SSRG International Journal of Civil Engineering
ISSN: 2348-8352/ https://doi.org/10.14445/23488352/IJCE-V13I1P111

Volume 13 Issue 1, 132-139, January 2026

Original Article

Comparative CFD Analysis of Cylindrical, Oval, and Finned Tubes for Thermosiphons

Heydi Karina Hinostroza Maravi[1], Nelfa Estrella Ayuque Almidon[2], Aron Jhonatan Aliaga Contreras[2],
Jean Fernando Perez Montesinos[3]

[1]*Faculty of Engineering, Universidad Continental, Huancayo, Junín, Peru.*
[2]*Faculty of Engineering, Universidad Nacional del Centro del Peru, Huancayo, Junín, Peru.*
[3]*School of Civil Engineering, University of Idaho, Idaho, USA.*

[1]*Corresponding Author : 70345903@continental.edu.pe*

Received: 07 November 2025 Revised: 09 December 2025 Accepted: 08 January 2026 Published: 14 January 2026

Abstract - *The purpose of the present study is to evaluate the geometric impact on the thermal performance of solar thermosiphon tubes through the use of parametric modeling and CFD simulation. To achieve this purpose, three configurations were designed: cylindrical, oval, and with longitudinal fins, and the constant internal volume was maintained to ensure comparable conditions. The models in this paper were developed in the Autodesk Inventor software and then integrated into Autodesk CFD in conditions of average solar irradiance (850 W/m²) and laminar internal flow. The key variables of these results, such as maximum temperature, total heat flow, pressure drop, and internal thermal distribution, were analyzed. The outcomes of the performed analysis show that the finned model is the one with the highest thermal efficiency due to the increased exchange surface area, while the oval design presents an offer with a more homogeneous distribution that does not affect hydraulic behavior significantly. In addition, the cylindrical configuration, although less thermally efficient, permits maintaining a more stable flow profile, and this feature is relevant in passive, pump-less applications. This work presents a technical alternative for residential buildings that has the characteristic of being a low-cost and replicable one, and this viable option combines structural design, thermal analysis, and architectural functionality in areas with a high rate of solar radiation.*

Keywords - *CFD simulation, Thermosiphon systems, Absorber tube design, Solar thermal performance, Parametric modeling.*

1. Introduction

E Domestic Hot Water heating accounts for up to 30% of energy consumption in residential buildings globally, constituting a significant challenge for the design of efficient and sustainable buildings from a civil engineering perspective [1, 2]. In response to this demand, passive thermosiphon solar systems have proven to be a viable alternative, as they reduce reliance on fossil fuels and eliminate the need for mechanical components by utilizing the natural circulation of the thermal fluid driven by density differences [3, 4]. These systems operate via a buoyancy-driven thermal loop generated by the thermal gradient between a heat collection area and a storage tank [5]. Their simplicity and reliability make them suitable for residential applications; however, most current implementations use flat-plate or evacuated tube collectors, which present thermal and structural limitations, especially in low-budget contexts or under variable climatic conditions [6].

The use of parabolic trough concentrators is emerging as a high-performance alternative, capable of intensifying incident solar radiation on the absorber and achieving temperature increases of up to 70% compared to flat-plate configurations [7]. Nevertheless, their integration into passive thermosiphon systems designed for housing remains limited, particularly from a technical approach that considers the structural, thermal, and constructional aspects relevant to civil engineering [5].

Research in Ethiopia involved the design and CFD simulation of a solar water heater with a flat-plate collector. Temperature, pressure, and velocity profiles were evaluated under different mesh conditions and flow rates using ANSYS Fluent and the Discrete Transfer Radiation Model. Although their focus was on flat-plate collectors, their CFD methodology is applicable to the analysis of more efficient systems, such as thermosiphons with parabolic reflectors, which allow for greater energy concentration without the need for pumping [8]. In a complementary study, a CFD analysis of a Parabolic Trough Collector was conducted to evaluate its thermal efficiency under various flow conditions and collector lengths. Using ANSYS, they simulated the thermal behavior of the system with a glass cover, observing an 8% increase in efficiency during peak solar radiation. The study demonstrated that using a cover improves thermal stability and reduces convective and radiative losses, and its results were validated against a theoretical mathematical model [9].

