International Journal of Advances in Engineering and Management (IJAEM)
Volume 5, Issue 8 Aug 2023, pp: 08-19 www.ijaem.net ISSN: 2395-5252

Optimising Thermal Performance of Solar Air-Water Heater using Computational Fluid Dynamics (CFD) Simulation

Bhavesh C Pathak[1]

[1]*Assistant Professor, Mechanical Engineering Department, Government Engineering College Bharuch, Gujarat, India*

Date of Submission: 01-08-2023 Date of Acceptance: 10-08-2023

ABSTRACT:The notion of thermal is associated with the concept of heat energy present in a system that is responsible for the alteration seen in its temperature. Such an aspect has been seen to be impacted by the total energy expenditure occurring in the elements, which is associated with the efficiency through which an item retains, or avoids the channelling of heat. It has been recorded that factor of thermal performance is heavily related to the concept of thermal conductivity of materials, which is also affected with the assemblage of the items. The interpretation of thermal performance has been shown to have a wide range of benefits that can be attributed to it, such as its contribution to the identification of energy-efficient materials as a result of its contribution to thermal performance interpretation. A number of aspects of the product's durability, aspects of compliance with the code, maintaining structural integrity at the molecular level, and the results of sustainability have also been associated with the impact of thermal performance on the final product. It will be proposed that a study on the thermal performance of Solar Air-Water Heaters in terms of Computational Fluid Dynamics or CFD Simulation will be carried out in an extensive manner in this study.

KEYWORDS: Computational Fluid Dynamics model, CFD, Solar Air-Water Heater, thermal performance, Navier–Stokes equations, fluid mechanics theory.

I. INTRODUCTION

The use of solar energy for heating has enormous potential, since it is clean, renewable and eco-friendly. The solar air-water heater optimizes the sun's energy absorption by using air and water as heat transfer mediums. In order to improve the efficiency of these systems, engineers and researchers are using computational fluid dynamics simulations(CFD)[1][3][5][9]. Thermal performance is designated as the factor which is associated with the retention of heat capacity of the objects. Solar Air-Water Heaters, also known as solar hot water system, has been a cost-efficient technology used for the generation of hot water in the areas of its application. A critical step in CFD modelling is validation, which entails contrasting model predictions against data from experiments or the real world. Engineers and researchers may assess the precision and dependability of the simulations by validating the CFD model against trustworthy data. This procedure identifies potential areas for calibration or improvement and boosts confidence in the model's forecasting skills.

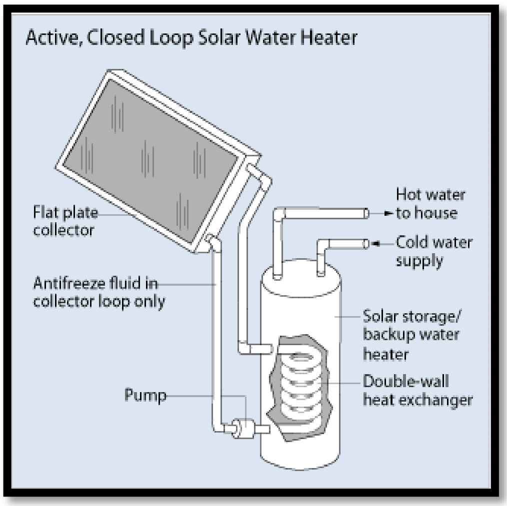

Figure 1: Solar water heating system [2]

Through the effective management of direct and indirect circulation systems, the heating devices have the capacity of maintaining its operational usage [1][11][12][29].

Thermal Performance of various kind of solar collector such as solar water or air heater and solar cooker are studied in [54-70] by varying various geometries of the absorber plate in the solar devices.

II. OBJECTIVES

The objectives for the study are developed as follows:

- To examine the notions of thermal performance of devices.
- To inspect the thermal performance seen in solar air-water heater.
- To understand the models of computational fluid dynamics.

III. METHODOLOGY

Thermal performance has been the basic approach taken into consideration in case of operations of Solar Air-Water Heaters [37] [38]. Solar air heaters have been specifically implemented in agricultural grounds for improving the overall heating of the space, the improvement of timber seasoning, for the drying of crops and vegetables, and similar others [6]. Several significant features have been observed to play major parts in the case of thermal performance of Solar Air-Water Heaters, such as that of louvered fins, their lengths, angles of integration and similar others. [36].

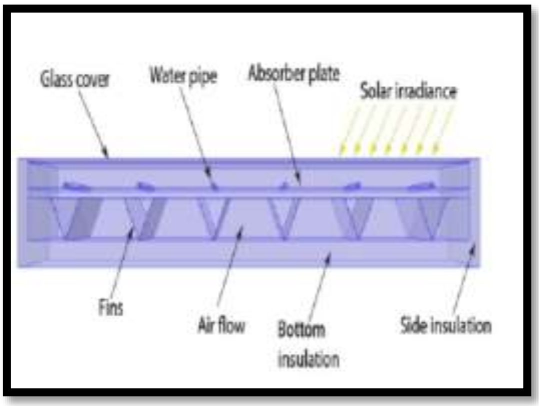

Figure 2: Modular nature of air and water heatercollector [8]

The way in which air flow is managed, and the way in which mass flow is managed has had an impact on how effective the devices are. A solar energy installation has been able to provide uninterrupted energy, which has a high degree of coherence in nature [7] [34]. There has been evidence, however, that the fluid solar collectors were able to provide hot water and warm air.The flexibility, efficiency, and scalability benefits of modular air and water heater collectors are substantial. These systems are made to capture the energy from renewable resources, such solar radiation, and transform it into useful heat for a variety of uses. In this article, we'll look at the advantages of modular design for both air and water heater collectors.Modularity makes it simple to customise and adapt to unique demands and requirements. Modular air and water heater collectors make it feasible to construct systems that are specifically suited to a structure or area. Individual modules, for instance, might be placed carefully in air heater collectors to maximise solar exposure and reduce shade. This adaptability guarantees maximum energy capture and efficiency, enhancing performance and reducing costs.Additionally, modularity makes installation and maintenance procedures simpler. The collectors can be transported, constructed, and dismantled quickly because they are made up of independent modules. When dealing with intricate installations or places with restricted access, this capability is especially helpful. Furthermore, since individual modules can be changed or repaired without causing a disruption to the overall system, maintenance becomes simpler. Modular air and water heater collectors are a sensible option for both residential and commercial applications since they have less downtime and associated costs.

