Computational Investigation of Solar Collectors for Optimal Power Generation

Vinod Sironjiya*, Prof. Khemraj Bairagi**

*Research scholar, Department of Mechanical Engineering, School of Engineering And Technology, Vikram University Ujjain (M.P.), India.
Email: Sironjiyavinod1234@gmail.com
**Assistant Professor, Department of Mechanical Engineering, School Of Engineering And Technology, Vikram University Ujjain, India.
Email: Khemrajberagi@gmail.com

Abstract: This research delves into various solar collector designs employed for power generation, utilizing a combination of flat and staggered configurations. The study employs advanced simulation tools, with the ANSYS CFX program facilitating fluid dynamics analysis and the Creo design software utilized to generate intricate CAD models. The investigation aims to assess the performance and efficiency of the selected solar collector designs through computational simulations, providing valuable insights for enhancing their effectiveness in harnessing solar energy for power generation. The integration of computational fluid dynamics and CAD modeling enables a comprehensive exploration of the thermal and structural aspects of the solar collectors, contributing to advancements in renewable energy technologies.

Keywords: Solar Chimney, CFD, Pressure, CAD Modeling , ANSYS CFX , Solar Collector

1. INTRODUCTION

According to energy experts, there is growing interest in using unconventional energy sources for power generation [1]. The use of green resource-based power production technologies would contribute to the improvement of several countries' balance of payments. Solar energy is very advantageous for arid and semi-arid regions due to its widespread availability and prevalence as the most abundant renewable energy source. Solar devices of various types are used worldwide to harness solar energy. Scientists have just introduced an intriguing innovation called the solar chimney. The solar chimney idea is shown in Figure 1. The facility is a solar thermal power plant that utilizes an advanced heat transfer technique to convert solar thermal energy into electricity. The successful completion of this project will facilitate the advancement of solar hot air flow power production in developing countries, which is essential for the establishment of

new energy sources and the commercialization of comparable power generation systems [2].

2. LITERATURE REVIEW

In 1982, Haaf et al. [3, 4] provided an explanation of the fundamental physical principles underlying centralized energy generation through the utilization of solar chimney power plants (SCPPs). Following the commencement of operations at the Manzanares pilot plant in June 1982, the initial experimental findings validated the core principles of the original physical model [3, 5]. Subsequently, a model based on empirical data and parameters was employed to forecast the monthly average electrical power generated by the pilot plant. This prediction was made using technical data collected from July 1983 to January 1984 [6]. During the period from July to October 1983, which experienced unusually low levels of precipitation, the model forecasts and experimental data demonstrated a satisfactory level of agreement. Unfortunately, the model failed to accurately replicate the following months of heavy rainfall that took place during the winter and spring of 1984. The performance of the collector is affected by factors such as evaporation, plant growth, and infrared absorption in the air when natural precipitation enters it [3].A more sophisticated parametric model was suggested to recreate extensive vegetation in climates resembling Manzanares [7]. This model captures the long-term effects of rainfall on the plants' performance, taking into account its recurring nature. The cost of electricity production was carefully calculated in [7] and estimated in [3, 7].

3. OBJECTIVE

Several solar collector designs utilized for power generation are investigated in this study. For conducting research on the solar collector, the ANSYS CFX program is utilized, whereas the Creo design software is employed to generate the CAD model. The inquiry employed a combination of flat and staggered collector designs.

