International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

CFD ANALYSIS OF THERMAL PROPERTIES OF SOLAR AIR HEATER

[1]Shwetanshu Kumar and [2]Vijaykant Pandey

[1]M-Tech Research Scholar, RKDF College of Technology, Bhopal (M.P.), India
[2]Assistant Professor, RKDF College of Technology, Bhopal (M.P.), India

ABSTRACT :

Solar air heating is a type of solar thermal technology that heats air by absorbing solar radiation and using that energy. A transparent front wall and an inlet and outlet are features of the solar air heater housing, which creates a flow channel for heated gases like air. An effort has been made to apply FLUENT, a CFD program, to analyze the fluid flow and heat transfer properties of solar air heaters. ANSYS Workbench was used to generate a 3D model of the solar air heater, which included the air intake, absorber plate, and glass. ANSYS was also used to create the unstructured grid. The ANSYS FLUENT program was utilized to acquire the desired outcomes. Computational fluid dynamics (CFD) is used in this work to analyze the flow and temperature distribution inside the solar air heater.

Keywords: Solar Air Heater, Flow Rate, Temperature, Heat Recovery, Stream Function, Turbulent Dissipation, Velocity Contour, Temperature Contour

INTRODUCTION :

Solar air heating is a solar thermal technology in which the energy from the sun, insolation, is captured by an absorbing medium and used to heat air. Solar air heating is a renewable energy heating technology used to heat or condition air for buildings or process heat applications. It is typically the most cost-effective out of all the solar technologies, especially in commercial and industrial applications, and it addresses the largest usage of building energy in heating climates, which is space heating and industrial process heating.

Solar Air Heater

Solar air heaters are systems that collect solar energy and transfers the heat to passing air, which is either stored or used for space heating. The collectors are often black to absorb more of the sun's energy and a conductive material, often metal, acts as a heat exchanger. There are many different designs and systems may include fans to increase the flow rate of air. Alternatively, a passive collector can be built such that when the hot air rises it draws fresh air through the bottom. Fans can often increase the performance of the system, but require additional parts and adds complexity. Solar air heaters can compliment traditional indoor heating systems by providing a free and clean source of heat (after initial costs). While clouds effect the energy output of the system, the metal will store energy on a hot day and will reduce the impact of momentary cloud cover. To achieve best results, the system should be unshaded and facing the general direction of the sun (south for the northern hemisphere, north for the southern hemisphere).

More than any other solar technology, solar heaters are DIY-friendly, since they require only a basic knowledge of carpentry and electrical skills, can be made of easy-to-find materials, and can be installed on a south-facing wall rather than on a potentially dangerous roof. Solar air heaters are tolerant of less-than-exact construction details. A small air leak will only reduce the heater’s overall efficiency, not leak fluid or potentially overheat or shock you if installed improperly. While care should be taken with any project, the consequences of potential mistakes are much less dire. For homeowners interested in the basics of renewable energy, building a solar air heater can be a great project.

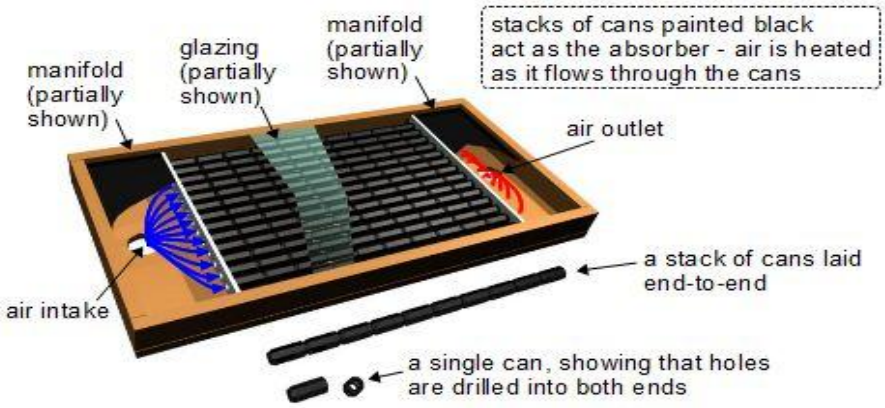

Fig. 1 Solar Air Heater

LITERATURE REVIEW :

