role in determining the thermal performance of solar collectors. Misaligned or inefficient configurations can lead to the formation of dead zones, reduced turbulence, and uneven heat absorption, which negatively affect the system's output. Therefore, analysing and optimising the positioning of these flow boundaries using CFD approaches offers an effective pathway to enhance the overall heat transfer efficiency. The present study deals with one relatively relevant yet often ignored aspect: the effective influence of water flow inlet and outlet configurations on the general productivity of solar collectors. Understanding how the path of water through these systems influences their performance may provide new opportunities for such systems to improve their efficiencies and, in turn, reinforce the viability of such systems as sustainable energy solutions. Meanwhile, the focus is on finding a technique that can effectively capture a large amount of solar energy. By running simulations, the goal is to pinpoint the exact placement of the inlet and outlet valves. In an attempt to prevent fluid from falling into the trap, the entry and the points of exit of the collector should be such that the fluid will not fall into the trap. COMSOL solves the problem by providing a numerical solution for the governing equations along with the boundary conditions.

3 Numerical approach and simulation setup

This analysis investigates the impact of inlet and outlet positions on the thermal performance of a solar collector. computational fluid dynamics (CFD) simulations are conducted using COMSOL Multiphysics to assess various inlet-outlet arrangements and identify the most efficient configuration for maximising heat transfer. The model is based on a rectangular plate heat exchanger, where fluid flow dynamics and heat conduction play a critical role in determining overall performance. Key components of the system include the solar collector surface, which absorbs solar radiation and transfers heat to the working fluid; the inlet and outlet, which regulate the flow path; and the internal region where heat exchange occurs. Three inlet-outlet configurations are studied: aligned (same direction), perpendicular, and opposite sides. Performance metrics such as temperature distribution, velocity profiles, and heat transfer rates are used to evaluate each configuration.

In the architectural model, the working fluid enters the system at a specific velocity and temperature, flows through the collector while absorbing heat, and exits at a higher temperature. The inlet and outlet positions strongly influence the efficiency of this energy transfer, as they affect the residence time and interaction with the collector walls. The simulation solves the governing equations of fluid flow and heat transfer, including the continuity equation (for mass conservation), the Navier-Stokes equations (for momentum conservation), and the energy equation (for thermal transport). The Reynolds number characterises the laminar flow regime. Boundary conditions are applied as follows: velocity inlet defined by Reynolds number, pressure outlet set to atmospheric conditions, and no-slip walls to realistically model fluid-structure interaction. The computational domain is discretised using a structured triangular mesh with boundary layer refinement to enhance solution accuracy. Grid independence is achieved by testing various mesh sizes. The finite element method (FEM) is used to solve the coupled equations iteratively (Pouraminian et al., 2024; Alizadeh et al., 2023). Comparative analysis of the flow and thermal fields reveals that the configuration with an inlet and outlet on opposite sides delivers the highest heat transfer efficiency due to the extended fluid travel path and enhanced mixing. The perpendicular arrangement induces localised vortices and moderate efficiency, while the same-direction configuration results in the lowest performance.

The study considers the governing equations essential for modelling the fluid dynamics and heat transfer in the solar collector. These include the continuity equation for mass conservation, the Navier-Stokes equations for momentum conservation, and the energy equation for heat transfer. To ensure accurate simulation, these equations are solved concurrently, with their interactions being analysed in the coupled framework. The continuity equation ensures the mass balance within the system. In contrast, the Navier-Stokes equations govern the movement of the fluid by accounting for the effects of pressure and viscous forces. The energy equation, on the other hand, captures the thermal transport within the collector, considering both convection through the air and solid conduction, as well as solar heat input. The solution of these equations simultaneously is crucial for modelling the complex interactions between fluid flow and heat transfer. The Navier-Stokes equations influence the velocity field, which in turn affects the temperature distribution as governed by the energy equation. In the presence of heat transfer, the velocity gradients and temperature gradients are interconnected, leading to complex flow patterns that directly influence the heat exchange efficiency. By solving these equations concurrently, we account for the coupled nature of fluid dynamics and heat transfer, ensuring a more realistic representation of the system’s behaviour. This approach significantly enhances the accuracy of the model and provides a more reliable basis for performance predictions.

Compared to traditional experimental or analytical approaches, the use of CFD in COMSOL Multiphysics offers distinct advantages. While experimental methods provide high accuracy, they are often costly and less flexible in terms of geometry or flow variation. Analytical models, though computationally inexpensive, typically require assumptions that may oversimplify the physical behaviour. In contrast, CFD simulations allow detailed modelling of conjugate heat transfer and fluid dynamics under multiple inlet-outlet configurations with parametric control. This makes the approach both efficient and scalable for design optimisation in solar energy systems. The energy equation is fundamental for modelling thermal transport across the solar collector. It incorporates several terms that contribute to heat transfer, including convection through the air, conduction through solid materials, and solar heat input. These terms are critical in understanding the overall thermal performance of the collector, as they account for the various mechanisms that facilitate heat exchange between the working fluid, the collector surface, and the environment. To improve the accuracy of the model, these components should be observed in the broader range of thermal dynamics and heat transfer processes. This includes considering factors such as radiation heat transfer, the impact of different fluid flow configurations on heat exchange efficiency, and the overall system behaviour over a range of operational conditions. By including these broader factors, the energy equation becomes a more comprehensive tool for analysing the collector's thermal performance across various scenarios.

This CFD-based methodology offers a clear comparison of various configurations and provides valuable insights into the optimal inlet-outlet arrangement for maximising thermal performance. The results not only contribute to designing more efficient solar collectors but also serve as a foundation for future studies on solar energy systems and renewable energy technologies. In conclusion, positioning the inlet and outlet on opposite sides significantly enhances heat absorption and thermal efficiency, providing a robust foundation for the design of advanced solar collectors. The proposed CFD-based methodology also serves as a versatile tool for future studies in renewable energy system optimisation. The core components modeled include (i) the solar collector surface, which absorbs solar radiation and transfers heat to the working fluid; (ii) the inlet and outlet positions, which determine flow paths and residence times; and (iii) the heat transfer zone where fluid-structure interaction governs energy exchange. Three distinct inlet/outlet configurations were selected for analysis:

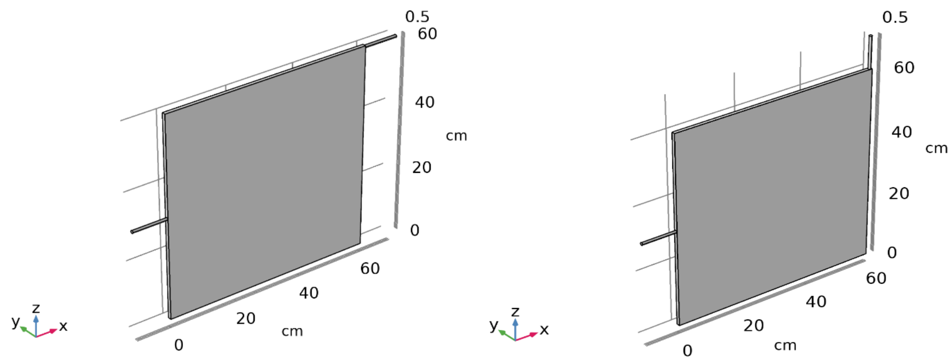
- *same-side configuration* – inlet and outlet on the same edge
- *perpendicular configuration* – inlet and outlet placed on adjacent sides
- *opposite-side configuration* – inlet and outlet on opposing edges.

These configurations were specifically chosen based on their frequent application in practical flat-plate solar collectors. Each provides unique advantages in terms of flow uniformity, pressure drop, and heat absorption potential. The same-side layout simulates a compact system design with minimal piping. The perpendicular setup encourages vortex generation and localised heat transfer due to fluid redirection. The opposite-side configuration allows for an extended fluid path and longer residence time, which typically promotes enhanced heat absorption. Overall, this methodology provides a comparative framework for optimising solar collector design through fluid path management. It serves as a foundation for future integration of adaptive fluid control strategies in renewable energy systems.