4. METHODOLOGY

The numerical investigation of solar chimney is conducted using techniques of computational fluid dynamics. The CFD is based on conservation of mass, momentum and energy. Computational Fluid Dynamics (CFD) serves as a transformative tool in engineering and scientific research, offering a virtual platform to simulate and analyze fluid flow, heat transfer, and related phenomena. By numerically solving complex equations governing fluid dynamics, CFD provides invaluable insights into the behavior of fluids in various applications, ranging from aerodynamics and heat exchangers to environmental studies and energy systems. This versatile technology enables researchers and engineers to visualize and comprehend fluid interactions, optimize designs, and predict performance, ultimately facilitating the development of more efficient and cost-effective solutions. With its ability to simulate real-world scenarios, CFD has become an indispensable asset across diverse industries, revolutionizing the design and analysis processes and contributing significantly to advancements in engineering and scientific research. The methodology employed in this study involves a systematic approach to conduct Computational Fluid Dynamics (CFD) analysis on a solar chimney system. The first step entails precisely defining the research objectives and key parameters for investigation, encompassing collector geometry, chimney height, and environmental conditions. A detailed 3D model of the solar chimney system is then created using CAD software, ensuring accurate representation of physical dimensions. The subsequent phase involves generating a mesh that captures the intricacies of the geometry, with particular attention to refining critical areas like the collector surface. Boundary and initial conditions, including inlet/outlet specifications, ambient temperature, and solar radiation, are rigorously defined. The fluid properties within the solar chimney are specified, accounting for variations in temperature and density. Numerical solvers and turbulence models are carefully chosen, considering the characteristics of the flow and validated against available experimental data. Simulation settings are configured, and the simulations are executed, monitoring convergence and stability throughout. Post-processing involves a thorough analysis of simulation results, including temperature distributions, velocity profiles, and heat transfer rates. Validation against experimental data and sensitivity analyses are conducted to ensure reliability, and insights gained are leveraged for optimization of the solar chimney design. The methodology concludes with comprehensive documentation and reporting of findings, limitations, and recommendations for future research and improvements. Utilizing the revolve and sketch functions of the Creo design software, a computer-aided design (CAD) depiction of the chimney is generated. The ANSYS Design Modeler is utilized to import the developed CAD model. Access to the model is possible through various software applications due to its importation in the Parasolid file format. The CAD model that was imported is depicted in Figure 1.

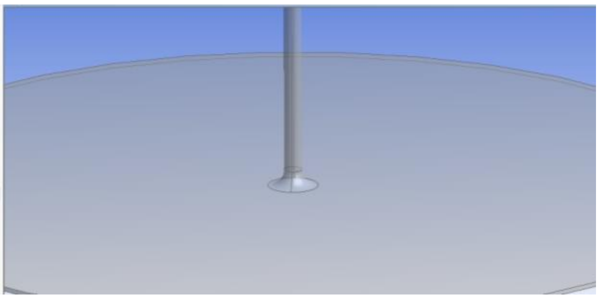

Figure 1: Imported CAD model of chimney

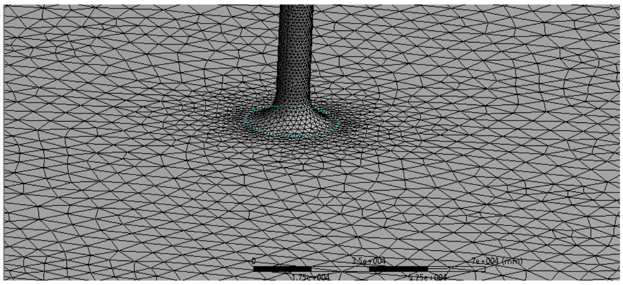

Figure 2: Meshed model of chimney

The model is equipped with precise scaling, a transition ratio of 1.2, and curvature effect meshing. The model of the chimney mesh is illustrated in the third image above.

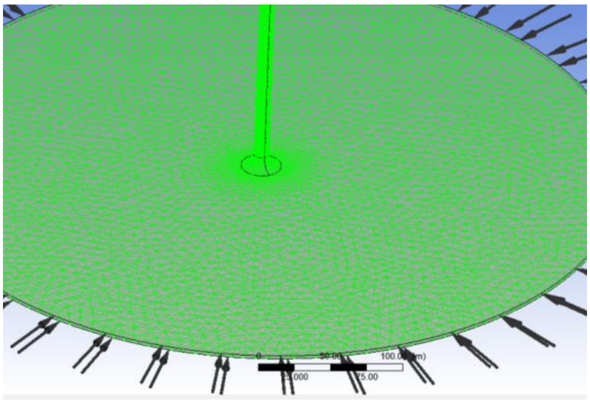

Figure 3: Fluid model definition of chimney

Air is the substance denoted within this particular framework, while the domain is characterized as a fluid type. The turbulence model employed in the research makes use of V-epsilon and reference pressure, both of which are initialized to 1 atm RNG. The exposed face of the chimney is exposed to heat fluxes measuring between 1000 W/m2 and 800 W/m2.

5. RESULTS AND DISCUSSION

Two contour graphs are generated to illustrate the pressure and velocity. One graph depicts a value of 1000W/m2, while the other graph shows a value of 800W/m2.