1. **Samir A. Dhatkar et.al.,** Solar air heater is one of the valuable heat sources with variety of applications such as space heating and cooling, industrial process heating and drying of fruits and vegetables etc. The major heat losses from a normal solar air collector are through the top cover which reduce the thermal efficiency also, the low heat transfer coefficient between the air stream and the absorber plate is another reason of low thermal efficiency in solar air heaters. Thermal efficiency of double pass solar air heater with porous media is higher than single pass and double pass solar air heater without porous media. Many experiments have been carried out on the performance analysis of double pass solar air heater with porous media and solar air heater with extended surfaces. Effect of various parameters of porous media like pitch, number of layers , bed depth, porosity, thermal conductivity, pitch to wire diameter ratio have been studied. Also these studies includes the design of double pass solar air heater, heat transfer enhancement, pressure drop, type of flow. It is found that more increase in thermal efficiency in comparison with conventional solar air heater. Based on literature review, it is concluded that most of the studies carried out on solar air heater with porous media and extended surfaces. Few studies are carried out on corrugated absorber plate. Improvement of thermal efficiency of solar air heater is to be obtained by enhancing the rate of heat transfer.
2. **Raheleh Nowzaria et.al.,** the double pass solar air heater is constructed and tested for thermal efficiency at a geographic location of Cyprus in the city Famagusta. The absorber plate was replaced by fourteen steel wire mesh layers, 0.2 × 0.2 cm in cross section opening, and they were fixed in the duct parallel to the glazing. The distance between each set of wire mesh layers is 0.5cm to reduce the pressure drop. The wire mesh layers were painted with black before installing them into the collector. The obtained results show that as the mass flow rate increases, the efficiency of the system also increases. The temperature difference (ΔT) between the inlet and outlet air through the system increases as the mass flow rate decreases. The maximum ΔT (53°C) is achieved at the flow rate of 0.011 kg/s. The range of the mass flow rate used in this work is between 0.011 and 0.037 kg/s. It is also found that the average efficiency obtained for the double pass air collector is 53.7% for the mass flow rate of 0.037 kg/s.
3. **Anil Singh Yadav et.al.,** this paper presents the study of heat transfer in a rectangular duct of a solar air heater having triangular rib roughness on the absorber plate by using Computational Fluid Dynamics (CFD). The effect of Reynolds number on Nusselt number was investigated. The computations based on the finite volume method with the SIMPLE algorithm have been conducted for the air flow in terms of Reynolds numbers ranging from 3000-18000. A commercial finite volume package ANSYS FLUENT 12.1 is used to analyse and visualize the nature of the flow across the duct of a solar air heater. CFD simulation results were found to be in good agreement with experimental results and with the standard theoretical approaches. It has been found that the Nusselt number increases with increase in Reynolds number.
4. **Hikmet Esen et.al.,** this paper presents an experimental energy and exergy analysis for a novel flat plate solar air heater (SAH) with several obstacles and without obstacles. For increasing the available heat-transfer area may be achieved if air is flowing simultaneously and separately overland under the different obstacle absorbing plates, instead of only flowing either over or under the different obstacle absorbing plates, leading to improved collector efficiency. The measured parameters were the inlet and outlet temperatures, the absorbing plate temperatures, the ambient temperature, and the solar radiation. Further, the measurements were performed at different values of mass flow rate of air and different levels of absorbing plates in flow channel duct. After the analysis of the results, the optimal value of efficiency is middle level of absorbing plate in flow channel duct for all operating conditions and the double-flow collector supplied with obstacles appears significantly better than that without obstacles. At the end of this study, the exergy relations are delivered for different SAHs. The results show that the largest irreversibility is occurring at the flat plate (without obstacles) collector in which collector efficiency is smallest
5. **Gurpreet Singh et.al.,** An experimental investigation of the effect of geometrical parameters of circular transverse ribs on heat transfer of rectangular duct with heated plate having rib roughness on its underside have been reported. The range of parameters for this study has been decided on the basis of practical considerations of the system and operating conditions of solar air heaters. The experimental investigation encompassed the Reynolds number (Re) range from 2564 to 6206; relative roughness pitch (P/e) of 8, angle of attack (α) of 90° and relative roughness height (e/Dh) is0.047. The thermal efficiency of roughened duct is observed to be 5%-9% more as compare to the smooth duct. The thermal efficiency is increased with increasing the value of Reynolds number
6. **Dharam Singh et.al.,** Solar air heater is solar energy collection devices and this collected solar energy is used for low temperature heating purpose, agriculture drying purpose etc. The heat conversion rate of solar energy to heat energy is low in solar air heater because air is flowing fluids and air has low heat transfer coefficient, which is directly influence the rate of heat transfer from the absorber plate to air. This fluid property heat transfer coefficient can be increase by creating turbulence in flow field and the turbulence is created by providing different shape artificial roughness on absorber plate. In this paper the study of heat transfer in a rectangular duct of a solar air heater having trapezoidal rib roughness on the absorber plate is done by using Computational Fluid Dynamics (CFD). The effect on Nusselt number, heat transfer coefficient, velocity, and temperature parameter are analysed .This CFD simulation is done by fluent 6.3 software using K-ε model
7. **Mokalla Srinivas et.al.,** A solar hybrid energy system having photovoltaic and thermal (PV/T) devices, which produces both thermal and electrical energies simultaneously is considered for analysis. A double pass hybrid solar air (PV/T) heater with slats is designed and fabricated to study its thermal and electrical performance. Air as a heat removing fluid is made to flow through upper and lower channels of the collector. The collector is designed in such way that the absorber plate is partially covered by solar cells. The raise in temperature of the solar cell is expected to decrease its electrical performance. Thin metallic strips called slats are attached longitudinally at the bottom side of the absorber plate to improve the system performance by increasing the cooling rate of the absorber plate. Thermal and electrical performances of the whole system at varying cooling conditions are also presented.

