iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 11 Issue: XII Month of publication: December 2023

DOI: https://doi.org/10.22214/ijraset.2023.57714

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue XII Dec 2023- Available at www.ijraset.com

CFD Analysis of Solar Chimney in Bhopal Latitude & Longitude

Meharban Alam[1], Vijaykant Pandey[2]
[1]M-Tech Research Scholar, Bhabha Engineering Research Institute, Bhopal (M.P.), India
[2]Assistant Professor, Bhabha Engineering Research Institute, Bhopal (M.P.), India

Abstract: In order to assess the system's viability, this paper attempts to theoretically study the solar chimney and design the energy flow. In order to verify the accuracy of the CFD results, the article begins with the creation of a CFD model of the solar chimney and compares it to experimental data. To get larger flow rates and compare three different types of solar chimneys with different heights and widths, the CFD model is typically used. The creation of a simple, precise model of the chimney is made possible by the CFD model, which depicts a constant temperature and velocity inside the chimney. Finally, a modification of solar chimney and other systems put in a structure is simulated, and another physical model for this type of building is constructed, providing a sense of how this solar home behaves in various situations.
Keywords: Solar Chimney, Flow Rate, Temperature, Heat Recovery, Stream Function, Turbulent Dissipation, Velocity Contour, Temperature Contour.

I. INTRODUCTION

Electricity generated today from fossil fuels like natural gas, oil, and coal is environmentally harmful and demonstrates the insufficiency of relying on nonrenewable energy sources. Many developing nations cannot afford these conventional energy sources, and nuclear power, in particular, is viewed as an unacceptable danger in many regions**.** It has been shown that a shortage of energy is linked to poverty and poverty to population booms. Consequently, an ecologicallyfriendly and cost-effective electricity-generating structure has emerged, and it will be likely to do so in the future.

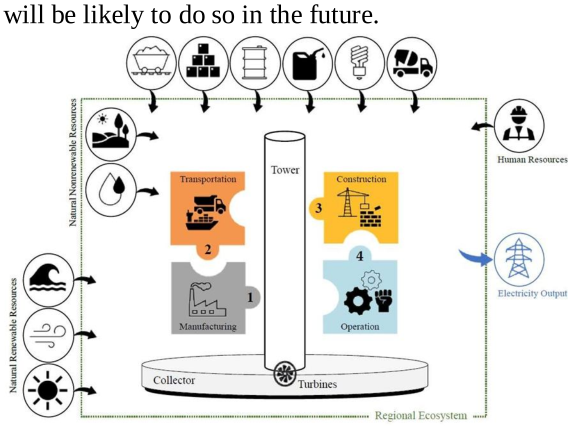

Figure 1. Graphical abstract of energy analysis for the lifecycle of solar updraft towerpower plants.

Solar energy might be feasible to the ever-increasing problem. It is readily available in nature—a non-conventional energy source that has to be aligned to be beneficial to society. Solar power plants are designed to convert solar radiation into electrical energy through a numerical or natural cycle. Few, on either, can store enough energy every day to maintain a stream at night and when energy from the sun is relatively low. This storage's critical capacity is typically far too high to be practical.

II. LITERATURE REVIEW

1) K. Lovchinov et al. conducted extensive wind tunnel experiments and appropriate theoretical research to design and commission 50-kilowatt test plants. This plant was built at Manzanares and had a collector with a diameter of 240 metres and 195 metres. Various features and effects on their research, including:
a) The durability and structural feasibility of various roof coverings on plant performance were investigated—effects of climatic conditions on plant performance.

b) Algorithms for mass flow control have been tested.

c) The operating costs and maintenance requirements were analysed.

Several plant sizes were evaluated using a thermodynamic model, and the dimensions, cost, and efficiency of this plant provided multiple tables. He also studied the system's economic feasibility and found it identical to a traditional power generation system. He explains the findings of his research on the implications of water bags, which enhance ground storage capacity. According to this research, a water layer 0.2 meters deep water layer will level out the daily fluctuations in power output, resulting in a daily peak of roughly 50% of those of a similar plant with no water storage.