Furthermore, another investigation performed a numerical optimization of a parabolic collector, considering the absorber geometry, aperture angle, and properties of the working fluid. It concluded that the geometric design and the appropriate choice of fluid are key factors in enhancing the system's thermal performance [10].

Diverse recent studies have demonstrated that parameters, like the system's inclination, the material of the conductive tube, and the geometry of the reflector, significantly impact the thermal performance and flow stability [11, 12]. Nevertheless, there are very few pieces of work that analyze these factors at the same time through the employment of digital tools like CAD and CFD, and, what is more important, none of them do a comparative evaluation of how the external geometries of the absorbing tube influence the thermal behavior under natural flow [13]. The vast majority of studies only focus on conventional cylindrical tubes or on specific internal modifications without considering geometric alternatives that can be implemented with no increment in the construction complexity. In this sense, the novelty of this study lies in comparing the three configurations: cylindrical, oval, and with longitudinal fins, while maintaining an internal equivalent volume and homogeneous conditions of operation. All these allow us to isolate the geometrical effect and provide technical evidence that has not been reported in previous research.

In this study, a solar thermosiphon system with a parabolic reflector for water heating in residential buildings is developed and analyzed. In addition, the use of copper was considered as the conductor material for this system, and the favorable structural inclination angles to natural flow were contemplated. A CAD environment was used to model the proposed system in this paper, and the evaluation was performed through CFD simulations. These activities allowed us to analyze the effects of the geometric, thermal, and structural variables on key performance indicators, which are the outlet temperature, flow velocity, heating time, and useful accumulated volume.

Through the integration of structural design and thermal modeling within the framework of civil engineering, this research proposes not only a replicable but also a low-cost technical solution for sustainable buildings in areas with a high rate of solar radiation. The results acquired through the comparative analysis provide a solid foundation that is applicable to architectural and building services design, aligning with global goals related to energy efficiency and sustainable development.

In spite of the fact that there is an increasing number of studies related to solar thermosiphon systems, most of the studies focus solely on absorbing tubes that have a cylindrical geometry, and do not analyze how alternative configurations can modify the thermal behavior under natural flow. Moreover, the integration of parabolic concentrators in residential thermosiphon systems has been explored mainly only at the conceptual level, with scarce CFD studies that permit the comparison of the different geometries of the tubes under equivalent operation conditions.

This absence of comparative analysis limits the technical comprehension that is needed for optimizing low-cost systems intended for domestic applications. It is in this context that this current research addresses this gap through the evaluation of cylindrical tubes, oval, and longitudinal finned ones that keep the internal equivalent volume, with the purpose of performing a systematic analysis of their hydraulic and thermal performance through the parametric model and the CFD simulation.

2. Literature Review

A great number of studies have already explored the design of solar thermosiphon systems that have parabolic collectors, as well as their evaluation through CFD simulation. The section of the literature review of this paper focuses on three critical areas, which are the passive operation of thermosiphon systems, the optical and thermal efficiency of parabolic concentration, and the application of CDF simulation as a tool to predict solar collectors.

2.1. Solar Thermosiphon Systems

These solar thermosiphon systems are a passive water heating technology that works using the natural circulation of the fluid because the density of the liquid changes when it absorbs solar energy, creating density differences that allow this technology to work. This is the thermosiphon principle, which allows fluid movement from the collector to the tank without the need for pumps or other tools. This principle triggers two key benefits: first, it simplifies the system; and second, it reduces its energy consumption. These are the two features that make this system ideal for domestic applications.

It is in these kinds of systems that different parameters influence directly, such as the collector inclination, pipe diameter, and thermal insulation. Another important factor that was observed is the following: a good thermal stratification within the tank helps conserve the energy overnight [14].

Another thing to mention is that these systems are particularly suitable for regions in which there is a high solar radiation and consistent hot water needs. Due to the characteristics of these systems, they offer an accessible, efficient, and low-maintenance alternative.

2.2. Parabolic Solar Collectors

Parabolic concentrators are optical tools that do not form an image of the light source because they use the reflection to concentrate both direct and diffuse radiation onto an absorber tube. These devices are specifically designed to operate without solar tracking, and this represents an advantage because it reduces their mechanical and operational complexity.

