https://doi.org/10.48047/AFJBS.6.Si4.2024.5592-5596

African Journal of Biological Sciences

Journal homepage: http://www.afjbs.com

Research Paper Open Access

Thermal Efficiency and Heat Transfer Analysis of Barrel vs. Non-Cylindrical Cavities in Solar Thermal Applications

[1]Avinash Kumar , [2] Dr. Manish Gangil

Research Scholar, Department of Mechanical Engineering,RKDF University, Bhopal
Professor, Department of Mechanical Engineering,RKDF University, Bhopal
[1]eravianil@gmail.com, [2]rkdfbhojpal@gmail.com

Volume 6, Issue Si4, Aug 2024

Received: 15 June 2024

Accepted: 25 July 2024

Published: 15 Aug 2024

doi: 10.48047/AFJBS.6.Si4.2024.5592-5596

Abstract- This study aims to probe into the effects of barrel cavity designs, among other non-cylindrical types, on thermal efficiency and heat transfer performance regarding solar thermal applications. In this study, three-dimensional Computational Fluid Dynamics analysis will be done using Fluent Software, Version 6.2.16. Two different cavity geometries are going to be simulated: one with a Barrel and the other with an Hourglass shape. It estimates thermal performance parameters like outlet temperature, coil temperature, thermal efficiency, and friction factor by detailed simulation, showing the influence of mesh size on the accuracy of results. In comparison, the barrel shape revealed better thermal efficiency and heat transfer performance than that of a non-cylindrical hourglass-shaped design. Mesh independence testing verified that a mesh size of about 533,912 cells produced stable and accurate solutions, and validation against theoretical and experimental data returned deviations within acceptable windows. In summary, the barrel cavity design is more efficient in the conversion of solar energy to thermal energy at improved heat transfer and flow efficiency. The findings are, therefore, of importance in optimizing the performance of solar-thermal systems and in the improvement of high-performance heat exchanger design.

Keywords- Computational Fluid Dynamics (CFD), Barrel Cavity Design, Heat Exchanger, Nusselt Number

1. Introduction

Heat exchangers (HEX) are devices for efficient heat transfer from one medium to another, which means they are an important component for successful heat removal from the source. These instruments are widely used in space heating, refrigeration, and air conditioning, as well as power plants, chemical plants, petrochemical plants, petroleum refineries, natural gas refining, sewage treatment, combustion engines, naval and aviation/aerospace applications, and so on (Mohammed Hussein et al. 2022). Heat exchanger is an important component in many industrial fields, such as chemical, petroleum, power, food and so on, in which it assumes the function of transferring heat between two or more supplies of different temperatures. Conventional heat exchangers include shell and tube heat exchangers, plate and fin heat exchangers, compact heat exchangers such as printed circuit heat exchangers, etc. (Yan, K et al. 2023). Advancements in heat exchanger technology have been directed toward increasing efficiency, reducing size, and improving durability of materials. Some of the innovations have involved compact heat exchangers with an increased surface area-to-volume ratio that will enable efficient heat transfer in smaller spaces. Improved materials such as graphene and nanofluids give better thermal conductivity and corrosion resistance (Patel, A. 2023). Moreover, computational fluid dynamics has been advanced to perfect the design of heat exchangers to ensure optimum performance, which means more efficient temperature control and less energy is wasted on both industrial and environmental levels. These technologies improve thermal performance and the development of sustainable solutions toward the establishment of an energy-efficient route for modern engineering challenges (Wang, C et al. 2023).