4 CFD simulation

An illustration of the geometry that is being studied in this research can be found in Figure 1. Essentially, this geometry consists of a plate solar collector inside which fluid flows. Fluid enters the collector horizontally through a tubing with a diameter of 0.9 cm and exits the culture through a tube with the same diameter (Figure 1(a)) or vertically. The purpose of the research is to determine the optimal inlet and outlet of the collector to achieve maximum thermal conduction (average temperature of the fluid exiting the pipe). Approximately 60 cm in length, 1 cm in width, and 1 cm in depth are the dimensions of the collector (Figure 1).

Figure 1 Geometry under study, (a) horizontal outlet (b) vertical outlet (see online version for colours)



(a) (b)

For the current problem, the following boundary conditions are appropriate:

- The inlet surface of the inlet pipe has been subjected to velocity boundary conditions. Thus, based on the input Reynolds number, the average velocity on this boundary is calculated, and based on this velocity, the fully developed velocity profile (fully developed flow) for this boundary is assigned. As a function of the average inlet velocity, the Reynolds number is defined as follows:

$$\mathrm{Re} = \frac{\rho U_{mean} D}{\mu} \tag{1}$$

  where ρ and μ are the density and dynamic viscosity of the fluid, and U mean, and D are the average velocity of the pipe inlet and the diameter of the pipe, respectively. Water fluid has been used as the working fluid in this research.

- It is considered that the outlet boundary (right boundary) represents the outlet boundary of constant pressure with atmospheric pressure.

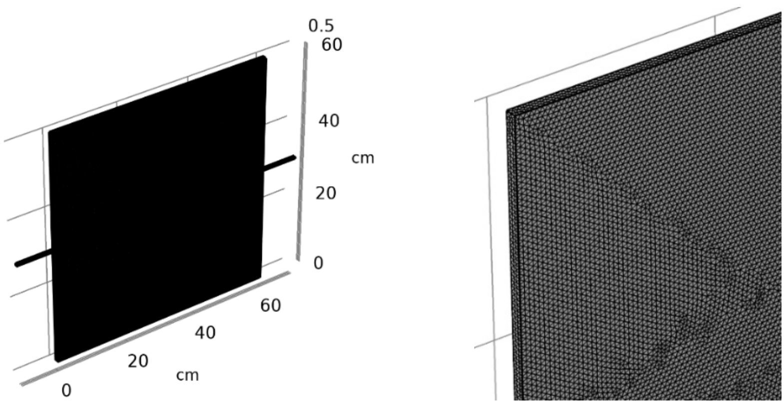
- On the entire solid surface, hydrodynamic non-slip conditions (zero velocity) have been applied.

- In this example, it is assumed that the fluid reaches the collector at a constant temperature of 293.15K.

- The outflow conditions consider the output. Therefore, there is no temperature gradient in the path of movement in the outlet of the culture fluid.

A mesh has been generated in the computational domain using 2D triangular elements. The mesh around has been reduced in size to provide better simulation and more acceptable results. The velocity gradients and other variables can be predicted more accurately by using a boundary layer mesh around the wall. A sample mesh employed in this investigation is displayed in Figure 2.

Figure 2 Mesh in the solution domain



To ensure accuracy and reproducibility, a mesh sensitivity analysis was conducted. A structured triangular mesh was used, with refinement near the boundary layers to capture thermal gradients. Three mesh densities were tested (coarse, medium, and fine), and results were considered grid-independent when temperature differences between the medium and fine meshes were below 1%. The final mesh used in the simulations contained approximately 640,000 elements. The simulations employed a segregated solver in COMSOL Multiphysics with second-order accuracy, and convergence was achieved when residuals for momentum and energy equations fell below 10^{-6}. The steady-state approach was used for all cases, with no temporal variation, to focus on thermal equilibrium behaviour. A direct linear solver (PARDISO) was applied for improved numerical stability and convergence performance. COMSOL software is employed to resolve the fluid domain using the Laminar Flow model. Throughout this study, it is assumed that the velocity of the flow entering the channel remains within the limits of laminar flow (Reynolds number less than 2,000).

Therefore, it is not necessary to model turbulence or to solve turbulence equations. Because the simulation is performed in a time-dependent and unstable manner, the correct time step must be chosen for the solution. Among the indicators of simulation accuracy in numerical solutions is the residual of the solution. Figure 3 illustrates the residuals of solving the problem and converging in the current simulation. The performance evaluation is conducted through a multi-step analytical framework that includes temperature distribution, velocity profile analysis, and quantification of heat transfer rates. These metrics are examined sequentially: first, the thermal gradients across the collector surface are assessed to identify heat absorption zones; second, the fluid flow behaviour is evaluated through velocity contour plots to detect recirculation zones or stagnation points; and finally, the heat transfer rates are computed and compared across configurations to determine the most effective arrangement. This structured approach ensures both adequacy and depth in capturing the collector's thermal performance.

Figure 3 Residual error solution (see online version for colours)

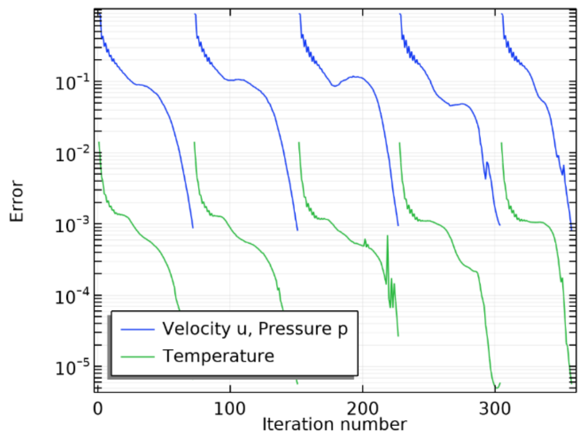


Table 1 Boundary conditions and simulation setup for solar collector analysis

Boundary condition	*Description*
Inlet	• Inlet velocity: determined based on the Reynolds number, with the average inlet velocity calculated. • Inlet temperature: 293.15 K (constant temperature for the inlet fluid).
Outlet	• Outlet pressure: constant pressure set to atmospheric pressure. • Outlet temperature conditions: no temperature gradient in the flow path at the outlet.
Solid surface	• No-slip condition: zero velocity on the solid surface.
Mesh	• 2D triangular elements: refined mesh near boundary layers for more accurate temperature gradients and other variables.
Boundary layer mesh	• Boundary layer mesh refinement: refined mesh around the boundary layers for more precise simulation.
Solver	• Segregated solver in COMSOL multiphysics: second-order accuracy. • Direct solver (PARDISO): used for numerical stability and improved convergence performance.
Reynolds number	• Laminar flow model: reynolds number less than 2,000; thus, turbulence modelling is not required.
Number of elements	• Final number of elements: approximately 640,000 elements.

As outlined in Table 1, the boundary conditions and simulation setup for the numerical analysis are defined based on the specifics of the solar collector system. The inlet velocity is determined according to the Reynolds number, with the temperature at the inlet fixed at 293.15 K. The outlet boundary is set to atmospheric pressure, with no temperature gradient in the fluid exit path. Additionally, non-slip conditions are applied to the solid surfaces, and the simulation employs a refined boundary layer mesh to ensure accurate thermal gradient predictions.

5 Results

The study has divulged substantial results of the computational analysis of the effect of inlet and outlet position on the thermal performance of a solar collector, thereby predicting the best designs for high heat transfer efficiency. Three main configurations subjected to scrutiny- that involved inlet and outlet in the same direction, inlet and outlet in a direction perpendicular to each other, and inlet and outlet positioned opposite each other- were found to be among the many immediate configurations. Consequently, it was possible to relate the placement of an inlet and outlet with the thermal conduction rate, velocity distribution, and temperature gradients within the collector. The optimal thermal conduction was achieved when the inlet and outlet were placed opposite to each other on the same side of the collector because the working fluid could have traversed the longest distance within the collector, thereby promoting more heat exchange. In this case, the computational results indicate that the peak outlet temperature for this configuration was approximately 5% higher than for cases where the inlet and outlet were positioned in the same direction. In addition, the thermal conduction rate of this configuration, termed optimal, was greater by nearly 1.5% than that of other configurations. Analysed collector velocity profiles show quite a variation with different configurations.