3. CFD

Computer primarily based simulation is mentioned during this chapter. procedure simulation is technique for examining fluid flow, heat transfer and connected phenomena like chemical reactions. This project uses CFD for analysis of flow and warmth transfer. CFD analysis accepted go in the various industries is employed in R&D and producing of craft, combustion engines and in powerhouse combustion similarly as in several industrial applications.

Why computational simulation

Three-dimensional (3D) numerical analysis of whorled coil tubes is dispensed by victimization business CFD tool ANSYS 18.2. this can become troublesome and time overwhelming, if this analysis is dispensed by experimentation. Experimental setup is extremely expensive that's why in my work I take facilitate of CFD to create it easier and fewer time overwhelming.

Computational fluid dynamics

Computational fluid dynamics, because the name implies, could be a subject that deals with procedure approach to fluid dynamics by means that of a numerical resolution of the equations that cause the fluid flow and though it's known as procedure fluid dynamics; it doesn't simply wear down the equations of the fluid flow, it's conjointly generic enough to be ready to solve at the same time along the equations that direct the energy transfer and similarly the equations that verify the chemical process rates and the way the chemical process takings and mass transfer takes place; of these things may be tackled along in a regular format. So, this define permits America to wear down a really complicated flow circumstances in fairly quick time, specified for a specific set of conditions, associate degree engineer would be ready to simulate and see however the flow is happening and what quite temperature distribution there's and what quite product area unit created and wherever they're fashioned, in order that {we can|we will|we area unit able to} build changes to the parameters that area unit below his management to switch the approach that these items are happening. So, therein sense procedure fluid dynamics or CFD becomes a good tool for a designer for associate degree engineer. it's conjointly a good tool for associate degree associate degreealysis for associate degree examination of a reactor or an instrumentality that isn't functioning well as a result of in typical industrial applications, several things is also happening associate degreed what a designer has had in mind at the time of fabricating or coming up with the instrumentality won't be really what an operator of the instrumentality introduces into the instrumentality at the time of operation, perhaps once 5 years or 10 years changes might need taken place in between; and in such a case, the presentation of the instrumentality won't be up to the quality and you'd wish to modify it in such some way that you just will restore performance. So, the question is then, what this can managed to the autumn within the performance associate degreed what quite measures we are able to build while not creating an overall adjustment within the finish of apparatus. Is it potential to urge improved performance from the equipment? Is it potential to extend the productivity? If you wish to appear on of these analysis, then procedure fluid dynamics is employed.

