Turkish Online Journal of Qualitative Inquiry (TOJQI)
Volume 11, Issue 4, December 2020: 2336-2342
DOI: 10.53555/tojqi.v11i4.10068

Research Article

The latest Advancements in HVAC Technology and their Potential Benefits.

Mohammed Hameeduddin Haqqani[1]*, Mohammed Azizuddin[2], Syed Shuibul Qarnain[3]

Abstract:

The main objective here is to study the performance of the solar chimney in the HVAC system. The main parameters of the study will be the temperature distribution, pressure distribution, and velocity of air within the system and chimney. The flow inside the chimney will also be a core part of the study. Based on governing equations numerical simulations are performed within the system. Volumetric flow rate of the chimney is the main parameter that will demonstrate its ventilation effectiveness. From the study, the pressure contours results, it was observed clearly that for a normal conventional room when it integrated with solar chimney its pressure reduces from 3.67 Pa to 1.19 Pa and its temperature increases from 301 K to 308 K because of chimney draught effect. In the Velocity contour results, it was clearly observed that HVAC system without having solar chimney its fluid velocity is very slow as compared to the HVAC system when it is integrated with solar chimney. And also at inlet of chimney fluid enters into the chimney with wider angle due to chimney effect. Velocity remains zero due to no slip conditions at the walls.

Keywords: HVAC, Cooling, CFD, Energy-efficient strategies, Buildings, solar chimney effect, ANSYS.

Introduction

A solar chimney is a natural passive ventilation system that absorbs solar energy from the sun to release it later to the airflow contained in the vertical flat plate channel. After that, the air temperature rises, and the airflow tends to move upwards. Then, the air removes hot air and dilutes contaminant concentration from inside, and it intentionally supplies fresh air into rooms. The passive design approach can help to maintain comfortable thermal conditions in buildings and reduce energy consumption by minimizing the use of auxiliary mechanical or electromechanical air-conditioning systems. In particular, the implementation of solar chimneys for cooling increases energy efficiency in buildings located in warm climates by decreasing electricity bills and CO_2 emissions [1-7].

In this work, natural ventilation flow through a three-dimensional but real-sized room is investigated numerically, using commercial CFD software ANSYS. An energy-based analysis in rooms or passive systems can be conducted by using computational fluid dynamics (CFD), A detailed CFD analysis of fluid flow and energy transport plays an important role in thermal comfort and air quality evaluations.

HVAC systems are milestones of building mechanical systems that provide thermal comfort for occupants accompanied by indoor air quality. HVAC systems can be classified into central and local systems. Primary HVAC equipment includes heating equipment, ventilation equipment, and cooling or air-conditioning equipment. There are other ways of achieving cooling as discussed by many

[1]*Research Scholar, Department of Mechanical Engineering, Career Point University , Kota-325003, hameed6162@yahoo.com
[2]Department of Mechanical Engineering, Deccan College of Engg &Tech, Hyderabad -500001
[3]Department of Mechanical Engineering, Kalasalingam Academy of Research and Education, Krishnankovil-626126

authors in the review section on Vortex tubes and its usage for spot cooling application.
The present paper focuses on the increase in productivity, buildings, and equipment Durability, filtration of dust, chemical containments, and odors through chemical and carbon-activated filters, and inside conditions of 20-23^{0}C & 50-60 % Relative Humidity is most Comfortable, and purity of air, increase life & Health.

METHODOLOGY

Over the past few decades, ANSYS Fluent has become the gold standard for CFD. Whether you analyze basic heat transfers or fluid flows or work with complex geometries that have transient reacting flows, ANSYS Fluent should be an integral part of your product optimization and designing process. ANSYS has helped solve complicated design challenges and engineer products that are limited only by imagination. ANSYS Fluent is CFD software that is particularly used for fluid flow modeling and heat transfer. ANSYS Fluent also has broad physical modeling capabilities that are needed for fluidflow, heat transfer, turbulence and reactions for industrial applications. To create a standalone fluent system in ANSYS, click over the Fluid Flow (Fluent) in the Analysis Systems. Once selected, drag it to the project schematics and drop it.

Each Fluent system has a set of five components. Each component has a specific set of functions and is necessary for them to be set correctly for the complete fluent system to run correctly. Each component has its significance.
Steps performed on ansys workbench are Geometry, Mesh, Setup, Solutions, and Results. ANSYS can import all kinds of CAD geometries (3D and 2D) from different CAD software and perform simulations, and also it has the capability of creating one effortlessly. ANSYS has inbuilt CAD developing software like Design Modeler and Space Claim which makes the workflow even smoother.