2) R. Mehdipour et al. reported a simple evaluation of the Solar Chimney, eliminating temperature and pressure variations due to height and chimney losses. There were suggested expressions for total efficiency.

3) R. Mehdipour et al. derive relevant governing differential equations that characterise Chimney performance. They then presented the findings of research findings on the viability of moderate to large-scale power generating and also rural power generation. They recently studied the implications of geometrical and operational factors on the solar chimney's overall performance.

4) P. Das and C. V.P investigated the validity of several driving potential models for prevailing ambient circumstances surrounding the Solar chimney (particularly elevation-dependent variables). They demonstrate that power production riseswith humidity and that condensation in the chimney can occur.

5) Guo et al. This study for data centres presents a unique solar chimney- based direct airside free cooling (SC-DAFC) technology to conserve electricity. Under specified climate circumstances, a conceptual model of the SC-DAFC system is created to analyse the data centre's internal thermal conditions, ventilation flow rate, and energy recovery. The impacts of increased ceiling height and turbo pressure drop mainly on chilling system performance are investigated using 3D numerical simulations. Calculating the project's present value and dynamic asset recovery time is also part of the economic study. The study demonstrates that under favourable climate situations, the temperature in the SC-DAFC data centre can fulfil cooling needs and that such a project is highly profitable. The SC-DAFC device is both technically and commercially practical, resulting in energy-saving and operational-cost-cutting solutions.

6) Zuo et al. The focus of this research is to examine the impact of operational and structural elements on the WSCPPDW's performance and, therefore, to evaluate the flow field characteristics. The previously suggested mathematical model of WSCPPDW could barely explain the influence of structural parameters and flue gas jet. Utilising CFD ANSYS Fluent, a 3D numerical simulation of WSCPPDW was performed. Simulations with various turbine rotational speeds, nozzle lengths, chimney outlet radiuses, and chimney mixing section lengths were performed to find optimal operational and structural characteristics of WSCPPDW. The findings demonstrate that the flue gas jet can create a high-temperature, high- speed jet region in the chimney, entrain the surrounding airflow. Furthermore, as the rotational speed of the turbine increases, the freshwater yield decreases, and the turbine shaft power increases initially, then declines, peak at 200rpm. The freshwater gain and turbine shaft power increase and then decrease as the nozzle length increases; the freshwater yield and turbine shaft power increase and decrease as the chimney outlet radius increases; as the chimney mixing section length increases, the freshwater gain and turbine shaft power increase and then decline.

7) Ramin Mehdipour, Baniamerian, et al. In this report have two distinct features in comparison with other setups that were used in the prior arts:Because the settings are indoor rather than outside, the environmental factors can be controlled entirely in this research, and a steady-state can be accomplished. Both arrangements can measure the collector's thermal performance and the solar chimney's performance. To analyse the influence of geometry on the actual efficiency of solar vents, several setups were used. The thermal and hydraulic performances of popular forms of solar chimneys are shown to be insufficient in this research, and inadequate thermal behaviour ofcollectors is among the significant causes for solar chimneys' poor power output. Nusselt number (Nu), airflow velocity, and convective heat transfer coefficient (h) increased 1225 percent, 245 percent, and 603 percent, respectively, when the collector form is changed from circular to that of an innovative square shape.

III. 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.

A. *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.

B. *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.

IV. CALCULATION & DESIGNING

The basis of the development of the solar chimney's CFD model is the experimental results taken from the project "Development of Air solar Collector". This project can be found in velocity, temperature, height, and experimental radiation data from the solar chimney.

As the CFD model will be a 2-D model, it only is necessary to take the midplane data from the solar chimney. I have studied six types of different configurations of boundary conditions to find which of these ones obtains a better converge of the problem and less affect the actual conditions.