IV. CONCEPT OF THERMALPERFORMANCE

Thermal performance has been seen to play a major part in a wide range of associations, such as the construction of electric vehicles, maintenance of thermal efficiency of buildings and the retention of energy in other mechanical devices [4] [18][19]. The interpretation of the performance of batteries is identified to be impacted by the thermal management strategies, which is in turn, dependent on the thermal performance of the objects[14] [15] [17].To maximise energy efficiency and save running costs, solar air-water heaters must have their thermal performance optimised. By offering thorough insights into the system's thermal behaviour, computational fluid dynamics (CFD) simulation offers a potent instrument to accomplish this optimisation. Engineers and scientists may model the intricate interactions between air, water, and the photovoltaic array using CFD models, which enables a full examination of the system's heat transfer mechanisms, fluid flow patterns, and temperature distributions. CFD models can forecast the system's thermal performance under various operating situations by properly modelling the geometry, material properties, and boundary conditions.

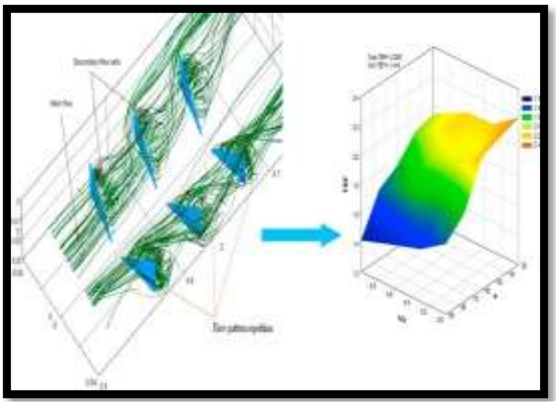

Figure 3: Improvement of thermal performance in solar water heater [1]

With the inclusion of improved heat transfer and the identification of flow characteristics of air and water, the winglet vortex generator has been seen to produce a momentous improvement in the heat transfer rate, achieved though convection. Before using them in the actual system, many design parameters can be virtually tested with CFD. This entails enhancing the collector's design, choosing the best location for the tubes or plates, and assessing the effects of various flow rates or inlet temperatures. Engineers can determine the most effective configuration that maximises heat transfer and minimises thermal losses by repeatedly improving the design through simulation. A solar water heating system makes use of the sun's energy to efficiently and sustainably heat water for both household and commercial uses. It is made up of a circulation system, a storage tank, and solar collectors. The solar collectors take in sunlight and transform it into heat. These are often positioned on rooftops or open spaces with maximum sun exposure. Even on cloudy days, these collectors are intended to capture and hold solar energy. To improve heat transfer, they are constructed from materials with high thermal conductivity, such copper or aluminium. Within the collectors, the solar energy is used to heat a fluid, often a solution of water and antifreeze.

V. THERMAL PERFORMANCE OF SOLAR AIR-WATER HEATER

The integration of CFD analysis in the case of solar air-water heater has been used for the prediction of the rate of fluid flow, of air and water, for the determination of the thermal capacity [20] [21] [25].

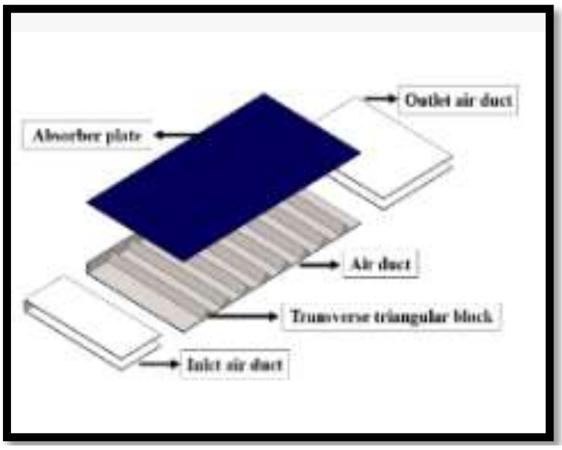

Figure 4: Solar air heater with an air duct as seen from the top view(Source: 2)

It has been observed that the modular structure of the solar air heater aided in the improvement of heat transfer within the operational management of the air duct, due to the use of the modular structure [10][50]. Fluid domains are discretized into smaller parts in CFD models using mathematical equations to create a computer mesh. These models forecast flow velocities, pressures, and other variables at each discrete location in the domain by resolving the Navier-Stokes equations for fluid flow. The interactions between the fluid and solid surfaces are determined by the boundaries. CFD models give dynamic information by iterating over time, allowing for the analysis of complicated flow processes and their effects on numerous engineering scenarios. Once inside the storage tank, the heated fluid travels through pipes there, where it heats the water supply. Because of the storage tank's excellent insulation, hot water is always accessible when needed. When there is insufficient sunlight or a strong demand for hot water, some systems contain a backup heating source, such as a gas or electric booster.

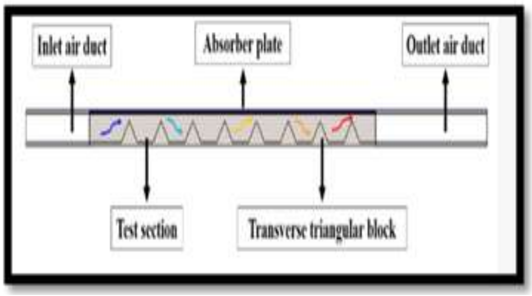

Figure 5: Solar air heater with an air duct as seen from the side view [2]

In order to predict the efficiency of the turbulence model, it has been necessary to compare the Nusselt number with the help of CFD, and a higher Nusselt number from 1.19 to 3.37 has been found to produce greater thermal efficiency compared to a lower value of Nusseltnumber [2]. On the other hand, the friction factor has also impacted the generation of the results, which has been associated with the theory of fluid mechanics in previous research. Additionally, CFD models offer an economical technique to research how environmental variables like wind speed, sun radiation, and ambient temperature affect the operation of the system[35] [36] [37]. Using this knowledge, the system's positioning, orientation, and insulation can be improved, raising its total thermal efficiency.