2. Related Work

The review by (Mahmoudinezhad et al., 2023) addresses high-temperature heat exchangers, indicating the need for concentrated efforts on special designs and materials above the 600 °C barrier. The fluids and configurations are highlighted along with the key materials, while also noting the challenges such as corrosion and material degradation. (Cui et al., 2024) provide a critical review of the development in Ground Heat Exchangers for use in Ground Source Heat Pumping systems in regard to the impact of local geology and groundwater flow on GHE efficiency and discuss various models that have been developed, both analytical and numerical, in this context. (Togun et al., 2024) focus on Latent Heat Thermal Energy Storage systems, which use phase change materials for efficient thermal energy storage. The authors describe the challenges in the selection of PCMs and the potential of hybridization techniques to improve their performance. (Careri et al., 2023) deal with Additive Manufacturing methods to make complex-shaped, lightweight, and leak-proof heat exchangers for aerospace applications. The study provides the status of the current AM technologies—including Laser Powder Bed Fusion—and discusses problems related to the fabrication process of high-efficiency compact heat exchangers.

3. Proposed Approach

A detailed approach to the study, which is backed up by three-dimensional Computational Fluid Dynamics analysis when evaluating several designs of cavity receivers for solar dish systems using Fluent Software, Version 6.2.16. CFD is applied as one of the very strong numerical tools now available in carrying out thermal and fluid dynamic studies of this nature. It is most suitable for use in analysing complex systems like solar dish receivers. Based on the main stages, this methodology is designed: model description, mesh generation, boundary conditions, material properties, assumptions, mathematical formulations, and numerical modelling and simulation; therefore, the evaluation of the designs is completed.

It is necessary to mention that, within the present study, two specific designs of cavity receivers are selected: the Barrel and Hourglass shapes. The geometries of the given designs are inbuilt with some particular geometrical features. In furtherance, the mentioned geometries are modelled in CAD, after which the mesh is generated within the ANSYS environment. The design of the Barrel

Cavity with helical grooves shall maximize the absorption and retention of solar energy, and the retaining cavity's shape itself is bound to have an effect. The design of the Hourglass Shape retaining cavity, on the other hand, shall maximize the exposure of the surface area to the concentrated solar radiation for optimum flow of the heat transfer fluids. In the mesh generation, structured grids of hexahedral volume elements are used, and essential boundary conditions are applied by solving the governing conservation equations using the Finite Volume Method (FVM) for an appropriate simulation. The study adopts advanced mathematical formulations for performance characterizations of the designs with the major equations that quantify thermal efficiency and fluid dynamics.

The numerical modelling and simulation adopts the k-ε model for characterizing the fluid flow and heat transfer in the cavities with the RNG variant. This allows the two cavity designs, i.e., Barrel and Hourglass, to be under a detailed examination of their thermal behaviour and flow physics under various conditions. The systematic creation of geometrical models, meshing, selection of appropriate solver formulations, and assigning material properties and boundary conditions are performed so that any useful conclusion can be drawn on how effectively the cavities are designed with the idea of efficient conversion of solar energy into productive thermal energy. Ultimately, the CFD-based approach targets the optimization of solar dish systems for perfecting their thermal efficiency, hence their contribution to the real development of technologies for the conversion of solar energy.