The highest velocity was recorded for the configuration in which the inlet and outlet were opposite each other, thus enhancing the convective heat transfer. Configuration poles generated local turbulence whenever the flow struck and bounced off the opposite wall. This increased heat transfer in the localised region but also led to higher pressure drops and possible inefficiencies in the long run. The aligned configuration, where the inlet and outlet were in the same direction, resulted in the lowest velocity and thermal conduction efficiency. The working fluid, therefore, travelled the shortest distance, leading to less effective heat absorption and reduced residence time in the collector. This was confirmed by the temperature distribution plots, which showed lower peak outlet temperatures and higher temperature stratification in this setup. Some significant findings emerged from the comparison of all the configurations. Two systems perform much better than the other, with most parameters cross-referencing them, while the counter-current configuration scored beyond the reach of other setups in all parameters evaluated since its capacity outlet temperature was complemented with the highest heat transfer efficiency of 92%. The perpendicular configuration aided moderately due to the different mixing and localised turbulence effects, falling short of the opposite direction configuration in performance.

Comparatively, a same-direction setup argued the least thermal efficiency, as the minimum travel distance of fluid brought it on the heated surfaces, thereby absorbing comparatively less heat. These findings were further substantiated by flow line and vortex analysis. A huge stagnant zone occurred at the same place when they were in the same direction setup, thereby limiting their ability to take full advantage of convective heat transfer. Some impingement areas are created by that perpendicular configuration, which increases turbulence and mixing of fluid by an impact but requires further energy dissipation as well. The opposite direction configuration had the most uniform distribution of flow and minimised stagnation areas while at the same time realising a highly effective heat transfer process across the total collector surface.

Results indicated that for proper selection of inlet and outlet conditions, the thermal conduction rate in the collector is increased by about 1.5% for an inlet-outlet aligned in the same direction. This was also analysed regarding the influence of such positions on the collector's temperature-absorbing capacity. Because the outlet was in the centre of the collector, the fluid stream was diverted towards the outlet by hitting the opposite wall first. This created a significant low-temperature vortex in the half closest to the outlet; the liquid jet exited through the outlet after heat exchange with still liquid inside the collector, resulting in a comparatively larger low-temperature zone than in any other cases, thereby improving heat-exchange efficiency. Some minor differences were observed in the flow of fluid and thermal conduction in various perpendicular layouts for the change direction after hitting the opposite wall and exiting from the collector vertically. This kind of movement affects the temperature distribution, resulting in lower cooling effects and higher outlet temperatures compared with the same direction setup. It has been derived from the flow patterns that for a case wherein first strikes at the opposite wall, there is a contribution of vortex formations toward mixing and improved thermal conduction, but the highest efficiency still occurs when the inlet and outlet are located opposite each other. From this, it is clear that the location of the inlet and the outlet plays a crucial role in optimising the thermal performance of a solar collector.

Figure 4 The collector's inlet and outlet

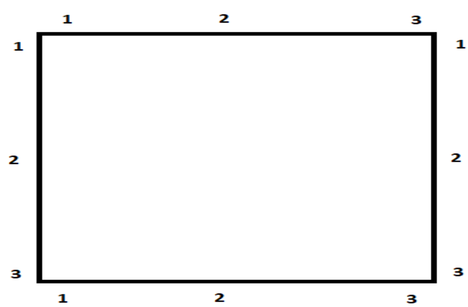
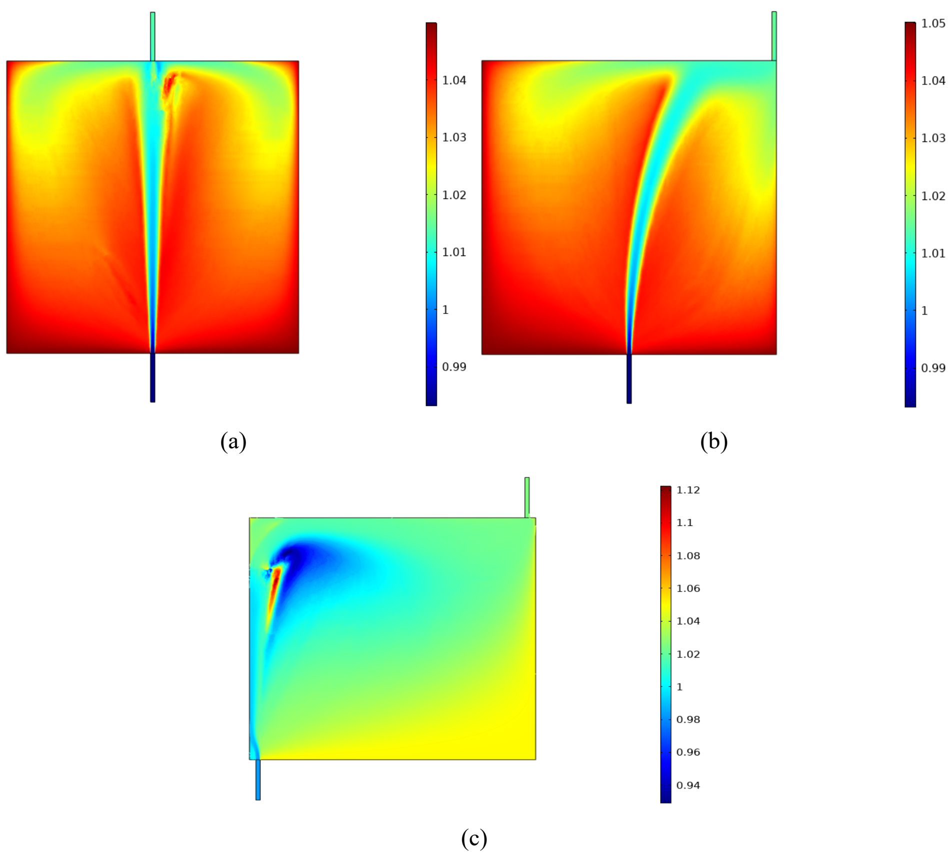


Figure 4 shows the inlet and outlet configurations for the solar collector numbered systematically from 1 to 3 horizontally and vertically. The numbering above is a systematic definition for different inlet and outlet arrangements in the CFD simulations conducted to analyse thermal conduction efficiency. Figure refers to how they access the dynamics of the flow, heat transfer rates, and temperature distributions due to the switching of inlet and outlet locations. Thus, the best thermal performance was found when the inlet and outlet were opposite because that configuration had the longest flow path, giving it the most heat absorption and conduction. With such a visual representation of these configurations, a clear basis for comparing different setups concerning maximising efficiency per collector becomes clear.