CALCULATION & DESIGNING :

Meshing updates

The big change on the meshing front for this release 15.0 is that the core meshing tools in Ansys now run across multiple cores/threads.
What's interesting is how this has changed. In previous releases, it worked in serial, meshing the first part, then the second, then the next and so on. Now, Ansys works in parallel on a per part basis and does so without the inherent limitations of Ansys.in short if you have got a workstation with multiple cores or threads available, it will use them to process your meshing tasks. In 15.0 onwards, it's now possible to begin a simulation process using previously created meshes, rather than pure analytic geometry from a 3D CAD system. This allows a couple of workflows to function.

Composite design

Moving onto more specialist updates, the Ansys 15 release sees work accelerating on the composites simulation front. While the last major release saw Workbench gain tools to assist with the complex business of defining how composite materials are defined in the system, this 15.0 release sees that work move on and greater integration with ACP (Ansys Composite Prep/Post) for pre and post processing these types of models, from sheet material definition to ply orientation and into how the models are built from the ground up.

Contact and bolt pre-tension

While nowhere near as specialist as composites, another area that's seen greater intelligence added is in the definition of bolt pretension. In previous versions simulation of define a bolted connection between two or more components is complex. 3D CAD integrated simulation tools tend to provide the user with pre-configured tools to define the bolt, tension and influenced areas on a mesh. Ansys workbench 15.0 extends this type of automated tool to give you much greater control we are not only working with real-world values (such as thread pitch), but also pitch diameter, thread starts, thread direction (left or right handed) etc.
The results are computed quickly and once established, can be reused wherever needed in the same model or indeed, across multiple studies.

Exporting Results for Fluid Solid Interface

Compared to the pervious release, the Export Results property of the Fluid Solid Interface boundary condition in workbench 15.0 is now, by default, set to No.

Solving Units

Mechanical analysis now supports a new unit quantity, Temperature Gradient, for temperature changes per unit length.

Contact and Connection Enhancements

The Connections Worksheet now provides information for the properties of Spring Connections and Beam Connections