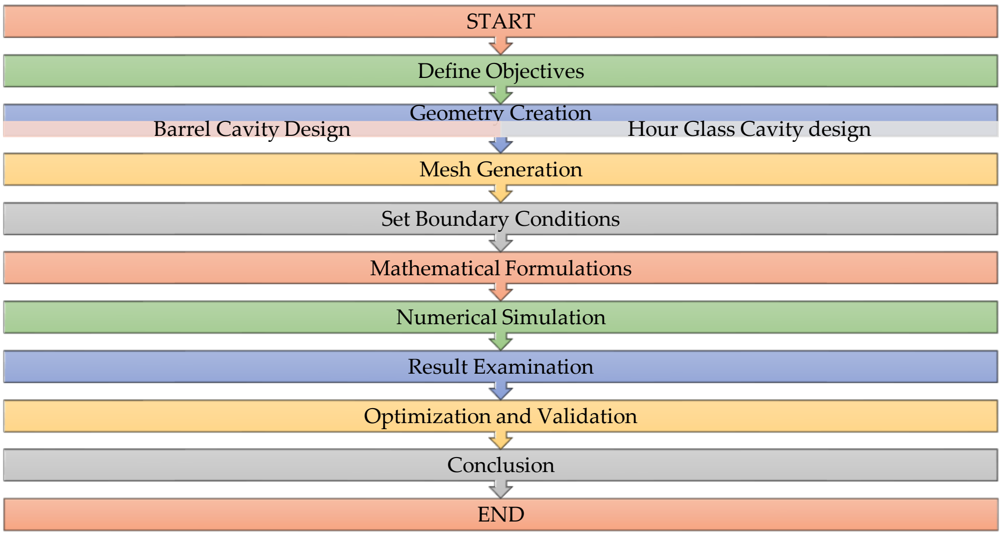


Fig.1 Flowchart of proposed approach

Such proportions in choice of design towards Barrel and Hourglass shapes have been made by keeping the relationship in mind among: the surface area, energy absorption, and heat retainment. The reasons are that the greater surface area, the larger the contact with the energy source, that eventually absorbs into the matrix, and is directly related to greater heat retainment in the matrix. Mesh quality is very crucial to accuracy in simulation, for it directly influences the level of precision for any given set of simulation outputs. A high-quality mesh guarantees credible metrics and precisely defined simulations that ensure valid and acceptable results from the analysis.

The boundary conditions directly affect the amount and kind of effect the conditions have on fluid flow and heat transfer within cavity receivers, and this significantly affects the accuracy in simulation. Well-defined boundary conditions will ensure that the simulation represents realistic reality of the physical phenomenon, therefore giving an accurate prediction of the performance of the designed system. RNG can be used in a k-ε model, and the model is one of the models generally used in the simulation of turbulent flow mainly because of the ability of the model to work out a fair approximation for the turbulence the fluid flow is characteristic for. A critical step in verification, ensuring the stabilization of a CFD simulation, is achieving convergence in quantities of interest.

Finally, the results are validated and optimized through comparison of achieved simulation results with the help of theoretical or experimental data in order to fix the iteration with the best fit. The optimization is then performed in an iterated, refinable method to produce the performance and design into levels of efficiency. The step is crucial because the final design reached through optimization will fulfil the designated criteria and purposes fully.

4. Result Analysis

In general, grid independence testing is an important exercise in numerical simulations for the assurance of accuracy and the reliability of computational results. The sensitivity of the results from simulations taken with increasing accuracy of the mesh is actually focused on during this testing procedure. Such a test usually enhances resolution from a coarse mesh to higher resolutions and checks how the results are stabilized when the grid becomes finer. Such an approach helps to choose a mesh size that provides a delicate balance between the computational accuracy and efficiency.