In this case, has been used a simple model of an asymmetric chimney to reduce the time of simulations. The six different types of configurations are the following

1) The limits of the model are walls with slip conditions and constant Temperature
2) There is a velocity inlet of 0.5 m down in the left wall, an outflow of 0.1 m in the left at the top division, and the rest is a wall with slip conditions. Both velocity inlet and wall have the same constant temperature.
3) except that the velocity is at the top and the outflow in the bottom.
4) The bottom is a velocity inlet, the top is an outflow condition, and the right side is a wall with slip conditions.
5) The bottom is a pressure inlet condition, the top a pressure outlet and right side a wall with slip condition.
6) Where velocity inlet conditions are used has to be careful with the velocity magnitude. In these cases have been used the ratio of Grashof and Reynolds numbers:
7) This number measures the importance of buoyancy forces in a mixed convection flow. When this number is more significant than unity, the buoyancy effects are more substantial than the velocity inlet. To be sure that the velocity inlet does not interfere with the buoyancy forces of the problem, select a ratio equal to or bigger than 100, which corresponds to a velocity of 0.01 m/s.
8) The model used in all the cases is the same. It is an asymmetric chimney with a radiator boundary condition in the bottom of this one with a heat flux of 300 W/m2. The externaltemperature is 300 K (exterior walls and inlets), and the turbulence model is standard k-e with standard wall functions and thermal buoyancy effects.

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue XII Dec 2023- Available at www.ijraset.com

A. Chimney CFD model

Before making the entire model of the chimney, different inlets have been simulated. This action has been made to avoid the recirculation flow that appears in the channel.

First, the solar collector had been replaced for a radiator boundary condition, fixing the heat flow, to study the flow around the chimney.

1) In the first case (without inlet), one can see the recirculation flow at the channel; that is why different inlet configurations have been tested to avoid this reflow. In any case, these configurations do not affect the stack effect, as these are always before the collector.
2) To find an inlet for the chimney, the reason is that this one will join to a ventilationduct, and this reflow hardly appears. Other reasons are the stability of the simulation and the energy balance in the chimney (some of the heat goes out and affects the stack effect).
3) The following pictures show the steps to achieve the inlet for the chimney

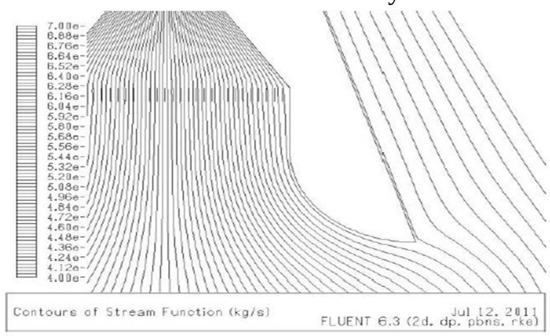

Figure 2 Chimney inletthe selected model to simulate the chimney

4) The boundary chosen conditions before.
5) The inlet selected without slip conditions.
6) The outside of the chimney is entirely different from the inside; the reason is toavoid the reflow of what appears outside the inlet.
7) The solar collector has been changed for 32 fins. One side of the fin will generate a heat flow, simulating the incident radiation; the other side will be adiabatic.
8) The inside (wall and glazing) of the chimney has been considered completely insulated.

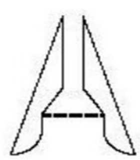

Figure 3 Chimney CFD Model

V. RESULTS

The results of the temperature in the solar collector and the glazing have been compared with the simulations carried out in other work, that show an increment around 30ºC in the solar collector and 12ºC in the glazing, which is similar to the results simulated, 28ºC and 13ªC correspondingly.

The comparison was carried out with eight simulations for each chimney, 4 different widths (h=3m) and 4 heights (w=0.5m).