1) ANSYS can perform advanced engineering simulations accurately and realistically through its variety of contact algorithms, time-dependent simulations, and nonlinear material models.
2) ANSYS can integrate various physics into one platform and perform the analysis. Just like integrating a thermal analysis with structural and integrating fluid flow analysis with thermal and structural, etc.,
3) ANSYS now has featured its development into a product called ANSYS AIM, which is capable of performing multi-physics simulations. It is a single platform that can integrate all kinds of physics and perform simulations.
4) ANSYS has its customization tool called ACT which uses python as a background scripting language and is used in creating customized user-required features in it.

GEOMETRY CREATION:

When a new simulation is started, the first step to do is to set the geometry that is wanted to be simulated. First of all, it’s important to check the properties of the geometry that can be ruled in 3D symmetry, as it would be easier and faster to obtain the results afterward. The three-dimensional geometry of MSC is developed in design modular (DM) associated with ANSYS 14.0 workbench. It is necessary to define all zone and domains before starting the mesh generation. So, to do it, ANSYS offers a wide range of analysis systems tools, but it has used fluid flow (Fluent) because it is which fits better.
In this work, we are creating the Geometry of the conventional HVAC System as shown in Fig 1 HVAC system integrated with a solar chimney as shown in Fig 2 created based on ANSYS software.

COMPONENTS CONSIDERED WHILE CREATING THE GEOMETRY:

COMPUTERS-4; LIGHTS-6; PERSONS-4; WALLS-4; CHIMNEY-1; TABLES-4 ; DOOR -1; CASSETTE -1; ROOM-1;

DIMENSIONS OF GEOMETRY:

1) Room Dimensions:

Table 1: Dimensions of Room

Name of distance	Distance in (mm)
Horizontal	7500
Vertical	3500
Depth	5500

2) Door Dimensions:

Table 2 : Dimensions of Door

Name of distance	Distance in (mm)
Horizontal	1500
Vertical	2000
Distance from Wall	2000

3) Lights Dimensions:

Table 3: Dimensions of Lights

Name of distance	Distance in (mm)
Horizontal	500
Vertical	500

4) Table Dimensions:

Table 4: Dimensions of Table

Name of distance	Distance in (mm)
Horizontal	700
Vertical	1800

5) Computer Dimensions:

Table.5: Dimensions of Computer

Name of distance	Distance in (mm)
Horizontal	850
Vertical	200

6) Human Dimensions:

Table 6: Dimensions of Humans

Name of distance	Distance in (mm)
Horizontal	250
Vertical	800

7) Cassette Dimensions:

Table 7: Dimensions of Cassette

Name of distance	Distance in (mm)
Horizontal	950
Vertical	950

8) Inlet Dimensions:

Table 8: Dimensions of Inlet

Name of distance	Distance in (mm)
Horizontal	450
Vertical	350

9) Outlet Dimensions:

Table 9: Dimensions of Outlet

Name of distance	Distance in (mm)
Horizontal	900
Vertical	350

10)Distance between Inlet& Outlet of Chimney:

Table 10: Dimensions of Chimney

Name of distance	Distance in (mm)
Distance between inlet & outlet of Chimney	5000

As it was observed that the Geometries are created below in thefig 1 based on dimensions given in the table by using ANSYS Fluent Software.Once the Geometry has been already created the next step is starting the meshing.After doing the geometry, it precedes meshing operation. The mesh influences the accuracy, convergence and speed of the solution. maximum values are 1:35 for the aspect ratio and 0.95 for the skewness. Both of these parameters are sufficient and the quality of the mesh is adequate for the case study.In this project we have already created the geometries of HVAC system integrated with solar chimney & Conventional HVAC System without Solar Chimney as you can see in the fig 1,now we are meshing these geometries by using ANSYS meshtools.