There are two elements that have a direct impact on the optical performance of the parabolic concentrator: the acceptance angle and the quality of the reflector [15]. Furthermore, this type of collector not only improves the heat transfer but also allows for obtaining a more homogeneous thermal distribution, and this reduces energy losses along the collector [14].

2.3. CFD Simulation in Solar Thermal Systems

This type of simulation (the CFD one) allows us to model the behavior of the fluid and the thermal distribution within the solar collector. This type of technique, that is a numerical one, is critical for predicting system efficiency because it allows optimizing the internal design and reducing the number of physical prototypes required. Research has employed CFD to simulate a parabolic trough collector, obtaining results that show a good correlation with experimental data and highlighting the impact of mass flow rate on thermal efficiency [9].

Likewise, Korres and Tzivanidis combined CFD simulations with ray-tracing techniques to evaluate the thermal performance of a Compound Parabolic Collector, succeeding in reducing the fluid temperature prediction error to less than 4%. This highlights the accuracy of the hybrid approach in the thermal characterization of solar systems [16]. Unlike the already mentioned studies that concentrate on just one type of absorbing tube or in specific operation conditions, this paper incorporates a direct comparison among three external geometries under homogeneous irradiance, operation, flow, and fluid properties. This approximation permits the evaluation, in a more precise manner, of the impact of the shape of the tube on the heat transfer, which is an aspect that has not been studied in any of the previous studies.

In summary, although the current literature contributes to important advancements in the thermal analysis of thermosiphon systems and parabolic collectors, most of the studies concentrate on conventional cylindrical settings or on internal modifications of the absorbing tube. In addition, the researchers who employ CFD usually evaluate just one type of geometry or specific conditions that limit the possibility of performing direct comparisons among alternative configurations.

Finally, research that keeps an equivalent internal volume among the models in order to isolate the geometrical effect on the hydraulic and thermal behavior was not identified. Thus, these limitations put in evidence the necessity of a broader comparative and systematic analysis, like the one developed in this paper.

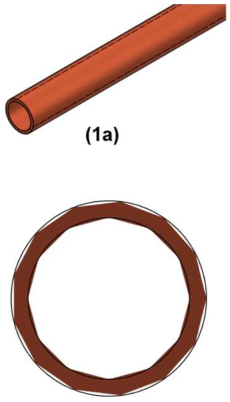
3. Methodology

3.1. Geometric Design of the Models

To evaluate the thermal performance of different thermosiphon tube configurations, Table 1 details the three geometric models parametrically modeled in Autodesk Inventor. An equivalent internal volume was maintained across all models to ensure a fair comparison under similar thermal conditions.

Table 1. Geometric models evaluated for CFD simulation

Modelo	Length (mm)	Outer Diameter (mm)	Inner Diameter (mm)	Additional Features
Cylindrical tube	700	30	34	Standard configuration
Oval tube	700	50 x 30 (major x minor axis)	—	Larger lateral surface area
Finned tube	700	30	34	4 longitudinal fins of 10 mm


(1a)

(1b)

Fig. 1(a) Isometric view of the standard cylindrical tube, and (b) Cross-section of the same model.

Figure 1 shows the conventional cylindrical model, used as a baseline reference for comparative evaluation. The isometric view (a) shows the simple geometry with a length of 1 m and an outer diameter of 30 mm*. The cross-sectional view (b) exhibits a circular section that is uniform; this section simplifies the analysis of the thermal distribution under standard flow conditions.

Figure 2 unveils the design with an oval geometry. The isometric view (a) reveals an elongation along the horizontal axis, while the cross-sectional view (b) shows an elliptical section with a major axis of 50 mm and a minor axis of 30 mm. The reason for this variation is that it allows for an increase in the contact surface area in strategic zones, so the heat transfer gets improved.

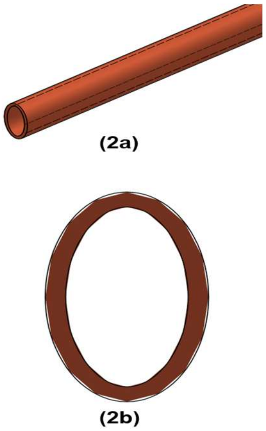


Fig. 2 (a)Isometric view of the oval tube, and (b)Cross-section of the same model.