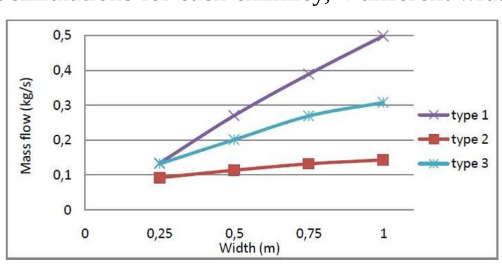

Figure : 4 Graph of Mass flow Vs. Width

It shows the chimney type 1 has a linear mass flow increase with the width; the reason is the area of solar collector increase. Note, if the width is small (w<0.25m), the flow rate of the chimney type 1 will be lower than the other chimneys; that is one of the reasons of this kind of chimneys (type 2 and 3) usually has a width around 0.25 m [1, 2, 7].

The function Pressure Drop Vs. Flow rate serves to know the behaviour of the chimney for different pressures losses and also to compare with fans since the cross between these curves (system and fan) marks the operations point.

A simple control valve was designed to simulate diverse work conditions; this valve consists of two plates situated in the outlet, which are closing from the outside.

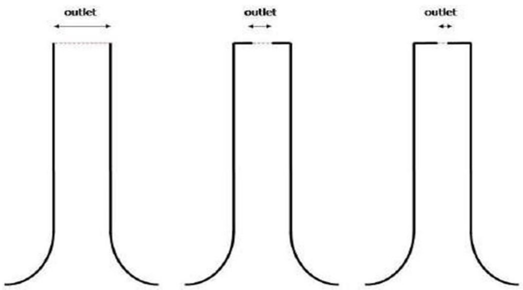

Figure 5: Behaviour of the control valve

Six simulations were carried out for various valve openings, from fully open to 10% of aperture. Lessening the beginning makes the simulation unstable, and the assumptionsof the adiabatic wall are not satisfied because of the increase in air temperature.

To calculate the pressure drop across the chimney, it has been taken the pressure data from 3 lines: one is the symmetric line, the other passes close to the wall (0.05m from the wall), and the other is parallel to these outside from the chimney.

The outside line severs calculated the hydrostatic pressure since Fluent uses a reduction constant (A) for the simulations. Once the hydrostatic pressure is calculated, this value is rested on data from the chimney's lines. Figure shows these values from 80% of valve aperture; except in the inlet and the outlet, the two lines follow a straight line calculated to know the average pressure drop across the chimney.

$\Delta P_{H} = A\rho g \Delta H$

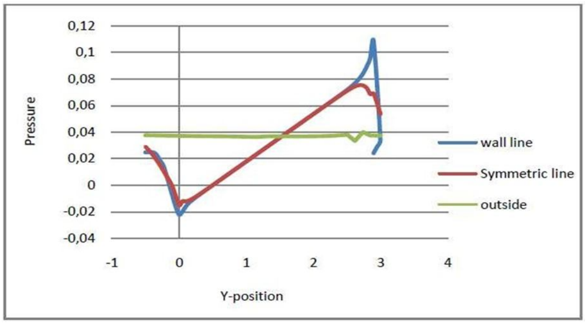

Figure 6 Hydrostatic pressure from the chimney

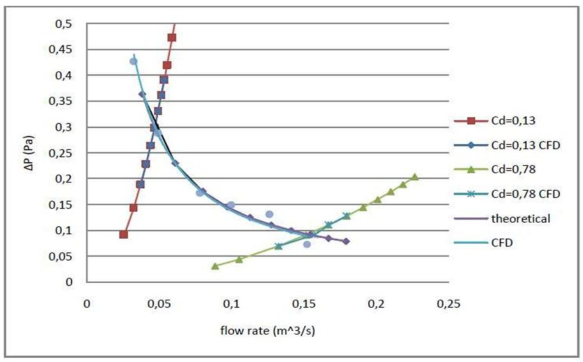

Figure 7 Pressure Vs Flow Rate

1) There is no maximum or minimum, which means that the slop is always negative,and therefore, the operation point is always stable.
2) Meanwhile, there is a lower limit for a stall in the fans. The chimney always cangenerate a flow rate no matter how small it is.
3) The disadvantage is the strong dependence between the flow rate and the pressure inthe chimney when the desire is to have the same pressure drop.