5.1 *Effect of outlet location on the heat absorbed by the collector with inlet and outlet in the same direction*

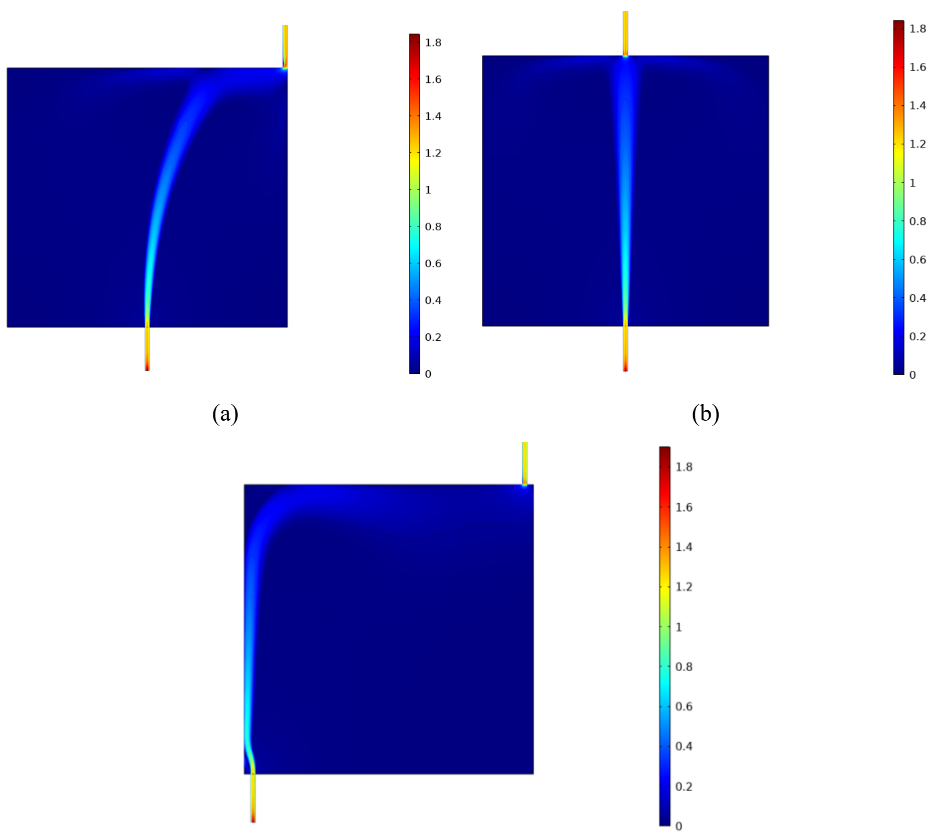
Specifically, in this part of the study, the effect of changing the location of the inlet and outlet on the speed and thermal spread within the solar collector is examined. Generally, the inlet and outlet of the collector are placed horizontally (the inlet and outlet are oriented in the same direction represented in Figure 4), with the inlet being horizontal and the outlet being horizontal. By optimising the placement of the inlet and outlet, the goal is to maximise the flux absorbed by the collector. Based on the discussion in the previous section, it is possible to identify the degree of thermal conduction in the collector by observing the average temperature of the fluid outlet with other parameters remaining constant. Figure 5 shows the dimensionless temperature contours on the collector's middle plate in 3 states, including when the inlet and outlet are positioned opposite each other (mode 2-2), when the input is in the middle and the outlet is in the sides (mode 1-2), and when the output is in the corners (mode 1-3). Many situations are either similar to these situations or can be justified by combining them.

Figure 5 Dimensionless temperature contour in the central plane of the collector in the input and output mode in the same direction for the modes (a) 2-2, (b) 2-1, and (c) 1-3 (see online version for colours)



If both the inlet and outlet are situated opposite each other, as can be seen in the figures above, the fluid will follow the shortest path from the inlet to the outlet and enter directly from the inlet to the outlet. As a result, the rate of thermal conduction is expected to be relatively low in this example. Based on the dimensionless temperature contours, it can be determined that the peak temperature in the computational domain is only about 4% more than the average inlet temperature. The process is slightly different in mode 2-2, where the input is from the middle of the collector, and the output is from the corners. It can be seen that the inlet flow enters the computational domain as an enclosed jet, which expands along its path inside the jet collector. This results in the jet expanding near the outlet and changing the temperature of the corresponding component. The maximum outlet temperature is approximately 12% higher than the inlet temperature in this case, which indicates an improvement in thermal conduction to the collector relative to the previous case. To understand this behaviour, the velocity contours within the computational domain can be examined.

Figure 6 Dimensionless velocity contour in the central plane of the collector in the input and output mode in the same direction for the modes (a) 2-2, (b) 2-1, and (c) 1-3 (see online version for colours)



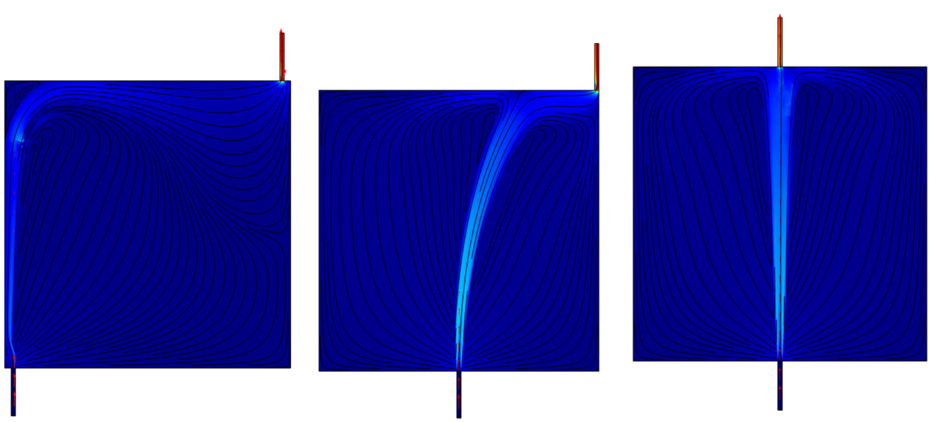
(c)

According to Figure 6, these contours are depicted for the modes that have been investigated. As can be seen, the maximum dimensionless velocity inside each collector is almost the same in all three cases. Because the convective thermal conduction coefficient is a function of the speed and intensity of heat exchange, it can be concluded from these contours that in the current problem, maximising the amount of temperature exchange, i.e., the path by which the flow travels, is the most important objective. The numerical results confirm the hypothesis that the farther the distance between input and output, the better the results.

Flow lines in different states are examined in Figure 7 to investigate the effects of the flow pattern on the heat absorbed. Additionally, the contours indicate the amount of convective flux that has been absorbed. Figure 7(a) reveals that the input current divides the computing domain into two equal parts when input and output are opposite one another. This results in the formation of large vortices on both sides with a low speed. Vortices cause heat to be trapped, and no heat is conveyed to the incoming flow as a result. In the case of an inlet in the middle and an outlet from the sides [Figure 7(b)], the vortices would stretch on one side and gather on the other. As a result, the vortices gather on the side with higher velocity, and thermal conduction takes place more efficiently on this side. In contrast, in this case, the inlet jet first strikes the opposite wall before deviating towards the outlet, which also contributes to an increase in thermal conduction.

It is noteworthy that in the third condition, the flow travels almost along one side before changing direction at the corner along the perpendicular side. The final section of this side concludes with the departure from the computing domain. This situation results in the formation of a vortex in the corner, which traps heat in that region. The law of conservation of mass also dictates that vortices form within the field under the influence of the jet movement.

Figure 7 Flow lines in the central plate of the collector in the input and output mode in the same direction for the modes (a) 2-2, (b) 2-1, and (c) 1-3 (see online version for colours)