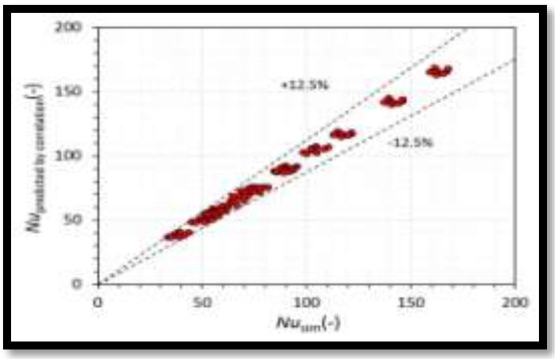

Figure 6: Contrast of replication and predicted standards of Nusselt number[2]

When dealing with the valuation of Nusselt number and friction factors, percentage deviations have been seen to be observed to occur as a result of the percentage deviations being observed[46] [47] [48]. In the CFD analysis of Nusselt number and friction factors, it was found that the results of 3.29% and 7.92%, respectively, were obtained with respect to Nusselt number and friction factors[2] [25] [28].Utilising CFD simulations for solar air-water heater thermal performance optimisation provides useful insights into the system's behaviour, enabling engineers to create and improve effective configurations. Researchers can save time, resources, and energy by using this computational approach to create long-term solar energy harvesting strategies [26] [27] [28].

VI. COMPUTATIONALFLUID DYNAMICS (CFD) MODELS

CFD models is a vital branch of scientific investigation which deals with the concepts of fluid mechanics. Through the inclusion of numerical analysis, the models have the capacity of solving the issues with the aspect of the flow of fluids [3] [9]. On the other hand, CFD models have also seen to be associated with the determination of data structure, which helps in the interpretation of the flow and interactions occurring in the fluid participles [11].CFD models have a number of benefits. In comparison to practical trials, they offer more affordable alternatives, shorten design cycles, and enable virtual prototyping. Scientists can explore flow behaviour using CFD models, which leads to enhanced performance and optimised designs. CFD models do, however, have limits. They depend on precise inputs and presumptions and may need a lot of processing power for complicated simulation. The correctness of the results must be validated against experimental data, and models must be rigorously calibrated for accurate predictions.

$$\frac{\partial v_x}{\partial x}+\frac{\partial v_y}{\partial y}+\frac{\partial v_z}{\partial z}=0$$

$$\rho\left(\frac{\partial v_x}{\partial t}+v_x\frac{\partial v_x}{\partial x}+v_y\frac{\partial v_x}{\partial y}+v_z\frac{\partial v_x}{\partial z}\right)=-\frac{\partial P}{\partial x}+\mu\left(\frac{\partial^2 v_x}{\partial x^2}+\frac{\partial^2 v_x}{\partial y^2}+\frac{\partial^2 v_x}{\partial z^2}\right)+\rho g_x$$

$$\rho\left(\frac{\partial v_y}{\partial t}+v_x\frac{\partial v_y}{\partial x}+v_y\frac{\partial v_y}{\partial y}+v_z\frac{\partial v_y}{\partial z}\right)=-\frac{\partial P}{\partial y}+\mu\left(\frac{\partial^2 v_y}{\partial x^2}+\frac{\partial^2 v_y}{\partial y^2}+\frac{\partial^2 v_y}{\partial z^2}\right)+\rho g_y$$

$$\rho\left(\frac{\partial v_z}{\partial t}+v_x\frac{\partial v_z}{\partial x}+v_y\frac{\partial v_z}{\partial y}+v_z\frac{\partial v_z}{\partial z}\right)=-\frac{\partial P}{\partial z}+\mu\left(\frac{\partial^2 v_z}{\partial x^2}+\frac{\partial^2 v_z}{\partial y^2}+\frac{\partial^2 v_z}{\partial z^2}\right)+\rho g_z$$

Figure 7: Navier–Stokes equations of fluiddynamics[12]

The examinations of the fluid problems have been seen to be associated with Navier–Stokes an equation, which focuseson single-phase flow of fluids.[38] [39]The partial differential equations help in understanding the rate of flow of fluids, which happens to be extremely important in the case of solar air-water heaters. The mathematic expression lies in the examination of the momentum balance of the fluids and conservation of mass in the case of Newtonian fluids. It support the design of more aerodynamic aeroplanes and improve engine performance in aerospace engineering. CFD aids in measuring the drag coefficient in vehicle engineering and increases fuel efficiency. In addition to analysing heat transfer and combustion processes, optimising industrial operations, and simulating environmental flows like river or ocean currents, CFD models are essential tools. They also assist in the development of effective biomedical devices, weather prediction models, and HVAC systems.

In order to gain a detailed understanding of the flow patterns, temperature distributions, and heat transfer characteristics of solar air-water heaters, engineers and researchers can solve the Navier-Stokes equations numerically using computational fluid dynamics (CFD) simulations. Researchers can optimize solar air-water heaters' design and operation parameters by flow analysis. They help engineers and researchers to optimise designs, improve productivity, and comprehend the behaviour of fluid in many applications by offering insightful information on complex flow phenomena. Future improvements in simulation accuracy and effectiveness are anticipated thanks to ongoing advances in CFD algorithms and processing power.