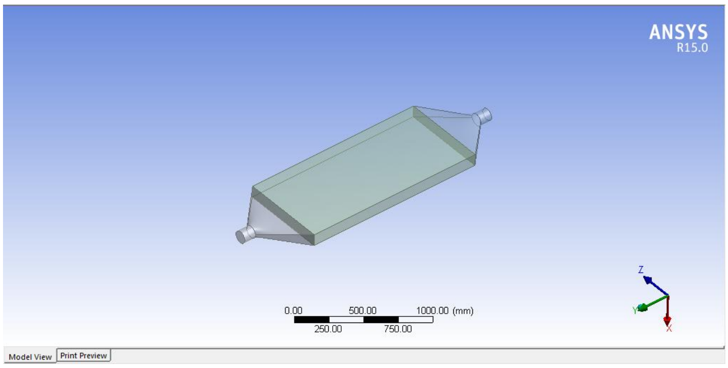

Fig. 2 Design 1 – Plain surface

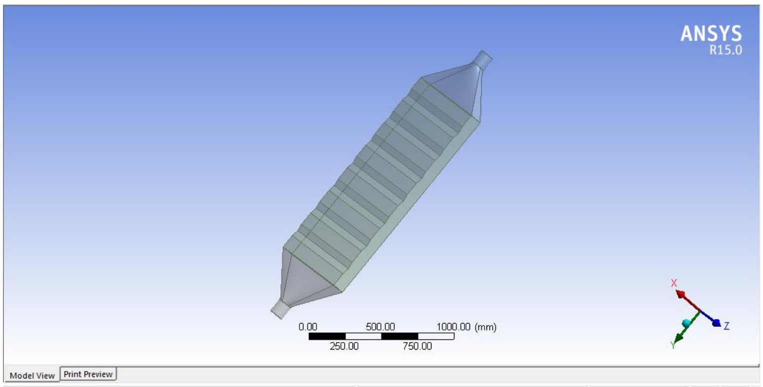

Fig. 3 Design 2 – Normal Zig zag surface

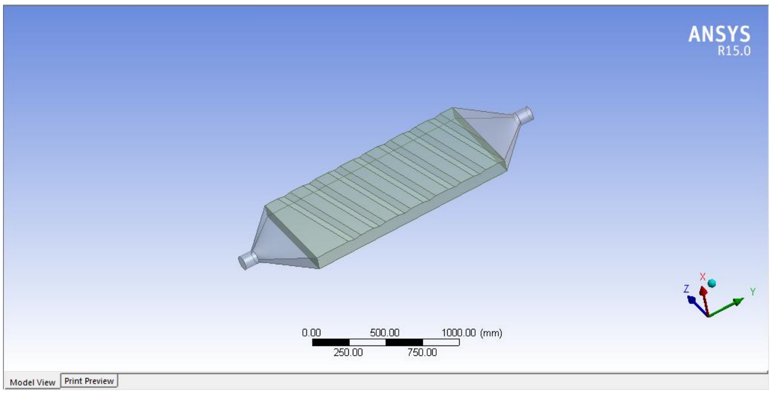

Fig. 4 Design 3 – Inclined zig zag surface

RESULTS :