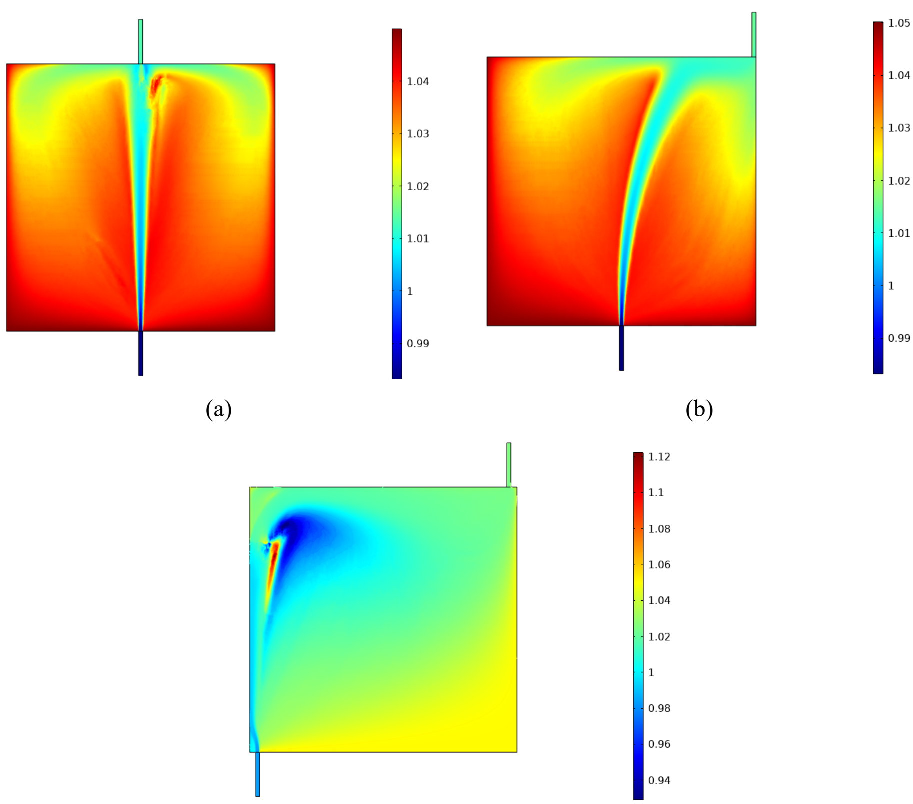
(a) (b) (c)

The results reveal that with the proper selection of the inlet and outlet locations, the thermal conduction rate in the collector will increase by about 1.5% when the inlet and outlet points are in the same direction.

5.2 The impact of perpendicular inlet and outlet positions on the heat absorbed by the collector

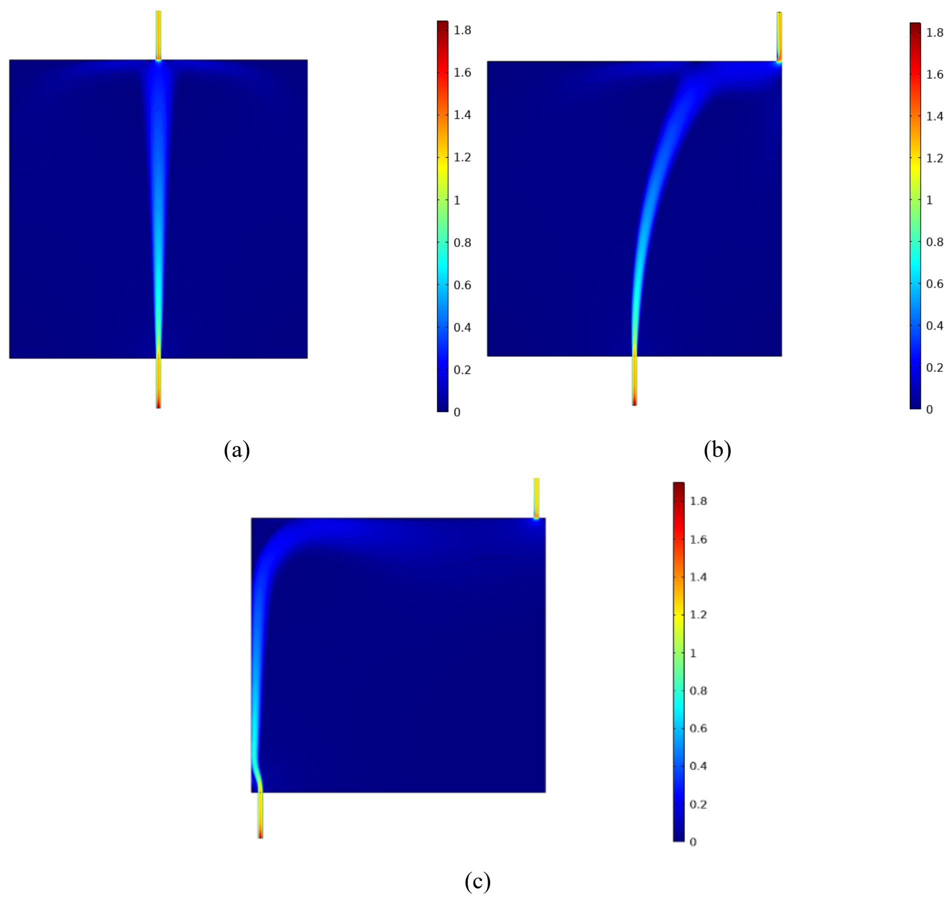
In this part, fluid movement and internal thermal conduction have been explored in a state where the inlet and outlet are not perpendicular to one another, for example, a horizontal inlet and a vertical outlet. As in the previous part, it can be shown that it is possible to analyse all states by analysing three basic states. It is important to note that the letter V serves to differentiate horizontal from vertical outputs in this case. In the example of mode V3, the output is displayed in position three with a vertical orientation. An illustration of the contours of the collector's central plate can be found in Figure 8. From Figure 8(a), it is obvious that the current enters the collector from the middle and proceeds to the opposite wall. It is at this point that the flow is split into 2 branches. A portion of the flow acts as an impinging jet on impact with the opposite plate, and it returns to the source. A vortex is formed on the side opposite to this collision. There is also another portion of the flow that tilts its way toward the outlet and leaves it directly. In this case, low-temperature areas will be observed in the upper part of the collector and near the opposite wall. However, the temperature of the inlet flow has little effect on the temperature of the other parts of the collector.

Figure 8 Dimensionless temperature contour in the central plane of the collector in the input and output states perpendicular to each other for the states (a) 1v-2, (b) 2v-2, and (c) 1v-1 (see online version for colours)



When the outlet is positioned in the middle of the collector [Figure 8(b)], the fluid flow is diverted towards the outlet by hitting the opposite wall first. During this process, a large vortex of low temperature is formed in half closest to the outlet, and the liquid jet exits the outlet after exchanging heat with the still liquid inside the collector. As can be seen in this case, the low-temperature area is larger than in the previous case, and consequently, this means that the heat exchange is improved compared to the previous case.

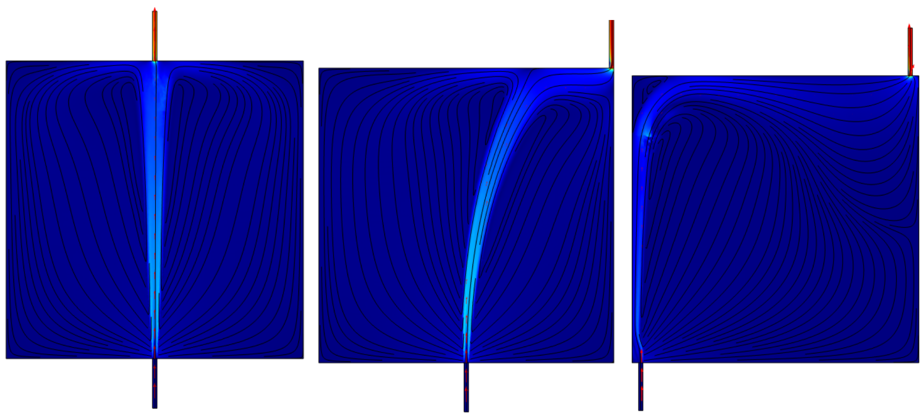
Figure 9 Dimensionless velocity contour in the central plane of the collector in the input and output states perpendicular to each other for the states (a) 1v-2, (b) 2v-2, and (c) 1v-1 (see online version for colours)



There is a slight difference between the movement of fluid and the thermal conduction field in the 1-1v state. The flow initially enters the solution domain in a horizontal direction, then changes direction by 90 degrees upon striking the opposite wall and exits the collector vertically. This case shows that the incoming flow affected the rest of the channel, except for the section located near the opposite corners of the flow exit, and decreased the temperature of the fluid in those sections. Due to this, one can expect that the collector was more effectively cooled by this mode than by the previous modes, resulting in a higher average fluid temperature at the collector outlet. As shown in

Figure 9, the temperature contours within the collector under this condition can be seen. In addition, Figure 10 displays the velocity vectors inside the collector and in the central plane.