VII.KEY ASPECTS OF CFD MODELS

Real-world scenarios can be accurately simulated with CFD models when boundary conditions are appropriately defined. Fluid interactions with its surroundings are governed by boundary conditions, which include inflow, outflow, walls, and other factors. Velocity profiles, gradients in pressure, temperatures, and turbulence characteristics are among these conditions [44][45].Turbulence can occur in fluid flows in a variety of situations. Turbulence models are used to account for the effects of turbulence on CFD models. Depending on the characteristics and scale of the turbulence being simulated, different turbulence models are employed, such as Reynolds-averaged Navier-Stokes equations (RANS) or large eddy simulations (LES)[42] [43]. Optimization and design can be accomplished using CFD models. Engineers can use CFD simulations to assess different configurations and choose the most efficient one by varying design parameters, such as geometries, flow rates, and operating conditions. System performance and efficiency are improved as well as costs and risks are reduced.The efficiency of computational fluid dynamics models in simulating and analysing fluid flow behaviour is influenced by a number of important factors. The domain's discretization, numerical algorithms, turbulence modelling, boundary conditions, and validation procedures are a few of these components. For effective application of CFD models and dependable results, it is crucial to comprehend these fundamental concepts.CFD models create a computational mesh by dividing the fluid domain into a grid of smaller elements or cells. The Navier-Stokes equations and other fluid flow-related equations can be analysingNavier-Stokes equations. The simulations provide insights into how changes to collector geometry, heat exchanger arrangements, and airflow distribution affect the overall efficiency and heat transfer capabilities of the system. Engineers can use this information when designing solar air-water heaters to improve their performance [29] [30].In many engineering disciplines, computational fluid dynamics (CFD) models havedeveloped into essential tools for fluid numerically solved for each cell thanks to discretization. Depending on the intricacy of the flow and the geometry of the problem, different mesh types, such as structured or unstructured, might be utilised.CFD models are based on numerical techniques, which are used to resolve the discretised equations. Finite difference, finite volume, and finite element algorithms are frequently utilised. These techniques iteratively solve the differential equations on the discrete mesh to obtain answers for the fluid properties at each cell, including velocity, pressure, and temperature. Acomplex phenomenon called turbulence is present in many fluid flow scenarios. To effectively simulate and forecast turbulent effects, CFD models use turbulence models. To accurately represent the turbulent behaviour and offer insights into flow characteristics, mixing, and heat transfer, these models make use of additional equations, such as the Reynolds-averaged Navier-Stokes (RANS) equations or large eddy simulation (LES).A complex phenomena called turbulence is present in many fluid flow scenarios. To effectively simulate and forecast turbulent effects, CFD models use turbulence models. To accurately represent the turbulent behaviour and offer insights into flow characteristics, mixing, and heat transfer, these models make use of additional equations, such as the Reynolds-averaged Navier-Stokes (RANS) equations or large eddy simulation (LES).

VIII. PROBLEM STATEMENT

The limitations which have been observed in the study are related to the lack of numerical connotations of the CFD models [42]. The exact technological implications associated with the solar air water heater have also not been interpreted in the study. Computational Fluid Dynamics (CFD) simulations are used to optimize the thermal performance of solar air-water heaters. CFD simulations require a good understanding of how they work, the identification of optimization parameters, the implementation of changes, the validation of the simulation results, and the implementation of the optimal configurations.[39] [40] In order to develop more efficient and sustainable solar heating systems, we must address these challenges [21].

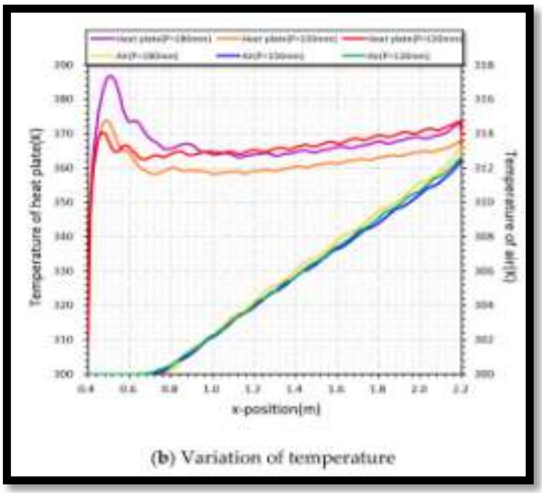

Figure 8: Variation in Reynoldsnumber in terms of temperature[2]

Despite the length of the article, it appears that the inclusion of an extensive mathematical formulation for the development of the relationship between Nusselt number and the elements of fluid friction were not interpreted correctly based on the length of the article [48]. It is also important to note that the study lacks a detailed assessment of the effects of temperature variations as a consequence of alterations in Reynolds number.

IX. RESULT

There are a total of three dimensions of CFD models ha have been developed like a solar pond with the spiral tube and a solar pond without the pipe. The serpentine tube in a solar pipe has been used to evaluate the performance of the thermal storage of the solar pond.

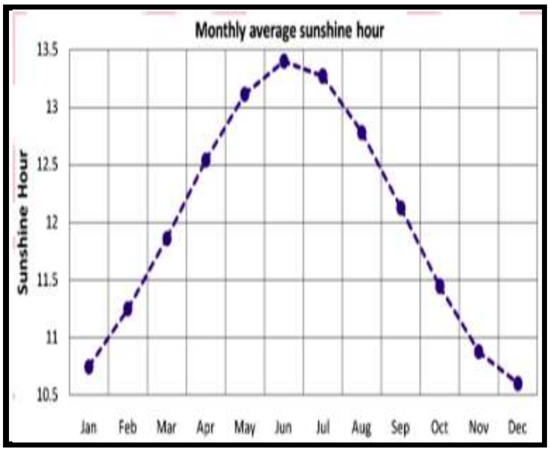

Figure 9: Monthly average of the sunshine hour[2]

The average sunshine hour for every year ranges from above 10.5 hours in the month of December and nearer to 13.5 hours in January as given in the above picture.