Figure 3 shows the model with radial fins that are distributed symmetrically on the tube's periphery. In the case of the isometric view (a), the longitudinally extended fins are visible; these fins are designed to increase the area that is exposed to the solar radiation. The cross-section (b) allows for the analysis of the geometric arrangement of the fins; this arrangement promotes the dissipation of heat while also favoring the circulation of the fluid in contact.

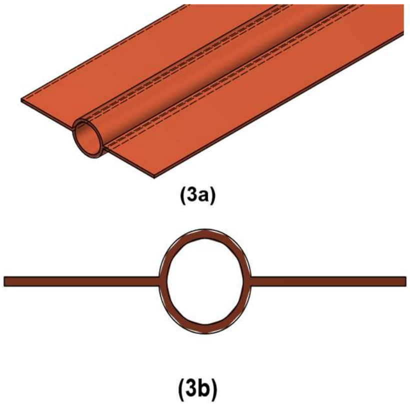


Fig. 3 (a)Vista isométrica del tubo con aletas, and (b)Sección transversal del mismo modelo.

The design of the three models was based on geometrical criteria employed in previous research that are related to thermal optimization, and this allows us to guarantee that the evaluated settings are technically viable and comparable. Jebbar et al. [17]; Wang et al. [18] demonstrated that geometric modifications, such as cylindrical inserts, improve thermal efficiency.

3.2. Study Parameters and Conditions

The numerical simulations were performed using Autodesk CFD. These simulations were carried out under internal flow conditions along with conjugate heat transfer. In this way, the thermal behavior of the solar thermosiphon system was represented. The parameters and conditions used in the simulations were the following:

Table 2. Parameters and conditions of the thermal model

Parameter	Value	Unit	Comment
Considered solar irradiance	850	W/m²	Average solar condition for collectors
Fluid velocity	0.125	m/s	Moderate laminar regime
Inlet temperature	25	°C	Cold water as a working fluid
Working fluid	Water	—	Standard properties: Cp=4180, ρ=997 kg/m ³
Inlet pressure	101325	Pa	Atmospheric pressure
Analysis type	Steady-state	—	Thermal evaluation under fixed conditions

The numerical analysis was performed while considering the interaction between the metallic walls of the tube and the fluid, which permits us to represent how the heat is transferred from the exposed Surface to the sun to the water in motion.

The operating conditions, like the solar irradiance, the fluid speed, and the inlet temperature, were chosen according to the typical values that were reported in previous experimental studies of thermosiphon systems. This ensures that the results that are obtained in this paper reflect a realistic behavior under residential conditions.

All parameters and conditions, such as solar irradiance: 850 W/m² laminar regime, are consistent with standard studies on solar collectors, such as those published by Jebbar et al. [17], who used similar parameters to evaluate the thermal performance in receiver tubes. Furthermore, the velocity of the fluid of 0.125 m/s corresponds to a laminar regime, and this result is consistent with different experimental studies of solar thermosiphons [5].

With the purpose to guarantee the reproducibility of this study, all the simulations followed a standardized procedure: (1) direct import of the geometry from Autodesk inventor in order to avoid dimensional inconsistencies, (2) definition of the thermal properties of the cupper and the water, according to values that were reported in the literature, (3) application of boundary conditions based on experimental thermosiphon system studies [5, 17], (4) the configuration of the laminar regime according to the obtained Reynolds number, and (5) the implementation of the analysis in a stationary state until reaching flow and thermal convergence.

The choice of the Autodesk CFD software is justified thanks to its native integration with parametric models and its capacity to solve the transfer of conjugated heat, which is consistent with employed methodologies in similar studies.

3.3. Meshing and Computational Setup

To adequately represent the thermal and fluid behavior of each model, a computational mesh adapted to the geometry of each tube was developed. The mesh divides the domain into small cells where the equations of heat transfer and movement of the fluids are solved. It is in the zones where major temperature and velocity variations happen, like the internal walls, the fins, and the curves, where an additional refinement was applied with the purpose of improving the precision of the calculus.

Different refinement levels were evaluated in order to ensure that the results were not dependent on the size of the mesh. Once the variations among the models were minimal, the final discretization was chosen for each geometry. This procedure guarantees that the observed differences among the tubes are due to their shape, and not the numerical effects. Figures 4, 5, and 6 depict the generated meshes for the cylindrical, oval, and fin models, respectively.