Figure 10 Flow lines in the central plate of the collector in the inlet and outlet states perpendicular to each other for the states (a) 1v-2, (b) 2v-2, and (c) 1v-1 (see online version for colours)



(a) (b) (c)

Based on the flow patterns, it can be determined that in cases 10a and 10b, the flow first hits the opposite wall and is then directed towards the outlet by forming a vortex. It should be noted that the inflow from the entrance to the opposite wall acts as a free jet as well. When the flow encounters the opposite wall, it will act as an impinging jet, and once it impacts the opposite wall, it will turn into a rotating vortex flow. A visual inspection of the movement pattern in case 10c displays the formation of a strong and elongated vortex near the inlet. This part of the flow is cooled by these vortices, which help to mix the flow and mix the flow. Also, the other branches of the vortex are spread inside the collector, which in turn results in the mixing of the movement and the increase of thermal conduction in these areas. Of course, the influence of the flow in these areas is less and is proportional to the movement speed value. The movement speed decreases as the main movement moves away. So, in a part of the flow that is on the opposite side of the outlet, the rotating currents and the speed of the secondary flow are so small that these parts are not affected by the incoming flow, and their temperature remains constant. As the maximum outlet temperature is determined by comparing the inlet and outlet currents in the same direction (state 3-1), and the average dimensionless temperature of the outlet is about 1.026, this value suggests that state 1-3 has a superior thermal conduction efficiency when the inlet and outlet are oriented in the same direction as when they are oriented perpendicular to each other.

6 Discussion

The results presented in this study did promise very valuable insights into the influence of inlet and outlet placement on thermal efficiency as far as solar collectors are concerned.

Indeed, one can say that the arrangements of all fluid flow paths within a collector have determinant characteristics influencing heat transfer within the collector, such as temperature distribution as well as the overall performance of the system. A comparative analysis of different configurations has demonstrated that measuring an optimum inlet and outlet position could result in some significant measurable improvements in thermal conduction, fluid mixture, and energy utilisation by comparison.

A very interesting one from the study is that the highest thermal conduction path would be obtained when the inlet and outlet were kept counter to each other. This allows such a long distance for the working fluid to flow inside the collector, which improves its contact with the heated collector surfaces and optimises the overall heat transfer. This, in turn, improves the travel distance and, as a consequence, better mixing and less thermal stratification, thus improving efficiency in heat absorption. This fact is confirmed by the velocity profiles and temperature contours, which prove that this arrangement minimises stagnant zones that are associated with inefficient heat transfer, as referred to in less optimal configurations. Testing a perpendicular inlet/outlet showed some improvement in thermal performance, but again, it was not as good as the opposite direction. The opposing wall has localised turbulence, hence improving convective heat transfer due to the fluid being bombarded against the wall. However, it also often produces high-pressure losses of flow resistance, which may affect system stability in the long run. Although vigorous mixing can be advantageous, the picture also shows that there can be deleterious energy waste resulting from excessive turbulence, and this is demonstrated by vortex formations at certain locations in the collector. The aligned inlet-outlet configuration, where both ports are positioned in the same direction, was said to have the least performance in terms of heat transfer efficiency. The research concluded that in such a case, the fluid travels the shortest route from inlet to outlet, hence lower residence time inside the collector and hence little heat absorbed by the working fluid. This arrangement was seen to correlate with low thermal conduction overall, as indicated by lower temperature differentials recorded in the simulations. The configuration is mandated further by the formation of temperature gradients and stratified flow regions that reduce its effectiveness even more.

Table 2 Comparative summary of thermal performance for different inlet-outlet configurations

Configuration (inlet-outlet)	*Outlet temp. increase (%)*	*Thermal efficiency (%)*	*Max. velocity (dimensionless)*	*Vortex presence*	*Remarks*
Same direction (2-2)	~4%	Baseline (ref.)	Moderate	Yes	Shortest flow path, high stratification
Same direction (2-1)	~12%	+1.5%	Moderate	Yes	The jet expansion improves conduction
Same direction (1-3)	~10%	+1.0%	Moderate	Yes	Heat-trapping in corner vortex
Perpendicular (V3)	~8%	+0.7%	Moderate-high	Yes	Mixed flow, localised turbulence
Opposite direction (optimal)	~14-15%	92% (highest)	Highest	No	Longest path, uniform conduction

The analysis reveals that the inlet and outlet positions significantly affect the thermal efficiency of the collector. In the crosswise configuration, the fluid jet impacts the opposite wall before exiting, creating vortices that enhance localised heat transfer. The highest thermal efficiency is observed when the inlet and outlet are positioned opposite each other, allowing the fluid to travel a longer distance for optimal heat absorption. Temperature distribution and velocity vectors support that this configuration yields the highest outlet temperature, making it the most suitable for solar collector design. As summarised in Table 2, the configuration with opposite inlet and outlet positions demonstrates the highest thermal efficiency and outlet temperature increase, achieving approximately 92% thermal efficiency and a 14–15% outlet temperature increase. This confirms the longer flow path significantly enhances heat transfer.

7 Conclusions

Using the dynamic method of fluid calculations and COMSOL software, this exploration investigated for the first time the effects of the inlet and outlet of the water fluid flow on the productivity of a solar collector. As a result of this research, the following findings have been identified:

- It is noteworthy that the location of the inlet and outlet substantially affects the movement area and thermal conduction within the solar collector.
- The best thermal conduction rate was obtained when the inlet and outlet of the collector were on the same side. Three different configurations were examined, including inlets and outlets perpendicular to each other, inlets and outlets perpendicular to each other, inlets and outlets perpendicular to each other, and inlets and outlets perpendicular to each other. The input and output must be oriented in the same direction.
- To attain the ideal thermal conduction rate, the inlet and outlet of the culture should be located on the same side of the collector.
- The results display that the thermal conduction rate in the collector elevated by 1.5% when the inlet and outlet were in the same direction when the location of the inlet and outlet was properly selected.

Acknowledgements

The author would like to take this opportunity to acknowledge that there are no individuals or organisations that require acknowledgment for their contributions to this investigation.

Declarations

All authors declare that they have no conflicts of interest.

The investigators claim no competing interests.

Data is accessible upon request.

This investigation received no specific grant from any funding agency in the public, commercial, or not-for-profit sectors.

The paper has received ethical approval from the institutional review board, ensuring the protection of participants' rights and compliance with the relevant ethical directions.

References

Aggarwal, S., Kumar, R., Lee, D., Kumar, S. and Singh, T. (2023) 'A comprehensive review of techniques for increasing the efficiency of evacuated tube solar collectors', *Heliyon*, Vol. 9, No. 4, pp.65–78.

Ajarostaghi, S.S.M. and Mousavi, S.S. (2022) 'Solar energy conversion technologies: principles and advancements', in *Solar Energy Advancements in Agriculture and Food Production Systems*, pp.29–76, Elsevier, Netherland.

Alizadeh, M.H., Ajri, M. and Maleki, V.A. (2023) 'Mechanical properties prediction of ductile iron with spherical graphite using multi-scale finite element model', *Physica Scripta*, Vol. 98, No. 12, p.125270.

Alizadeh, S. (1999) 'An experimental and numerical study of thermal stratification in a horizontal cylindrical solar storage tank', *Solar Energy*, Vol. 66, No. 6, pp.409–421.

Alobaid, M., Hughes, B., Heyes, A. and O'Connor, D. (2018) 'Determining the effect of inlet flow conditions on the thermal efficiency of a flat plate solar collector', *Fluids*, Vol. 3, No. 3, p.67.

Alshukri, M.J., Hussein, A.K., Eidan, A.A. and Alsabery, A.I. (2022) 'A review on applications and techniques of improving the performance of heat pipe-solar collector systems', *Solar Energy*, Vol. 236, No. 4, pp.417–433.