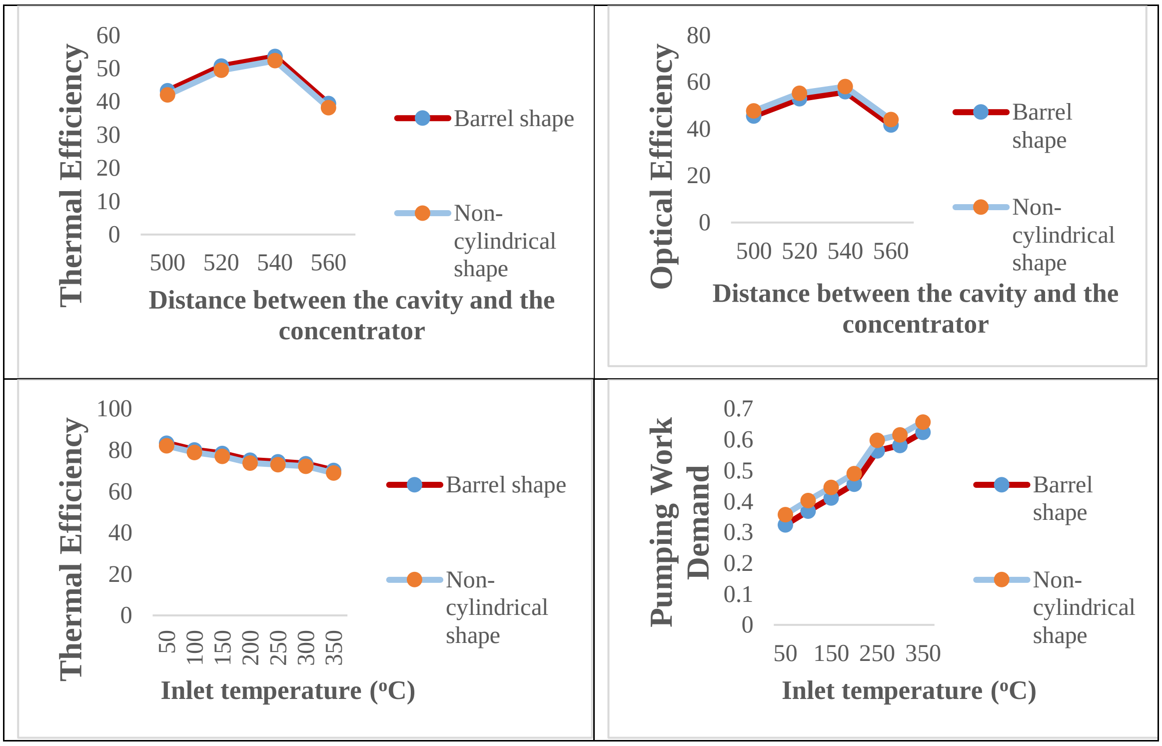
For the purposes of a study to analyze radial bearing stiffness by computational simulation, grid independence has been checked through the use of different mesh sizes. All results were summarized in one table, summarizing how the main parameters (T_{out} , T_{coil}, $\eta_{th}$and f_r) change as the mesh resolution increases. Initially, the Tout temperature shows the increment of the number of cells from 72,342 to 214,226, reaching a temperature of 28.7°C. However, after 214,226 cells, the outlet temperature did not deviate much from 28.6°C, indicating that beyond this number, further refining does not change much. Meanwhile, the coil temperature drops from 28.4°C to 27.3°C with mesh refinement but thereafter almost stays at around 27.5°C. The thermal efficiency increased from 51.3% to 53.4% on mesh increasing and then kept between 53.4% and 53.6%. The friction factor decreased drastically from 0.69 to 0.13 with increased mesh refinement. Further studies let out that a mesh size of about 533,912 cells was capable of producing acceptable and stable results since more refinement did not significantly change the output parameters of interest but consumed more computational resources.

The increases in temperature, Nusselt number, and friction factor in the validation run have been compared. The comparison showed that the results of the present study generally agreed well with the experimental, theoretical data with deviation of 2.82-12.00% for increase in temperature, 0.57-7.3% with Nusselt numbers and friction factors, respectively. The validation study proved that the model development is accurate, and reliable results are yielded for the thermal and flow analyses.

The influence of the cavity shape on the heat transfer performance was also studied, for which a barrel and a non-cylindrical geometry were considered. It was shown that the barrel shape approached better performance than the non-cylindrical one in terms of the Nusselt number and thermal efficiency. In the larger aspect, it maintained a higher Nusselt number besides having lower friction factors in all volumetric flow rates compared to the non-cylindrical geometry. That is, the barrel shape gave a better heat transfer with flow efficiency.

The impact of cavity length and distance between the cavity and concentrator on thermal and optical efficiency was then addressed with further analysis. The results indicated that, for all cases of cavity length and distance, the barrel shape displayed the best thermal and optical efficiencies among all non-cylindrical shapes. For example, in different cavity lengths, the barrel shape pointed to marginally higher thermal efficiency, while in optical efficiency, the barrel shape showed better performance in the tested differences in distance.

Therefore, this study completes an assessment of mesh independence, model validation, and the influence of geometry on the heat transfer performance. The results underline that the mesh size should be appropriate for the accurate simulation, whereas the computational model should be verified against established data for its reliability. A comparative analysis of various geometries and operating parameters indicates that the barrel shape is much more efficient in terms of heat transfer efficiency and therefore will be a preferential choice in optimization of performance for similar applications.