VI. CONCLUSION

This report studies the basic principles of a solar chimney, especially the chimney installed in EPT-lab, utilising as tools commercial CFD software, Fluent, and basic physical models.
The following conclusion could be drawn.

1) A simple CFD model without external radiation is enough to simulate and study the solar chimney proposed in the with reasonable accuracy.
2) The design of the solar chimney allows using simple physical models, knowing its properties (flow rate and temperature) and modelling the size (height andsolar collector area) due to the uniform velocity and temperature appears inside the chimney. Also, these characteristics of the flow make it possible to use allthe knowledge of the standard flues in this solar chimney.
3) The EPT- solar chimney can be used in both types of climates (warmer and cold), and for reasonable height, it is the type of solar chimney that gives the highest flow rate. Only in warmer temperatures and in tall buildings the solar chimney type 3 will provide better results.
4) The chimney's solar efficiency depends on the magnitude of solar radiation and the difference in temperatures between the building and the surroundings. Therefore, only when the temperature difference is near zero, the solar chimney makes sense to use it.
5) The height of the chimney is usually essential to increase the flow rate. Still, when the temperature inside the building is lower than the external, it is more important to increase the area of the solar collector than the height of thechimney.
6) A solar chimney with a heat recovery improves solar efficiency but hinders air exchange. For that reason, only hybrid ventilation systems with the support ofa fan and enough flexibility to change the course of the flow rate should be used with the solar chimney and heat recovery

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.
[13] Gan, G. and S.B. Riffat, A numerical study of solar chimney for natural ventilation of buildings with heat recovery. Applied Thermal Engineering, 1998. **18**(12): p. 1171-1187.
[14] I. Chand and P. Bhargava, Studies on design and performance of a non- conventianal system of natural ventilation in buildings. Solar and Wind Technology 1990.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue XII Dec 2023- Available at www.ijraset.com

[15] Ong, K.S., A mathematical model of a solar chimney. Renewable Energy,2003. **28**(7): p. 1047-1060.
[16] Sakonidou, E.P., et al., Modeling of the optimum tilt of a solar chimney for maximum air flow. Solar Energy, 2008. **82**(1): p. 80-94.
[17] Dai, Y.J., et al., Enhancement of natural ventilation in a solar house with a solar chimney and a solid adsorption cooling cavity. Solar Energy, 2003. **74**(1):p. 65-75.
[18] Agung Murti Nugroho and M.H.b. Ahmad, POSSIBILITY TO USE SOLAR INDUCED VENTILATION STRATEGIES IN TROPICAL CONDITIONS BY COMPUTATIONAL FLUID DYNAMIC SIMULATION. 2005: p. 8.
[19] Åke Blomsterberg, Åsa Wahlström, and M. Sandberg, Refurbishment of the Tångaschool. 2002, INTERNATIONAL ENERGY AGENCY, ENERGY CONSERVATION IN BUILDINGS AND COMMUNITY SYSTEMS.
[20] Daivid Etheridge and M. Sandberg, Pressure Differences generated bytemperature, in Building Ventilation: Theory and Measurement 1996. p. 28-29.
[21] http://www.brynmawr.edu/geology/206/gruenstein2.htm. SolarTower Technology.
[22] http://en.wikipedia.org/wiki/Cubic_function. Cubic function.
[23] Carmen Viñas, et al., Analysis of the Behavior of a Turbulent Jet Indoor. InformacioTecnológica, 2008. 19 Nº5: p. 9.
[24] ANSYS, I., ANSYS FLUENT 12.0 Theory Guide. 2009. 816.
[25] Shao, L., S.B. Riffat, and G. Gan, Heat recovery with low pressure lossfor natural veltilation. Energy and Buildings, 1998. **28**(2): p. 179-184.