Ameer, P. and Jalil, A. (2024) 'The effect of hormonal supplements on metabolic syndrome', *FMDB Transactions on Sustainable Applied Sciences*, Vol. 1, No. 2, pp.74–84.

Anand, P.P., Jayanth, G., Rao, K.S., Deepika, P., Faisal, M. and Mokdad, M. (2024) 'Utilising hybrid machine learning to identify anomalous multivariate time-series in geotechnical engineering', *AVE Trends in Intelligent Computing Systems*, Vol. 1, No. 1, pp.32–41.

Assari, M.R., Tabrizi, H.B. and Savadkohy, M. (2018) 'Numerical and experimental study of inlet-outlet locations effect in horizontal storage tank of solar water heater', *Sustainable Energy Technologies and Assessments*, Vol. 25, No. 2, pp.181–190.

Chandra, Y.P. and Matuska, T. (2019) 'Stratification analysis of domestic hot water storage tanks: a comprehensive review', *Energy and Buildings*, Vol. 187, No. 3, pp.110–131.

Chandratreya, A. (2024) 'Psychological impacts of climate change and environmental degradation on mental well-being', *FMDB Transactions on Sustainable Techno Learning*, Vol. 2, No. 2, pp.75–87.

Conceicao, R., Gonzalez-Aguilar, J., Merrouni, A.A. and Romero, M. (2022) 'Soiling effect in solar energy conversion systems: a review', *Renewable and Sustainable Energy Reviews*, Vol. 162, No. 7, p.112434.

Dawood, M.M.K., Nabil, T., Kabeel, A.E., Shehata, A.I., Abdalla, A.M. and Elnaghi, B.E. (2020) 'Experimental study of productivity progress for a solar still integrated with parabolic trough collectors with a phase change material in the receiver evacuated tubes and in the still', *Journal of Energy Storage*, Vol. 32, No. 12, p.102007.

Dzikevics, M., Veidenbergs, I. and Valančius, K. (2020) 'Sensitivity analysis of packed bed phase change material thermal storage for domestic solar thermal system', *Environmental and Climate Technologies*, Vol. 24, No. 1, pp.378–391.

Elaiyaraja, P. and Boinapalli, N.R. (2024) 'Enhanced ferroelectric and piezoelectric properties in $NaNbO_3$-based materials via $BiGd/YKZrTiO_3$ two-step sintering', *FMDB Transactions on Sustainable Applied Sciences*, Vol. 1, No. 1, pp.10–20.

Eltawil, M.A. and Omara, Z.M. (2014) 'Enhancing the solar still performance using solar photovoltaic, flat plate collector and hot air', *Desalination*, Vol. 349, No. 9, pp.1–9.

Femi, D., Satheesh, C., Sakthivanitha, M., Maruthi, R., Gnanaguru, G. and Paslavskyi, M. (2024) 'Sustainable environmental optimization of abrasive water jet machining parameters of AZ31D/B_4C composite using TOPSIS technique', *FMDB Transactions on Sustainable Environmental Sciences*, Vol. 1, No. 3, pp.138–148.

Gnanaguru, G., Agalya, G., Hanirex, D.K., Poornima, V., Padmapriya, G. and García, A.G. (2024) 'Experimental investigation of sustainable hybrid composite material for marine application technique', *FMDB Transactions on Sustainable Environmental Sciences*, Vol. 1, No. 3, pp.149–160.

Gunjo, D.G., Mahanta, P. and Robi, P.S. (2017) 'CFD and experimental investigation of flat plate solar water heating system under steady state condition', *Renewable Energy*, Vol. 106, No. 6, pp.24–36.

Ha, V.H.T. (2024) 'Current situations of climate change in Ca Mau Province, Mekong Delta, Vietnam technique', *FMDB Transactions on Sustainable Environmental Sciences*, Vol. 1 , No. 2, pp.81–90.

Hoang, A.T., Le, T.H., Chitsomboon, T. and Koonsrisook, A. (2021) 'Experimental investigation of solar energy-based water distillation using inclined metal tubes as collector and condenser', *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, Vol. 47, No. 1, pp.1–17.

Hussein, H.M.S. (2007) 'Theoretical and experimental investigation of wickless heat pipes flat plate solar collector with cross flow heat exchanger', *Energy Conversion and Management*, Vol. 48, No. 4, pp.1266–1272.

Ikwuagwu, C.V., Achebe, C.N. and Ononiwu, N.H. (2024) 'Investigating variability in power capacity through experimental testing of lithium-ion batteries', *FMDB Transactions on Sustainable Energy Sequence*, Vol. 2, No. 1, pp.12–20.

Kabeel, A.E., Sathyamurthy, R., Mageshbabu, D., Madhu, B., Anand, P. and Balakrishnan, P. (2020) 'Effect of mass flow rate on fresh water improvement from inclined PV panel basin solar still', *Materials Today: Proceedings*, Vol. 32, No. 3, pp.374–378.

Kadam, S.S. and Deming, C. (2024) 'Reforming solar efficiency with a ground-breaking journey through advanced material innovations for high-performance photovoltaics', *FMDB Transactions on Sustainable Applied Sciences*, Vol. 1, No. 1, pp.44–53.

Kenjo, L., Inard, C. and Caccavelli, D. (2007) 'Experimental and numerical study of thermal stratification in a mantle tank of a solar domestic hot water system', *Applied Thermal Engineering*, Vol. 27, Nos. 11–12, pp.1986–1995.

Khodadadi, S., Ghanavati, A., Assari, M.R., Hatami, M. and Ganji, D.D. (2015) 'Experimental study on the water inlet and outlet position of a solar collector reservoir for maximum efficiency', *Journal of Mechanical Science and Technology*, Vol. 29, No. 5, pp.2279–2284.

Kumar, C.S., Sathya, A., Deb, R. and Rahman, M.M. (2024) 'Managing electronic waste: a comprehensive review of current state and challenges technique', *FMDB Transactions on Sustainable Environmental Sciences*, Vol. 1, No. 1, pp.10–18.

Li, Q., Huang, X., Tai, Y., Gao, W., Wenxian, L. and Liu, W. (2021) 'Thermal stratification in a solar hot water storage tank with mantle heat exchanger', *Renewable Energy*, Vol. 173, No. 8, pp.1–11.

Madhuranthakam, R.S. (2024) 'Scalable data engineering pipelines for real-time analytics in big data environments', *FMDB Transactions on Sustainable Computing Systems*, Vol. 2, No. 3, pp.154–166.

Mahadewi, N.M.E. (2024) 'Evaluating ecosystem resilience through innovative carbon sequestration methods amidst climate change technique', *FMDB Transactions on Sustainable Environmental Sciences*, Vol. 1, No. 2, pp.107–116.

Manikandan, S., Sakthivanitha, M., Dayana, T.R.N., Preethika, S.K.P., Ali, M.M.S. and Padmapriya, G. (2024) 'Experimental investigation of natural hybrid composite material for engineering application technique', *FMDB Transactions on Sustainable Environmental Sciences*, Vol. 1, No. 3, pp.126–137.

Mary, L., Anand, D.P., Xavier, A.C. and Selvam, L.A. (2024) 'Molecular structure, vibrational spectroscopic (FT-IR, FT-Raman), UV and NLO analysis of N-methylurea by density functional method', *FMDB Transactions on Sustainable Applied Sciences*, Vol. 1, No. 2, pp.54–63.

Mathew, I.O., Otaraku, I.J., Kareem, S.A., Oji, A. and Ikenyiri, P.N. (2024) 'Delignification of different lignocellulosic biomass using hydrogen peroxide and acetic acid (HPAC) technique', *FMDB Transactions on Sustainable Environmental Sciences*, Vol. 1, No. 1, pp.19–30.

Mehdipour, R., Baniamerian, Z., Golzardi, S. and Murshed, S.M. S. (2020) 'Geometry modification of solar collector to improve performance of solar chimneys', *Renewable Energy*, Vol. 162, No. 12, pp.160–170.