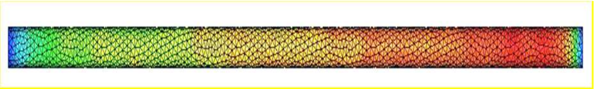


Fig. 4 CFD mesh of the cylindrical tube model, showing a regular unstructured discretization adapted to the simple tubular geometry

Fig. 5 CFD mesh of the oval tube model, with localized refinement in curvature zones to capture thermal gradients more accurately

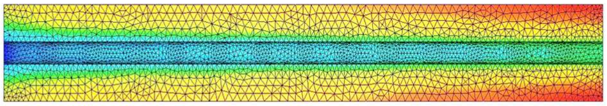


Fig. 6 CFD mesh of the finned tube model, including densified elements around the fins to improve resolution in heat transfer regions

3.4. Analysis Parameters

The CFD analysis performed in this study focused on the evaluation of the following key thermal and fluid-dynamic performance variables, and of course, all of them will be presented in the results section:

- Maximum temperature on the inner surface
- Total heat flow (W)
- Pressure drop between the inlet and outlet
- Mean fluid velocity
- Reynolds number
- Global thermal distribution
- Heat transfer coefficient

All these already shown variables allow for performing accurate technical comparisons between the three models; these comparisons are related to the criteria of energy efficiency, hydraulic behavior, and their applicability in solar thermosiphon systems.

The decision to choose the variables. Like the Reynolds number and the pressure fall, they are based on the metrics that are broadly standardized in CFD studies of thermosiphon systems because they permit evaluating, in a direct way, the glow regime and the hydraulic losses. This criterion is consistent with employed approaches in previous investigations, like the experimental analysis of natural convection performed by Huang et al. [5].

3.5. Justification of the Numerical Approach

Despite the fact that one of the most widely used CFD software packages is used in different studies, like the Jebbar et al. [17], this study decided to use the Autodesk CFD software for its integration with parametric designs in Inventor, and this represents a notable advantage for complex geometries. The decision to choose this software goes in line with the following considerations:

- Direct integration with designs created in Autodesk Inventor allows for geometric consistency without the need to import external meshes.
- The capability to solve conjugate heat transfer phenomena, which includes the conduction in solids and internal convection under a laminar regime.
- Computational feasibility to perform multiple comparative simulations without the need to incur high processing costs.

Furthermore, after performing the literature review, it could be noticed that the reviewed literature predominantly focuses on cylindrical models. Meanwhile, this study proposes a detailed geometric comparison that includes oval tubes and finned configurations. By making this comparison, this study expands the technical analysis of thermal efficiency in thermosiphon systems.

This methodological approach coincides with the utilized standards in comparative studies of geometries in solar collectors, where the geometric consistency, the explicit definition of the boundary conditions, and the verification of the convergence are fundamental requisites that guarantee the reproducibility of the results.

4. Results

In this section, all the results obtained from the CFD simulations of the three proposed models are presented. The three models that were analyzed are: cylindrical tube, oval tube, and finned tube. Different key variables had to be analyzed so we could establish not only thermal and fluid-dynamic comparisons between the different configurations, but also relevant patterns in the flow behavior could be identified.

4.1. Thermal and Dynamic Indicators

Table 3 summarizes the values obtained for each model; these values are: maximum internal temperature, total heat flow, pressure drop, mean fluid velocity, and Reynolds number. It is important to mention that these indicators allow for quantifying the geometric impact on the thermal efficiency of the thermosiphon system, and they also permit evaluating the hydraulic stability in residential conditions.

Table 3. Thermal and Dynamic performance indicators

Indicator	Cylindrical tube	Oval tube	Finned tube
Maximum temperature (°C)	58.2	60.9	65.4
Total heat flow (W)	22.5	24.1	27.8
Pressure drop (Pa)	98	110	145
Mean velocity (m/s)	0.125	0.121	0.119
Reynolds Number	480	470	460

Through the results, it is observed that the finned model is the one that reaches the highest internal temperature and total heat flow, and this is a consequence of the increase in the thermal exchange surface area. On the other hand, when looking at the cylindrical model, it maintains a more stable hydraulic profile with a lower pressure loss, but it is less efficient in heat transfer; this feature makes it more favorable in contexts where natural circulation is critical.