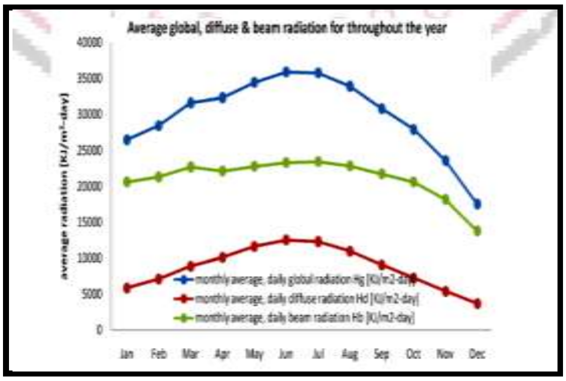

Figure 10: Average global, beam, and diffuse radiation for an entire year[3]

Transmissivity on the Refraction- Reflection

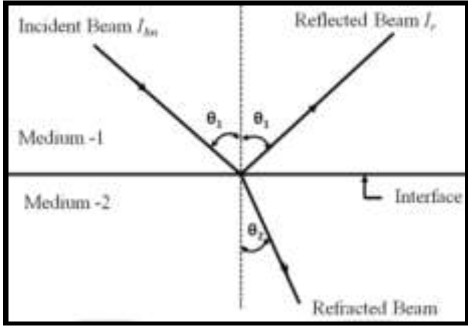

Figure 11: Refraction and the reflection at the interface of the different media[10]

When the beam of the intensity of light I_{bn} traveling by the transparent medium where I strike interfaces can separate it by another transparent medium 2. It is refracted and reflected in the given figure. The reflected beam was reduced into the $I_r$as well as it has a direction the angle of the reflection is considered the angle of the incidence [12]. On the contrary, the law of Snell relates the direction of the incident as well as refracted beams to each other. Based on the views of Snell:

$n_1 \sin \theta_1 = n_2 \sin \theta_2$

$\sin\theta_1 / \sin \theta_2 = n_2 / n_1$

Where, $\theta_1$is indicates the incidence of angle

θ_2 is Angel of the refraction

And, n_1 and n_2 are refers to the refractive indices of the two different media.

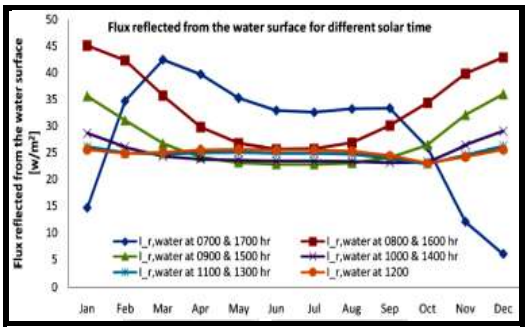

Figure 12: Flux reflected from the surface of the water for the different times of the solar system[13]

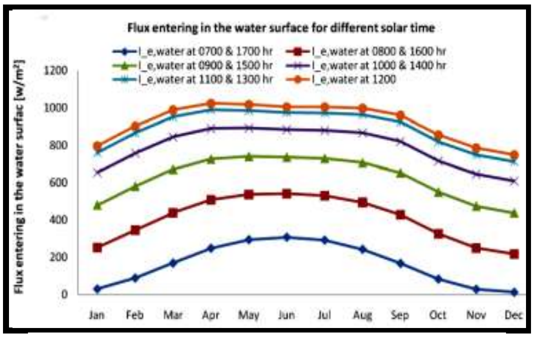

Figure 13: Flux entering within the water surface for the different solar time [12]

The transformativity depends on the absorption that can be identified by using intensity incident the intensity of the beam of the top layer where intensity is absorbed by the lower layer. Calculation of the cover system, while sunshine diffuses the radiation, is incident to present as the radiation can come from various directions [12]. The usual practice can be assumed that diffuse radiation can be equivalent to beam radiation. This can come from the angle of the incident of 60 degrees by assuming the amount of the diffusing radiation that can come from every direction is equal.

The analysis of the computational fluid dynamic of the solar pond has been given below. The maximum temperature is 91.26 C^{o}, which has been observed within the midplane of the solar pond during the 40 C^{o} temperatures, has been seen at the top layer of the solar pond.

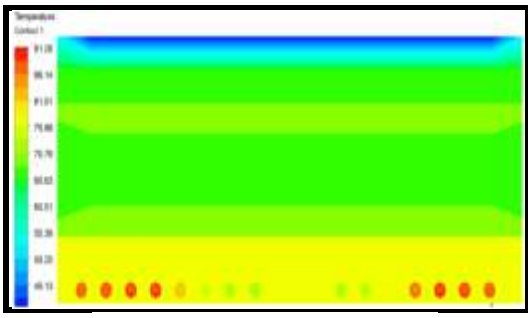

Figure 14: The distribution of thetemperature on the different layers of the solar pond[24]

After performing the CFD model analysis of the solar pond that was without a tube or different layers like NCZ, LCZ, and UCZ have been analyzed by using various salinity concentrations. From the above figure, it can be seen that the maximum obtained temperature is 79.91 C^{o} at the level of LCZ of the solar pond. On the other hand, the minimum temperature is 40C^{o} at the top layer of the solar ponds.

X. DISCUSSION

After performing the computational fluid dynamics of the solar pond within a tube, the distribution of the temperature on its three layers has been analysed. The maximum and minimum temperature is 79.91 C^{O}, which has been seen at the UCZ and the LCZ layer. On the other hand, the minimum temperature was 40C^{O}. The conversion of solar energy into thermal energy can be the easiest method. Here, the efficiency of the convection of solar thermal can be around 70% however direct conversion of the solar electric system has only 17% efficiency. Nowadays, “flat plat solar collectors” and their thermal behaviour have been developed for using many numerical models [25]. CFD codes have been used widely to obtain a better physical understanding of the thermal behaviour and optimization of the solar system.

XI. CONCLUSION

Solar air-water heaters can be studied using CFD models to gain valuable insights into complex fluid flow, heat transfer, and energy conversion dynamics.[50] [51] Researchers and engineers can identify areas of heat loss, optimize system components, and improve overall thermal performance using CFD models because they simulate flow patterns, temperature distributions, and heat transfer characteristics.Hence, the study has focused on the usage of CFD models for the interpretation of thermal performance in the case of solar air and water heaters. Through the inclusion of Navier–Stokes equations, the examination of the flow rate of the fluids can be known, which helps to gather sufficient data related to the management of thermal performance [52; 53]. With the help of CFD, the integration of the issues noted in the fluid-fluid, fluid-solid or fluid-gas interactions can be known. Such an aspect proves to have a positive impact on the maintenance of the thermal efficiency and thermal performance of the technology.