Mehta, G., Bose, S.R. and Selva Naveen, R. (2023) 'Optimizing lithium-ion battery controller design for electric vehicles: a comprehensive study', *FMDB Transactions on Sustainable Energy Sequence*, Vol. 1, No. 2, pp.60–70.

Munshi, M. (2024) 'Review on pulsed electric fields: a new horizon in food processing health techniques', *FMDB Transactions on Sustainable Health Science Letters*, Vol. 2, No. 1, pp.1–9.

Nomula, V.K., Steffi, R. and Shynu, T. (2023) 'Examining the far-reaching consequences of advancing trends in electrical, electronics, and communications technologies in diverse sectors', *FMDB Transactions on Sustainable Energy Sequence*, Vol. 1, No. 1, pp.27–37.

Pandey, P., Lawanya, T. and Hasan, S.M. S. (2024) 'Heat transfer analysis of a stretching sheet in TRI particle-enhanced nanofluid systems', *FMDB Transactions on Sustainable Applied Sciences*, Vol. 1, No. 1, pp.21–33.

Panyaram, S. (2024) 'Enhancing performance and sustainability of electric vehicle technology with advanced energy management', *FMDB Transactions on Sustainable Energy Sequence*, Vol. 2, No. 2, pp.110–119.

Pierre, T.J., Rahman, M.M., Amin, M.A. and Munshi, M. (2024) 'Deciphering urban footprints on climate variability through an innovative inquiry into heat island phenomena', *FMDB Transactions on Sustainable Applied Sciences*, Vol. 1, No. 1, pp.34–43.

Pouraminian, M., Akbari Baghal, A.E. andalibi, K., Khosravi, F. and Arab Maleki, V. (2024) 'Enhancing the pull-out behavior of ribbed steel bars in CNT-modified UHPFRC using recycled steel fibers from waste tires: a multiscale finite element study', *Scientific Reports*, Vol. 14, No. 1, p.19939.

Raj, A., Gupta, A.K., Pachauri, R.K. and Sharma, V. (2024) 'Performance evaluation of Walrus Optimization-based MPPT with other algorithms under partial shading condition', *FMDB Transactions on Sustainable Energy Sequence*, Vol. 2, No. 2, pp.88–101.

Ramya, L.N., Bose, S.R., Selva Naveen, R. and Islam Chowdhury, R. (2023) 'Advanced solar charge controller: integrating mppt technology and online data logging for efficient energy management', *FMDB Transactions on Sustainable Energy Sequence*, Vol. 1, No. 2, pp.107–120.

Saxena, R.R., Saxena, R. and Patel, A. (2023) 'Functional electrical stimulation as a significant bioelectronic intervention in the domain of fitness: a review', *FMDB Transactions on Sustainable Energy Sequence*, Vol. 1, No. 2, pp.94–106.

Senapati, B., Regin, R., Rajest, S.S., Paramasivan, P. and Obaid, A.J. (2023) 'Quantum dot solar cells and their role in revolutionizing electrical energy conversion efficiency', *FMDB Transactions on Sustainable Energy Sequence*, Vol. 1, No. 1, pp.49–59.

Shenbagavalli, T., Lakshmi, K.V.N., Nikhil, M.S. and Mehmood, A. (2024) 'Greenhouse gas control and sustainable economic development: an environmental impacts analysis technique', *FMDB Transactions on Sustainable Environmental Sciences*, Vol. 1, No. 1, pp.1–9.

Shoeibi, S., Mirjalily, S.A.A., Kargarsharifabad, H., Khiadani, M. and Panchal, H. (2022) 'A comprehensive review on performance improvement of solar desalination with applications of heat pipes', *Desalination*, Vol. 540, No. 10, p.115983.

Shoeibi, S., Saemian, M., Khiadani, M., Kargarsharifabad, H. and Mirjalily, S.A.A. (2023) 'Influence of PV/T waste heat on water productivity and electricity generation of solar stills using heat pipes and thermoelectric generator: an experimental study and environmental analysis', *Energy Conversion and Management*, Vol. 276, No. 1, p.116504.

Siddique, M., Sarkinbaka, Z.M., Abdul, A.Z., Asif, M. and Elboughdiri, N. (2023) 'Municipal solid waste to energy strategies in Pakistan and its air pollution impacts on the environment, landfill leachates: a review', *FMDB Transactions on Sustainable Energy Sequence*, Vol. 1, No. 1, pp.38–48.

Sikiru, S., Oladosu, T.L., Amosa, T.I., Kolawole, S.Y. and Soleimani, H. (2022) 'Recent advances and impact of phase change materials on solar energy: a comprehensive review', *Journal of Energy Storage*, Vol. 53, No. 9, p.105200.

Singh, J. (2024a) 'Advancements in understanding and mitigating shock wave boundary layer interaction (SWBLI) effects on hypersonic airfoil performance: a comprehensive review', *FMDB Transactions on Sustainable Energy Sequence*, Vol. 2, No. 2, pp.102–109.

Singh, J. (2024b) 'Predicting the gamma-ray impact on Earth from potential supernovae of Betelgeuse, Sirius, and Proxima Centauri', *FMDB Transactions on Sustainable Applied Sciences*, Vol. 1, No. 2, pp.64–73.

Srilakshmi, U., Sandhya, I., Manikandan, J., Bindu, A.H., Regin, R. and Rajest, S.S. (2024) 'Design and implementation of energy-efficient protocols for underwater wireless sensor networks', in *Advances in Computer and Electrical Engineering*, pp.417–430, IGI Global, USA.

Srinivasan, P. and Arulvendhan, K. (2024) 'An enhanced low voltage ride-through control scheme for DFIG-based grid-integrated wind energy conversion systems: environmental impact analysis technique', *FMDB Transactions on Sustainable Environmental Sciences*, Vol. 1, No. 3, pp.161–172.

Suraj, D., Sulthan, N., Balaji, R., Swathi, R., Roberts, C. and Vaidianathan, B. (2024) 'Data-driven and domain adaptation framework for optimizing energy usage forecasting in smart buildings using ARIMA model', *FMDB Transactions on Sustainable Energy Sequence*, Vol. 2, No. 1, pp.1–11.

Tin, H.H.K., Thu, S. and Maung, K.K. (2024) 'A systematic review of rice husk gasifiers: technology, performance, and sustainability technique', *FMDB Transactions on Sustainable Environmental Sciences*, Vol. 1, No. 1, pp.31–40.

Uçkan, İ. and Yousif, A.A. (2021) 'Investigation of the effect of various solar collector types on a solar absorption cooling system', *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, Vol. 43, No. 7, pp.875–892.

Verduci, R., Romano, V., Brunetti, G., Yaghoobi Nia, N., Di Carlo, A., D'Angelo, G. and Ciminelli, C. (2022) 'Solar energy in space applications: review and technology perspectives', *Advanced Energy Materials*, Vol. 12, No. 29, p.2200125.

Victor Ikwuagwu, C., Ononiwu, N.H. and Adeyemi, K. (2023) 'Comprehensive energy analysis and performance evaluation of lithium-ion battery integration in photovoltaic systems: a comparative study on reliability and environmental impact', *FMDB Transactions on Sustainable Energy Sequence*, Vol. 1, No. 2, pp.83–93.

Zachar, A., Farkas, I. and Szlivka, F. (2003) 'Numerical analyses of the impact of plates for thermal stratification inside a storage tank with upper and lower inlet flows', *Solar Energy*, Vol. 74, No. 4, pp.287–302.

Zanardo, E. (2024) 'Chronostamp: a general-purpose run-time for data-flow computing in a distributed environment', *AVE Trends in Intelligent Computing Systems*, Vol. 1, No. 2, pp.106–115.

Zhang, D., Li, J., Gao, Z., Wang, L. and Nan, J. (2016) 'Thermal performance investigation of modified flat plate solar collector with dual-function', *Applied Thermal Engineering*, Vol. 108, No. 9, pp.1126–1135.