4.2. Thermal Distribution and CFD Visualization

The simulations that were performed in this study demonstrate that geometric modifications are the ones that directly influence the temperature and flow behavior profiles; also, these geometric modifications generate variations in both thermal uniformity and transfer efficiency.

Figure 7 shows that, in the cylindrical model, the temperature tends to concentrate in the central zone of the tube, while more pronounced gradients appear towards the ends. This distribution indicates less uniform heat transfer and the presence of hotspots that can reduce the system's efficiency under natural circulation conditions.

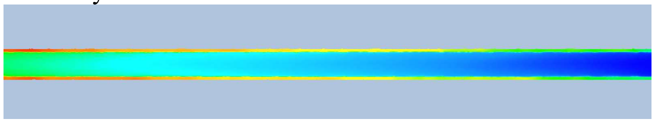


Fig. 7 Thermal distribution of the cylindrical tube model, highlighting central heat accumulation and pronounced gradients toward the ends

Figure 8 shows that the oval model exhibits a more uniform thermal distribution along the length of the tube. The larger lateral surface area promotes contact between the fluid and the internal walls, allowing for more homogeneous energy absorption and reducing the formation of high-temperature zones.

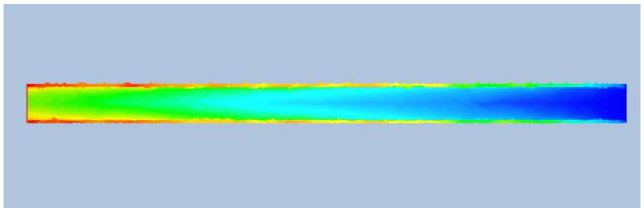


Fig. 8 Thermal distribution of the oval tube model, showing a more uniform temperature profile along the tube due to increased lateral surface area

Figure 9 shows that the finned model achieves greater thermal absorption. The extended surfaces increase the heat exchange area, generating higher temperatures and more efficient heat transfer compared to the other geometries evaluated.

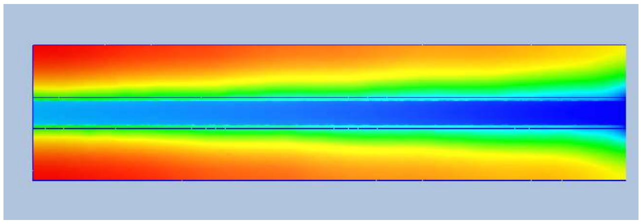


Fig. 9 Thermal distribution of the finned tube model, illustrating enhanced heat absorption and higher temperatures near the extended surfaces

4.3. Technical Interpretation of the Results

In contrast to the majority of findings:

- The finned model achieves the highest internal temperature and total heat flow, attributed to the increase in heat exchange surface area.
- The oval model slightly improves thermal performance compared to the cylindrical one, with a more homogeneous distribution, although its pressure drop is slightly higher.
- The cylindrical model maintains a more stable hydraulic profile with a lower pressure loss, but it is less thermally efficient. Thus, this model can be useful in contexts where pumping efficiency or natural circulation is critical.

Table 4. Comparative summary with previous studies

Study	Geometry	Key Findings	Comparison with Present Work
Huang et al. [5]	Cylindrical	Stable natural circulation; moderate heat gain	Consistent with cylindrical model behavior
Jebbar et al. [17]	Cylindrical with inserts	Improved heat transfer via internal modifications	The finned model achieves a similar improvement externally
Korres & Tzivanidis [14]	CPC with cylindrical tube	Higher uniformity with optimized geometry	The oval model shows similar uniformity improvement

Table 4 depicts a summary of the comparisons between the results obtained in this study and the findings reported in previous studies. This permits the contextualization of the performance of the evaluated geometries within the broader literature on thermosiphon collectors.

5. Discussion

All the results obtained in this study through the simulations show how the geometry has an impact on the thermal performance of the thermosiphon tubes. In particular, the model with radial fins is the one that presented the highest internal temperature (65.4 °C) and total heat flow (27.8 W). This thermal performance can be attributed to two things: first, the increase in surface area, and second, the concentration of solar radiation. This finding is consistent with the work presented by Habeeb et al. [9]. In this study, they demonstrated, through CFD simulations, that different geometric modifications, such as the incorporation of heat transfer elements, can increase collector efficiency.