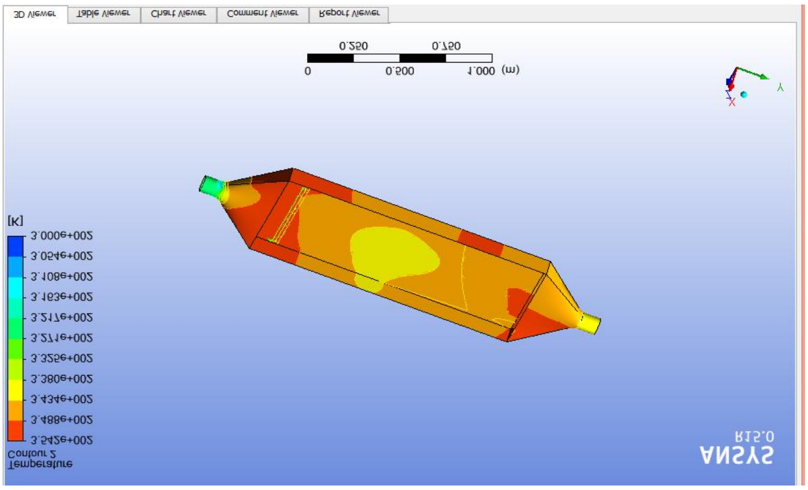

Fig. 5 Result 1 – Plain surface

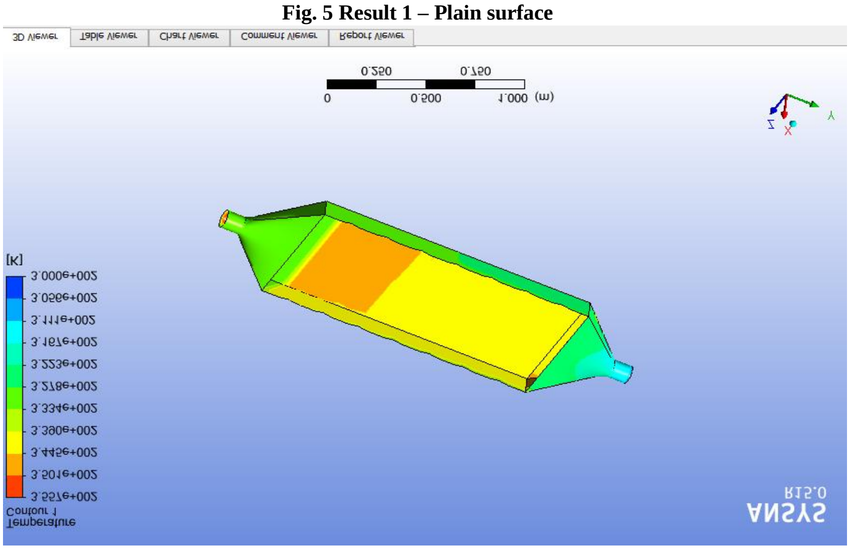

Fig. 6 Result 2 – Normal zig zag surface

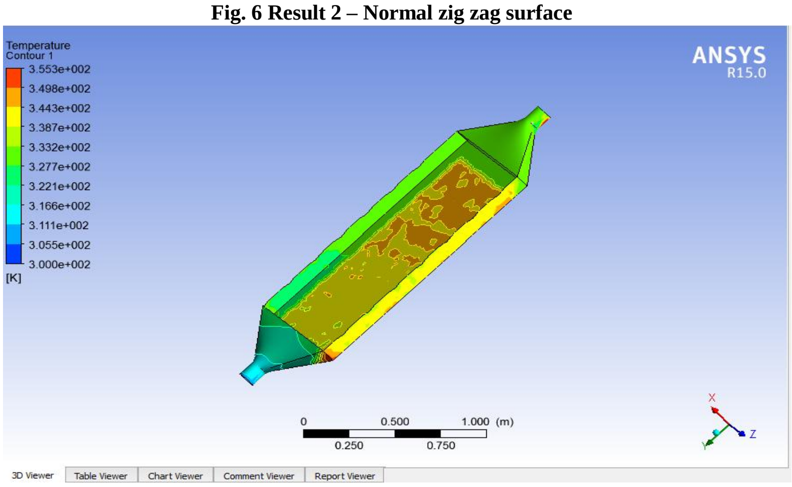

Fig. 7 Result 3 – Inclined zig zag surface

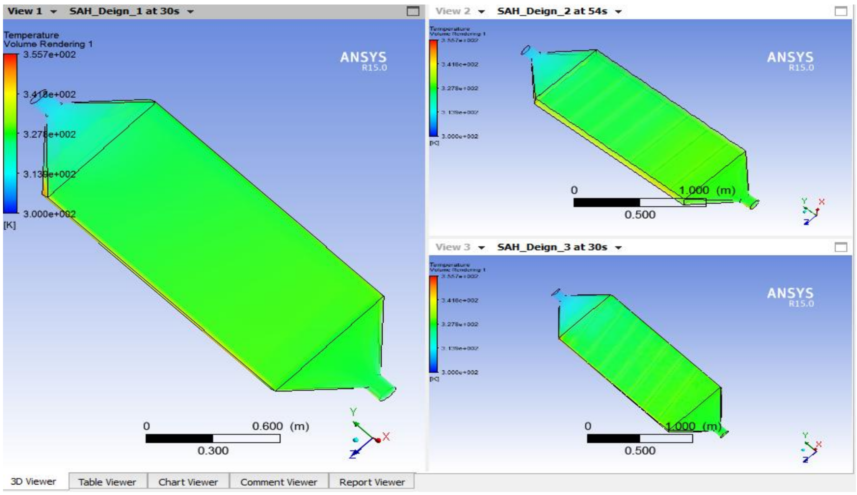

Fig. 8 Result

A Three- dimensional Computational fluid dynamics (CFD) analysis has been carried out to study heat transfer behavior in a rectangular duct of solar air heater having plain surface and artificial roughness. Three surfaces are taken into consideration: plain surface, normal zig zag surface and inclined zig zag surface. By the CFD analysis, we found that efficiency of solar air heater with plain surface rectangular duct is more than the solar air heater with normal zig zag and inclined zig zag surface. It can be seen that the enhancement in heat transfer of the abnormal duct with respect to the smooth duct is less.