5. Conclusion

This study represents an overall assessment of solar thermal applications on barrel and non-cylindrical cavity designs, with a keen eye for the geometry that drastically affects heat transfer performance. The study found that through comprehensive CFD simulations, the barrel-shaped cavity far outperformed its counterpart in thermal efficiency and heat transfer effectiveness, which was of non-cylindrical hourglass design. It was suc-cess-fully dem-on-strated that, in com-par-i-son with the bare tube, the barrel design showed an improved heat transfer per-for-mance with higher Nusselt num-bers and lower friction fac-tors under var-i-ous volu-met-ric flow rates, mean-ing this design is asso-ci-ated with bet-ter heat transfer and flow efficiency.

Mesh independence testing showed that a mesh resolution of roughly 533,912 cells was optimal for balancing accuracy and computational efficiency. The validation process of the study proved that results from the CFD model were reliable; deviations from theoretical and experimental values were acceptable. This establishes greater faith in the model's robustness and its applicability to real-life situations.

The obtained results confirm the role of proper cavity geometry in order to realize maximum enhancement in a solar thermal system. The barrel shape was preferred for high solar energy conversion because of its higher thermal and optical efficiency. This study contributes to knowledge in the design and development of more efficient solar thermal systems and heat exchangers, moving the field of sustainable energy technologies further ahead.

References

[1] Mohammed Hussein, Hind Azeez, Rozli Zulkifli, Wan Mohd Faizal Bin Wan Mahmood, and Raheem K. Ajeel. "Structure parameters and designs and their impact on performance of different heat exchangers: A review." *Renewable and Sustainable Energy Reviews* 154 (2022): 111842. https://doi.org/10.1016/j.rser.2021.111842

[2] Yan, K., Wang, J., & Deng, H. (2023). Numerical investigation into thermo-hydraulic characteristics and mixing performance of triply periodic minimal surface-structured heat exchangers. *Applied Thermal Engineering*, *230*, 120748. https://doi.org/10.1016/j.applthermaleng.2023.120748

[3] Patel, A. (2023). Enhancing heat transfer efficiency in solar thermal systems using advanced heat exchangers. *Multidisciplinary International Journal of Research and Development (MIJRD)*, *2*(06), 31-51.

[4] Wang, C., Lu, Q., Liu, Y., Huang, H., & Sun, J. (2023). Progressive review of heat transfer enhancement technologies in 2010–2020. *Sustainable Energy Technologies and Assessments*, *56*, 103121. https://doi.org/10.1016/j.seta.2023.103121

[5] Mahmoudinezhad, S., Sadi, M., Ghiasirad, H., & Arabkoohsar, A. (2023). A comprehensive review on the current technologies and recent developments in high-temperature heat exchangers. *Renewable and Sustainable Energy Reviews*, *183*, 113467. https://doi.org/10.1016/j.rser.2023.113467

[6] Cui, P., Yang, W., Zhang, W., Zhu, K., Spitler, J. D., & Yu, M. (2024). Advances in ground heat exchangers for space heating and cooling: Review and perspectives. *Energy and Built Environment*, *5*(2), 255-269. https://doi.org/10.1016/j.enbenv.2022.10.002

[7] Togun, H., Sultan, H. S., Mohammed, H. I., Sadeq, A. M., Biswas, N., Hasan, H. A., ... & Talebizadehsardari, P. (2024). A critical review on phase change materials (PCM) based heat exchanger: different hybrid techniques for the enhancement. *Journal of Energy Storage*, *79*, 109840. https://doi.org/10.1016/j.est.2023.109840

[8] Careri, F., Khan, R. H., Todd, C., & Attallah, M. M. (2023). Additive manufacturing of heat exchangers in aerospace applications: a review. *Applied Thermal Engineering*, 121387. https://doi.org/10.1016/j.applthermaleng.2023.121387