A complementary work is the study performed by Boukheit et al. [19], in which they analyzed smooth absorber tubes versus tubes with helical fins, and after the analysis, they concluded that the internal geometry can significantly increase the heat transfer coefficient. Despite the fact that the present work uses external longitudinal fins, it could be observed that the principle of enhancement by increasing surface area remains valid, and those results validate the design choice as an effective thermal strategy. Considering the oval model, the results showed that it is less efficient than the finned one, but it also revealed a more homogeneous thermal distribution and even higher temperatures than the cylindrical model. These results suggest that the tube's shape can influence not only the heat capture but also the stability of the internal flow, and this phenomenon was mentioned by Korres and Tzivanidis [14] in their studies on compound collectors. On the other hand, with regard to the cylindrical model, it presented a lower pressure drop, maintaining hydraulic stability. This behavior can be valuable in natural circulation systems because in those systems, the pressure losses affect the thermosiphon flow. The comparison performed among these three models reinforces the importance of simultaneously considering thermal efficiency and hydrodynamic behavior when designing passive solar systems.

6. Conclusion

Through the results obtained from the simulations, this study demonstrates that the geometry of the absorber tube in solar thermosiphon systems directly influences their thermal and fluid-dynamic performance. After the comparisons of three models, the configuration with external longitudinal fins is the one that achieved the best thermal performance, while the oval model provided greater homogeneity in the temperature distribution.

Through the CFD simulations carried out in Autodesk, it was validated that the increased surface area and an optimized structural design are the variables that can improve passive solar heating efficiency without the need for mechanical components. These findings go in line with the discoveries of previous studies that address internal fins and other alternative geometries [19].

As a result, this present research provides a replicable methodology for the comparative analysis of tubes in solar thermosiphons through the use of accessible tools like Autodesk Inventor and Autodesk CFD. These outcomes can be very useful to guide future implementations in residential buildings with high solar irradiance, since this aftermath represents low-cost, high-efficiency solutions with an easy architectural integration. For future studies related to this field, it is recommended to evaluate hybrid geometries, as well as alternative materials such as aluminum or conductive alloys, and even transient operating conditions, so the technical understanding of thermosiphon behavior in real-world scenarios can be expanded.

In spite of obtaining positive results, this study presents certain limitations. For instance, the simulations were executed in conditions of a stationary state and with constant thermal conditions, which do not allow for either to capture the transitory behavior or the dependent variations of temperature. Also, the value of the solar irradiation and the inlet velocity were the variables evaluated, which limits the generalization of the results into other operative scenarios. Nonetheless, the outcomes obtained have relevant implications for the design of low-cost thermosiphon systems because they demonstrate that the external geometric modifications can improve the thermal performance without increasing the complexity of the construction. Thus, Future investigations should explore transitory simulations, alternative conducting materials, hybrid geometries, and experimental validation in order to broaden the technical comprehension of the thermosiphon behavior under real conditions.

References

[1] Global Energy and Climate Model: Scenario Analysis of Future Energy Trends, International Energy Agency, 2025. [Online]. Available: https://www.iea.org/reports/global-energy-and-climate-model

[2] Tanveer Ahmad, and Dongdong Zhang, “A Critical Review of Comparative Global Historical Energy Consumption and Future Demand: The Story Told So Far,” *Energy Reports*, vol. 6, pp. 1973-1991, 2020. [CrossRef] [Google Scholar] [Publisher Link]

[3] A. Jamar et al., “A Review of Water Heating System for Solar Energy Applications,” *International Communications in Heat and Mass Transfer*, vol. 76, pp. 178-187, 2016. [CrossRef] [Google Scholar] [Publisher Link]

[4] Ruslan Botpaev et al., “Drainback Solar Thermal Systems: A Review,” *Solar Energy*, vol. 128, pp. 41-60, 2016. [CrossRef] [Google Scholar] [Publisher Link]

[5] C.S. Huang et al., “Experimental Observation of Natural Convection Heat Transfer Performance of a Rectangular Thermosyphon,” *Energies*, vol. 12, no. 9, pp. 1-12, 2019. [CrossRef] [Google Scholar] [Publisher Link]

[6] Neeraj Mehla, and Amit Kuma, “Experimental Evaluation of used Engine Oil based Thermal Energy Storage Coupled with Novel Evacuated Tube Solar Air Collector (NETAC),” *Journal of Energy Storage*, vol. 39, 2021. [CrossRef] [Google Scholar] [Publisher Link]