CONCLUSION :

A Three- dimensional Computational fluid dynamics (CFD) analysis has been carried out to study heat transfer behavior in a rectangular duct of solar air heater having artificial roughness. There is a good agreement between the experimental and simulated results for outlet air temperatures. The Nusselt number of CFD results has maximum ±8.73 % over experimental results. In this present investigation, a prediction has been conducted to study heat transfer behaviors of a rectangular duct of a solar air heater having plain surface and artificial roughness on the absorber plate. The main conclusions are:

1. There is no doubt that a major focus of CFD analysis of solar air heater is to enhance the design process that deals with the heat transfer and fluid flow.
2. In recent years CFD has been applied in the design of solar air heater. The quality of the solutions obtained from CFD simulations are largely within the acceptable range proving that CFD is an effective tool for predicting the behavior and performance of a solar air heater.
3. Solar air heater with plain surface have better efficiency of heat transfer as compared to abnormal surface.

REFERENCES :

1. P. K. Singh, A. Kumar, P. Mishra, and V. K. Sethi, "Study on Future of Solar Thermal Storage System Using Concentrated Solar Power," Int. Conf. Power Electron. Control Autom. ICPECA.
2. P. Das and C. V.P, "Performance evaluation of solar vortex engine and optimization of number of air entry slots and turbine location," Energy Sources, Part A Recover. Util. Environ. Eff., vol. 00, no. 00, pp. 1–17, 2020
3. P. Guo, S. Wang, Y. Lei, and J. Li, "Numerical simulation of solar chimney- based direct airside free cooling system for green data centers," J. Build. Eng., vol. 32, no. June, p. 101793, 2020.
4. L. Zuo et al., "Numerical analysis of wind supercharging solar chimney power plant combined with seawater desalination and gas waste heat," Energy Convers. Manag., vol. 223, no. August, 2020.
5. Q. Wang, G. Zhang, W. Li, and L. Shi, "External wind on the optimum designing parameters of a wall solar chimney in building," Sustain. Energy Technol. Assessments, vol. 42, no. September, p. 100842, 2020
6. Elsayed and Y. Nishi, "Emergy and sustainability ternary diagrams of energy systems: Application to solar updraft tower," Sustain., vol. 12, no. 24, pp. 1–17, 2020.
7. E. Ö. Yapıcı, E. Ayli, and O. Nsaif, "Numerical investigation on the performance of a small scale solar chimney power plant for different geometrical parameters," J. Clean. Prod., vol. 276, 2020.
8. G. He, "A general model for predicting the airflow rates of a vertically installed solar chimney with connecting ducts," Energy Build., vol. 229, p. 110481, 2020.
9. R. Mehdipour, Z. Baniamerian, S. Golzardi, and S. M. S. Murshed, "Geometry modification of solar collector to improve performance of solar chimneys," Renew. Energy, vol. 162, pp. 160–170, 2020.
10. Clito Oliveira, A.A., Solar chimneys: simulation and experiment. Energy and Buildings, 2000. 32(1): p. 71-79.
11. Khedari, J., B. Boonsri, and J. Hirunlabh, Ventilation impact of a solar chimney on indoor temperature fluctuation and air change in a school building. Energy and Buildings, 2000. 32(1): p. 89-93.
12. Szikra, C., Hybrid ventilation systems. 2002, Budapest University of Technology and Economics.