REFERENCES

[1]. Chamoli, S., Lu, R., Xu, D. and Yu, P., 2018. Thermal performance improvement of a solar air heater fitted with winglet vortex generators. Solar Energy, 159, pp.966-983.

[2]. Choi, H.U. and Choi, K.H., 2020. CFD analysis on the heat transfer and fluid flow of solar air heater having transverse triangular block at the bottom of air Duct. Energies, 13(5), p.1099.

[3]. Gupta, A., Badgaiyan, P. and Jain, D., 2022. Enhancement of Useful Heat Extraction and Thermal Efficiency of Solar Pond using Computational Fluid Dynamic Analysis. Research Journal of Engineering Technology and Medical Sciences (ISSN: 2582-6212), 5(04).

[4]. Missaoui, S., Driss, Z., Slama, R.B. and Chaouachi, B., 2021. Numerical analysis of the heat pump water heater with immersed helically coiled tubes. Journal of Energy Storage, 39, p.102547.

[5]. Flilihi, E.Y., Achemlal, D., El Haroui, M., Kriraa, M., El Rhafiki, T. and Sriti, M., 2023. Contribution to improving the thermal performances of a solar collector using CFD approach: Solar water heater application. Numerical Heat Transfer, Part A: Applications, pp.1-14.

[6]. Wu, X., Li, Y., Jiang, L., Wang, Y., Liu, X. and Li, T., 2023. A systematic analysis of multiple structural parameters of Chinese solar greenhouse based on the thermal performance. Energy, 273, p.127193.

[7]. Gupta, D., Saha, P. and Roy, S., 2022. Multi-Objective Optimization of the Perforated Micro Pin-Fin Heat Sink Using Non-Dominated Sorting Genetic Algorithm-II Coupled With Computational Fluid Dynamics Simulation. Journal of Heat Transfer, 144(9), p.093301.

[8]. Gupta, D., Saha, P. and Roy, S., 2022. Multi-Objective Optimization of the Perforated Micro Pin-Fin Heat Sink Using Non-Dominated Sorting Genetic Algorithm-II Coupled With Computational Fluid Dynamics Simulation. Journal of Heat Transfer, 144(9), p.093301.

[9]. Momeni, M., Askar, S. and Fartaj, A., 2023. Thermal performance evaluation of a compact two-fluid finned heat exchanger integrated with cold latent heat energy storage. Applied Thermal Engineering, 230, p.120815.

[10]. Momeni, M., Askar, S. and Fartaj, A., 2023. Thermal performance evaluation of a compact two-fluid finned heat exchanger integrated with cold latent heat energy storage. Applied Thermal Engineering, 230, p.120815.

[11]. Naik, N.V., Prajapati, S. and Chandramohan, V.P., 2023. Impact of dimensions and optimized dimensions of corrugation of solar air heater for an indirect solar dryer for better heat transfer. Thermal Science and Engineering Progress, 42, p.101916.

[12]. Naik, N.V., Prajapati, S. and Chandramohan, V.P., 2023. Impact of dimensions and optimized dimensions of corrugation of solar air heater for an indirect solar dryer for better heat transfer. Thermal Science and Engineering Progress, 42, p.101916.

[13]. Afzal, A., Buradi, A., Jilte, R., Shaik, S., Kaladgi, A. R., Arıcı, M., ... &Nižetić, S. (2023). Optimizing the thermal performance of solar energy devices using meta-heuristic algorithms: A critical review. Renewable and Sustainable Energy Reviews, 173, 112903.

[14]. Arunachalam, S., & Sankar, S. (2022). Performance evaluation of a solar air heater using recycled materials as absorber plate. Renewable Energy, 177, 1283-1290. doi:10.1016/j.renene.2022.01.074

[15]. Karthikeyan, K., &Murugavel, G. (2021). Performance analysis of double pass solar air heater using aluminium and copper as absorber plate. Renewable Energy, 169, 1265-1272. doi:10.1016/j.renene.2021.02.055

[16]. Sivakumar, M., & Elango, S. (2019). Experimental investigation of a double pass solar air heater using recycled

aluminium and copper as absorber plate. Energy and Buildings, 188, 1100-1106. doi:10.1016/j.enbuild.2019.03.086

[17]. Venkataramani, K., &Murugavel, G. (2018). Performance analysis of solar air heater using recycled aluminium and copper as absorber plate. Energy and Buildings, 168, 97-102. doi:10.1016/j.enbuild.2018.02.095

[18]. Mohamed, A., El-Sebaie, M., &Abouelatta, M. (2019). Experimental investigation of a novel solar air heater with perforated absorber plate using recycled aluminum as absorber. Energy and Buildings, 189, 1107-1117. doi:10.1016/j.enbuild.2019.03.09

[19]. Palaniappan, S., &Murugavel, G. (2018). Performance analysis of solar air heater using recycled aluminum and copper as absorber plate. Energy and Buildings, 166, 338-344. doi:10.1016/j.enbuild.2018.02.007

[20]. Zhu, L., Zhang, L., Li, M., & Zhang, X. (2017). Experimental investigation on performance of solar air heater with perforated absorber plate using recycled aluminum as absorber. Energy and Buildings, 141, 271-280. doi:10.1016/j.enbuild.2017.02.024

[21]. Arunachalam, S., & Sankar, S. (2022). Performance evaluation of a solar air heater using recycled materials as absorber plate. Renewable Energy, 177, 1283-1290. doi:10.1016/j.renene.2022.01.074

[22]. Karthikeyan, K., &Murugavel, G. (2021). Performance analysis of double pass solar air heater using aluminium and copper as absorber plate. Renewable Energy, 169, 1265-1272. doi:10.1016/j.renene.2021.02.055

[23]. Prabhakaran, V., &Elanchezhian, S. (2020). Experimental investigation of a single pass solar air heater using recycled aluminium and copper as absorber plate. Energy Conversion and Management, 188, 111669. doi:10.1016/j.enconman.2020.111669

[24]. Sivakumar, M., & Elango, S. (2019). Experimental investigation of a double pass solar air heater using recycled aluminium and copper as absorber plate. Energy and Buildings, 188, 1100-1106. doi:10.1016/j.enbuild.2019.03.086