[7] Mengyan Cao, Xiujian Zhao, and Xiao Gong, “Achieving High-Efficiency Large-Area Luminescent Solar Concentrators,” *JACS Au*, vol. 3, no. 1, pp. 25-35, 2023. [CrossRef] [Google Scholar] [Publisher Link]

[8] Lemi Negera Woyessa et al., “Design and CFD Simulation of Solar Water Heater used in Solar Assisted Biogas System,” *International Journal of Innovative Technology and Exploring Engineering (IJITEE)*, vol. 9, no. 3, pp. 371-380, 2020. [CrossRef] [Google Scholar] [Publisher Link]

[9] Monaem Elmnifi et al., “Computational Fluid Dynamics Analysis of a Parabolic Trough Solar Collector for Enhanced Efficiency and Thermal Performance,” *Mathematical Modelling of Engineering Problems (MMEP)*, vol. 10, no. 3, pp. 1039-1046, 2023. [CrossRef] [Google Scholar] [Publisher Link]

[10] Behzad Shaker et al., “CFD Analysis of Al_2O_3-Syltherm Oil Nanofluid on Parabolic Trough Solar Collector with a new Flange-Shaped Turbulator Model,” *Theoretical and Applied Mechanics Letters*, vol. 12, no. 2, pp. 1-13, 2022. [CrossRef] [Google Scholar] [Publisher Link]

[11] Soteris A. Kalogirou, “Solar Thermal Collectors and Applications,” *Progress in Energy and Combustion Science*, vol. 30, no. 3, pp. 231-295, 2004. [CrossRef] [Google Scholar] [Publisher Link]

[12] Mondher Bouteraa, and Chérif Nouar, “Rayleigh-Bénard Convection in Non-Newtonian Carreau Fluids with Arbitrary Conducting Boundaries,” *International Communications in Heat and Mass Transfer*, vol. 76, pp. 77-84, 2016. [CrossRef] [Google Scholar] [Publisher Link]

[13] Lei Li, Carlos F. Lange, and Yongsheng Ma, “Association of Design and Computational Fluid Dynamics Simulation Intent in Flow Control Product Optimization,” *Proceedings of the Institution of Mechanical Engineers, Part B: Journal of Engineering Manufacture*, vol. 232, no. 13, pp. 2309-2322, 2018. [CrossRef] [Google Scholar] [Publisher Link]

[14] Dimitrios N. Korres, and Christos N. Tzivanidis, “Numerical Investigation and Optimization of an Experimentally Analyzed Solar CPC,” *Energy*, vol. 172, pp. 57-67, 2019. [CrossRef] [Google Scholar] [Publisher Link]

[15] Chuan Jiang et al., “A Review of the Compound Parabolic Concentrator (CPC) with a Tubular Absorber,” *Energies*, vol. 13, no. 3, pp. 1-31, 2020. [CrossRef] [Google Scholar] [Publisher Link]

[16] Javier E. Barrón-Díaz et al., “FEM-CFD Simulation and Experimental Study of Compound Parabolic Concentrator (CPC) Solar Collectors with and without Fins for Residential Applications,” *Applied Sciences*, vol. 11, no. 8, pp. 1-19, 2021. [CrossRef] [Google Scholar] [Publisher Link]

[17] Yasser Abidnoor Jebbar et al., “Improvement of Heat Transfer in a Parabolic Trough Collector Receiver Tube with Hollow Cylindrical Inserts: A CFD Study,” *International Journal of Heat and Technology*, vol. 42, no. 3, pp. 924-932, 2024. [CrossRef] [Google Scholar] [Publisher Link]

[18] Yinfeng Wang et al., “CFD Simulation of an Intermediate Temperature, Two-Phase Loop Thermosiphon for use as a Linear Solar Receiver,” *Applied Energy*, vol. 207, pp. 36-44, 2017. [CrossRef] [Google Scholar] [Publisher Link]

[19] Gulom Uzakov, and Sokhiba Shamurotova, “CFD Analysis for Absorber Tube of a Solar Water Heater Collector,” *E3S Web Conference*, vol. 498, pp. 1-8, 2024. [CrossRef] [Google Scholar] [Publisher Link]