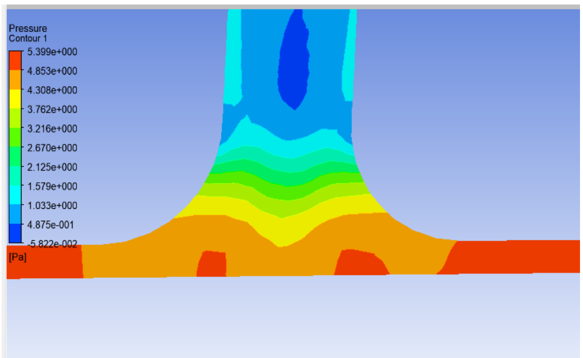

Figure 4: Pressure plot for smooth surface

The pressure distribution in the vertical zone is illustrated in the diagram shown in Figure 8, with an intensity of 1000W/m2. The pressure in this area fluctuates between 4.01 and 5.31 Pascal. When air expands in the direction of the vertical element, a pressure transfer takes place. The pressure is reduced dynamically, resulting in a reading of 1.316 Pa. This is visually represented by a range of vibrant blue colors.

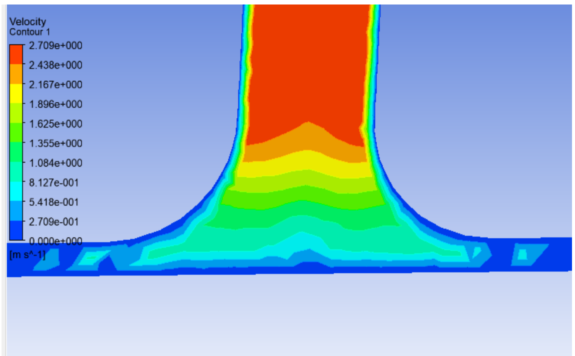

Figure 5: Velocity plot for smooth surface

The velocity curve shown in Figure 9 has a minimum value in the central region located at the bottom. As you near the exit, the air experiences a slight decrease in velocity to 0.0745 m/s before it starts to pick up speed again. The colors green, yellow, and red are associated with different levels of speed. The achieved maximum velocity is 2.65 meters per second.

6. CONCLUSION

The Computational Fluid Dynamics (CFD) analysis conducted on solar chimney technology has provided valuable insights into its performance and efficiency. The utilization of CFD tools has facilitated a detailed examination of fluid dynamics, heat transfer, and overall system behavior within the solar chimney. The findings reveal the significance of various parameters, such as collector design, chimney height, and ambient conditions, in influencing the system's effectiveness. The CFD simulations have enabled a deeper understanding of the complex interactions within the solar chimney, allowing for the optimization of key components to enhance energy conversion. Through systematic analysis, researchers can fine-tune design parameters and improve overall system performance, addressing challenges associated with temperature differentials, flow patterns, and heat losses. As solar energy continues to play a pivotal role in the quest for sustainable solutions, the CFD analysis presented in this study adds a crucial dimension to the optimization and development of solar chimney technology. The knowledge generated through this research offers a pathway for engineers, designers, and policymakers to harness the full potential of solar chimneys as a reliable and efficient source of renewable energy. The continuous integration of CFD methodologies in solar chimney research ensures a robust foundation for further innovations, fostering a greener and more sustainable energy landscapes.

REFERENCES

[1] R. K. Rajput, Power System Engineering. Firewall Media, 2006,pp.198.
[2] Y.Chen, Y.Yang, Y.Wei, Y.Jing-Hui, and Y.Tian, ― Energy Research and Information, vol. 26, no2, p. 117- 121, 2010.
[3] S. Terol, ―The present state of research at the solar chimney power plant in Manzanares (Spain) and future prospects for large scale plants,‖ in Proc. International Congress on Renewable Energy Sources, vol.1,p.1644,1987.
[4] W. Haaf, K. Friedrich, G. Mayr, and J. Schlaich, ―Solar Chimneys Part I: Principle and Construction of the Pilot Plant in Manzanares,‖ International Journal of Solar Energy, vol. 2, pp. 3-20, 1983.
[5] W .Haaf, ―Solar Chimneys - Part II: Preliminary Test Results from the Manzanares Pilot Plant,‖ International Journal of Solar Energy 2 (2), pp.141–161, 1984.
[6] N. Pasumarthi et S. A. Sherif, ― Experimental and theoretical performance of a demonstration solar chimney model—Part I: mathematical model development ‖, International Journal of Energy Research, vol. 22, no. 3, pp. 277-288, 1998.

[7] N. Pasumarthi et S. A. Sherif, ― Experimental and theoretical performance of a demonstration solar chimney model—Part II: experimental and theoretical results and economic analysis ‖, International Journal of Energy Research, vol. 22, no. 5, p. 443-461, 1998.
[8] Ganguli AA and Deshpande S (2020) Three Dimensional CFD Studies of a Solar Chimney: Effect of Geometrical Parameters and Diurnal Variations on Power Generated. Front. Chem. Eng. 2:2. doi: 10.3389/fceng.2020.00002