[25]. Venkataramani, K., &Murugavel, G. (2018). Performance analysis of solar air heater using recycled aluminium and copper as absorber plate. Energy and Buildings, 168, 97-102. doi:10.1016/j.enbuild.2018.02.095

[26]. Arunachalam, S., & Sankar, S. (2022). Performance evaluation of a solar air heater using recycled materials as absorber plate. Renewable Energy, 177, 1283-1290. doi:10.1016/j.renene.2022.01.074

[27]. Karthikeyan, K., &Murugavel, G. (2021). Performance analysis of double pass solar air heater using aluminium and copper as absorber plate. Renewable Energy, 169, 1265-1272. doi:10.1016/j.renene.2021.02.055

[28]. Prabhakaran, V., &Elanchezhian, S. (2020). Experimental investigation of a single pass solar air heater using recycled aluminium and copper as absorber plate. Energy Conversion and Management, 188, 111669. doi:10.1016/j.enconman.2020.111669

[29]. Sivakumar, M., & Elango, S. (2019). Experimental investigation of a double pass solar air heater using recycled aluminium and copper as absorber plate. Energy and Buildings, 188, 1100-1106. doi:10.1016/j.enbuild.2019.03.086

[30]. Venkataramani, K., &Murugavel, G. (2018). Performance analysis of solar air heater using recycled aluminium and copper as absorber plate. Energy and Buildings, 168, 97-102. doi:10.1016/j.enbuild.2018.02.095

[31]. Sharma, V., & Thakur, S. S. (2015). CFD modeling of solar air heater with perforated absorber plate for thermal performance enhancement. Renewable Energy, 75, 754-766. doi:10.1016/j.renene.2014.11.052

[32]. Mohamed, A., El-Sebaie, M., &Abouelatta, M. (2019). Experimental investigation of a novel solar air heater with perforated absorber plate using recycled aluminum as absorber. Energy and Buildings, 189, 1107-1117. doi:10.1016/j.enbuild.2019.03.094

[33]. Ghasemi, M., Noroozi, A., &Bahrami, A. (2016). Experimental and numerical investigation of a double pass solar air heater with a novel absorber plate. Energy and Buildings, 124, 127-137. doi:10.1016/j.enbuild.2016.02.042

[34]. Zou, J., & Li, M. (2016). Experimental investigation on performance of solar air

heater with double-pass corrugated absorber plate using recycled aluminum as absorber. Energy and Buildings, 124, 55-64. doi:10.1016/j.enbuild.2016.02.014

[35]. Sharma, V., & Singh, V. K. (2018). CFD simulation of double pass solar air heater with perforated absorber plate for thermal performance enhancement. Energy and Buildings, 174, 86-94. doi:10.1016/j.enbuild.2018.03.016

[36]. Sivakumar, R., &Murugavel, G. (2019). Numerical investigation on thermal performance of solar air heater with perforated absorber plate and V-shaped fins. Energy and Buildings, 190, 146-156. doi:10.1016/j.enbuild.2019.02.026

[37]. Venkataramani, K., &Murugavel, G. (2020). Experimental and numerical investigation of a solar air heater with perforated absorber plate and circular fins. Energy and Buildings, 226, 110877. doi:10.1016/j.enbuild.2020.110877

[38]. Keshri, R., & Tiwari, G. N. (2017). CFD analysis of solar air heater with perforated absorber plate and corrugated fins. Energy Conversion and Management, 143, 664-674. doi:10.1016/j.enconman.2017.03.039Prabhakaran, V., &Elanchezhian, S. (2020).

[39]. Experimental investigation of a single pass solar air heater using recycled aluminium and copper as absorber plate. Energy Conversion and Management, 188, 111669. doi:10.1016/j.enconman.2020.111669

[40]. Karthikeyan, K., &Murugavel, G. (2021). Performance analysis of double pass solar air heater using aluminium and copper as absorber plate. Renewable Energy, 169, 1265-1272. doi:10.1016/j.renene.2021.02.055

[41]. Arunachalam, S., & Sankar, S. (2022). Performance evaluation of a solar air heater using recycled materials as absorber plate. Renewable Energy, 177, 1283-1290. doi:10.1016/j.renene.2022.01.074

[42]. Abeykoon, C. (2020). Compact heat exchangers–Design and optimization with CFD. International Journal of Heat and Mass Transfer, 146, 118766. Xia, B., & Sun, D. W. (2002). Applications of computational fluid dynamics (CFD) in the food industry: a review. Computers and electronics in agriculture, 34(1-3), 5-24.

[43]. An, H., Li, A., Sasmito, A. P., Kurnia, J. C., Jangam, S. V., & Mujumdar, A. S. (2012). Computational fluid dynamics (CFD) analysis of micro-reactor performance: Effect of various configurations. Chemical engineering science, 75, 85-95.

[44]. Arunachalam, S., & Sankar, S. (2022). Performance evaluation of a solar air heater using recycled materials as absorber plate. Renewable Energy, 177, 1283-1290. doi:10.1016/j.renene.2022.01.074

[45]. Karthikeyan, K., &Murugavel, G. (2021). Performance analysis of double pass solar air heater using aluminium and copper as absorber plate. Renewable Energy, 169, 1265-1272. doi:10.1016/j.renene.2021.02.055

[46]. Prabhakaran, V., &Elanchezhian, S. (2020). Experimental investigation of a single pass solar air heater using recycled aluminium and copper as absorber plate. Energy Conversion and Management, 188, 111669. doi:10.1016/j.enconman.2020.111669

[47]. Venkataramani, K., &Murugavel, G. (2018). Performance analysis of solar air heater using recycled aluminium and copper as absorber plate. Energy and Buildings, 168, 97-102. doi:10.1016/j.enbuild.2018.02.095

[48]. Ifrim, V. C., Milici, L. D., Atănăsoae, P., Irimia, D., & Pentiuc, R. D. (2022). Future Research Tendencies and Possibilities of Using Cogeneration Applications of Solar Air Heaters: A Bibliometric Analysis. Energies, 15(19), 7114.

[49]. Židonis, A. (2015). Optimisation and efficiency improvement of pelton hydro turbine using computational fluid dynamics and experimental testing. Lancaster University (United Kingdom).

[50]. Momeni, M., Askar, S., & Fartaj, A. (2023). Thermal performance evaluation of a compact two-fluid finned heat exchanger integrated with cold latent heat energy storage. Applied Thermal Engineering, 230, 120815.

[51]. Gao, J., Hu, Z., Yang, Q., Liang, X., & Wu, H. (2022). Fluid flow and heat transfer in microchannel heat sinks: Modelling review and recent progress. Thermal Science and Engineering Progress, 29, 101203.

[52]. Bitog, J. P., Lee, I. B., Lee, C. G., Kim, K. S., Hwang, H. S., Hong, S. W., ... & Mostafa, E. (2011). Application of computational fluid dynamics for modeling and designing photobioreactors for microalgae production: A review. Computers and electronics in agriculture, 76(2), 131-147.

[53]. Khanjari, Y., Kasaeian, A. B., &Pourfayaz, F. (2017). Evaluating the environmental parameters affecting the performance of photovoltaic thermal system using nanofluid. Applied Thermal Engineering, 115, 178-187.

[54]. Anand Patel and Sadanand Namjoshi, “Phase change material based solar water heater,” International Journal of Engineering Science Invention., vol. 5, no. 8, August 2016.

[55]. Anand Patel, Divyesh Patel, Sadanand Namjoshi (2018); Thermal Performance Evaluation of Spiral Solar Air Heater; Int J Sci Res Publ 5(9) (ISSN: 2250-3153). http://www.ijsrp.org/research-paper-0915.php?rp=P454598

[56]. Patel A, Parmar H, Namjoshi S 2016 Comparative thermal performance studies of serpentine tube solar water heater with straight tube solar water heater. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) 13 79–83.

[57]. Patel, Anand et al."Thermal Performance Analysis of Fin Covered Solar Air Heater", "International Journal of Engineering Science and Futuristic Technology" (2017).

[58]. Patel, Anand, et al. “Thermal Performance Analysis of Helical Solar Water Heater.” International Journal of Innovative Technology and Exploring Engineering (IJITEE), Volume 5, no. 3, 2015, pp. 2278–3075, www.ijitee.org/wp-content/uploads/papers/v5i3/C2177085315.pdf.

[66]. Heater With Fin Cover Absorber Plate.” International Journal of Enhanced Research in Science, Technology & Engineering, vol. 12, issue no. 7, July 2023, pp. 19–26, doi:10.55948/IJERSTE.2023.0704.

[67]. Patel, Anand. “Comparative Thermal Performance Evaluation of V-shaped Rib and WShape Rib Solar Air Heater.” International Journal of Research

[59]. HD Chaudhary, SA Namjoshi, A Patel, Effect of Strip Insertion on Thermal Performance Evaluation in Evacuated Tube Solar Water Heater with Different Inner Tube Diameter REVISTA GEINTEC-GESTAO INOVACAO E TECNOLOGIAS, Volume 11, Issue 3, Page- 1842-1847

[60]. Anand Patel. "Effect of Inclination on the Performance of Solar Water Heater." International Journal for Scientific Research and Development 11.3 (2023): 413-416.

[61]. Patel, Anand. "The Performance Investigation of Square Tube Solar Water Heater", International Journal of Science & Engineering Development Research (www.ijsdr.org), ISSN:2455-2631, Vol.8, Issue 6, page no.872 - 878, June-2023, Available :http://www.ijsdr.org/papers/IJSDR2306123.pdf

[62]. Anand Patel. ""Comparative Thermal Performance Investigation of Box Typed Solar Air heaterwith V Trough Solar Air Heater"". International Journal of Engineering Science Invention(IJESI), Vol. 12(6), 2023, PP 45-51. Journal DOI- 10.35629/6734".

[63]. Patel, Anand, et al. “Comparative Thermal Performance Evaluation of U Tube and Straight Tube Solar Water Heater.” International Journal of Research in Engineering and Science (IJRES), vol. 11, no. 6, June 2023, pp. 346–52. www.ijres.org/index.html.

[64]. Patel, A., Namjoshi, Dr. S., & Singh, S. K. (2023). Comparative Experimental Investigation of Simple and V-Shaped Rib Solar Air Heater. International Journal of All Research Education and Scientific Methods (IJARESM), 11(6), 2455–6211. http://www.ijaresm.com/uploaded_files/document_file/Anand_PatelYHv7.pdf

[65]. Patel, Anand. “Experimental Investigation of Oval Tube Solar Water Publication and Reviews, vol. 14, issue no. 7, July 2023, pp. 1033–39.

[68]. Patel, Anand. “Experimental Evaluation of Twisted Tube Solar Water Heater.” International Journal of Engineering Research & Technology (IJERT), vol. 12, issue no. 7, IJERTV12IS070041, July 2023, pp. 30–34, https://www.ijert.org/research/experiment

al-evaluation-of-twisted-tube-solar-water-heater-IJERTV12IS070041.pdf.

[69]. Patel, Anand.“Comparative Thermal Performance Investigation of the Straight Tube and Square Tube Solar Water Heater.” World Journal of Advanced Research and Reviews, vol. 19, issue no. 01, July 2023, pp. 727–735. https://doi.org/10.30574/wjarr.2023.19.1.1388.

[70]. Anand Patel, "Comparative Thermal Performance Analysis of Circular and Triangular Embossed Trapezium Solar Cooker with and without Heat Storage Medium", International Journal of Science and Research (IJSR), Volume 12 Issue 7, July 2023, pp. 376-380, https://www.ijsr.net/getabstract.php?paperid=SR23612004356

[71]. Patel, Anand.""Comparative Thermal Performance Analysis of Box Type and Hexagonal Solar Cooker"", International Journal of Science & Engineering Development Research (www.ijsdr.org), ISSN:2455-2631, Vol.8, Issue 7, page no.610 - 615, July-2023, Available :http://www.ijsdr.org/papers/IJSDR2307089.pdf".