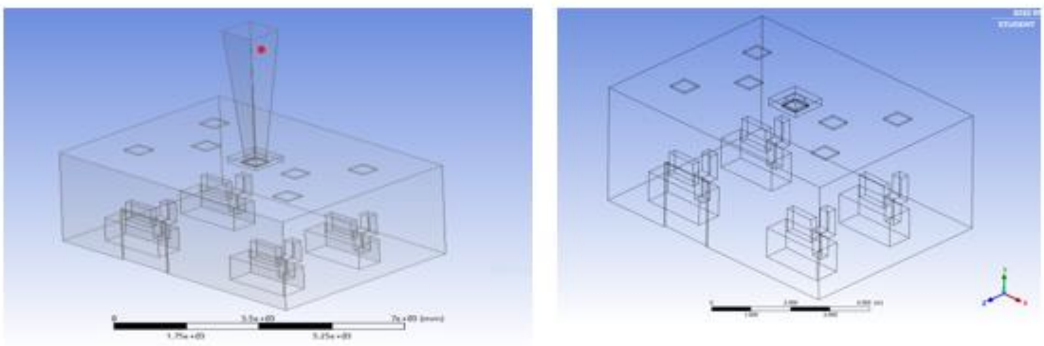

Fig 1: Shows the Geometry of HVAC System Solar Chimney

In this study, the Navier-Stokes equations are solved numerically .For turbulence modeling, the K-Epsilon Realizable model is used. Also the ideal gas model is used for the calculation of the air density at different temperatures. The coupled method is used for coupling the velocity and pressure equations in ANSYS Fluent software. The discretization of the equations of pressure, momentum, turbulent kinetic energy, turbulence loss, and energy is set as the second order. For the incompressible flow simulation, continuity equation, and the momentum equation considering the gravity force.

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + 2\mu_t E_{ij}E_{ij} - \rho\varepsilon$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_j}(-\rho\overline{u'_i u'_j}) + f_i.$$

Momentum Equation:

The Reynolds-averaged Navier Stokes equations governing the flow, take the form, the Reynolds stresses are related to the mean velocity gradients by employing the Boussiness hypothesis and fi is the external body force in the direction (N/m^3) [8-11].

Energy Equation:

For the temperature distribution calculations, the energy equation can be written as

$$\frac{\partial}{\partial x_j}[u_j(\rho E + P)] = \frac{\partial}{\partial x_j}\left[\left(\lambda + \frac{c_p\mu_t}{pr_t}\right)\frac{\partial T}{\partial x_j} - \sum_j h_j J_j\right] + S_h,$$

Where k is the thermal conductivity of the fluid, J_j is the diffusion flux of species j, and $S_h$is any volumetric energy source.

$$\frac{\partial\rho}{\partial t} + \nabla.(\rho u) = 0$$

Continuity Equation:

Turbulence kinetic energy equation:

RESULTS & DISCUSSIONS

In this work we have conducted a numerical simulation based analysis for HVAC room or passive systems with solar chimney and compared with the passive system or HVAC room without solar chimney conducted by using computational fluid dynamics (CFD).A detailed CFD analysis of fluid flow and energy transport plays an important role in thermal comfort and air quality evaluations. And the obtained results are compared between HVAC System integrated with Solar Chimney and HVAC System without Solar Chimney [12-14]. Here the results are compared based on Velocity of aircontour, Mass flow rate, Pressure Contour, Temperature contour by creating the 3 planes within the system and the obtained results are:

Pressure contour of HVAC System without Solar Chimney:

$$\rho\left[\frac{\partial E}{\partial t} + u\frac{\partial E}{\partial x} + v\frac{\partial E}{\partial y} + w\frac{\partial E}{\partial z}\right] = \nabla.(k\nabla T) - \nabla.p\vec{v} + Q_v + Q_g$$

Here in this pressure contours results, it was observed clearly that for a normal conventional room when it integrated with solar chimney its pressure reduces from 3.67 Pa to 1.19 Pa because of chimney draught effect. And also at the outlet of chimney its pressure equals with the atmospheric pressure.

Temperature contour of HVAC System without Solar Chimney and with Solar Chimney:

Here in this temperature contour result, it was observed clearly that for a normal conventional room when it integrated with solar chimney its temperature increases from301 K to 308 K because of chimney effect. And it was observed that in the chimney velocity of fluid increases.

Velocity contour of HVAC System without Solar Chimney:

Here in this Velocity contour result,it was clearly observed that HVAC system without having solar chimney its fluid velocity is very slow as compared to the HVAC system when it is integrated with solar chimney. And also at inlet of chimney fluid enters into the chimney with wider angle due to chimney effect. Velocity remains zero due to no slip conditions at the walls.

Mass flow rate contour of HVAC System without Solar Chimney and with Solar Chimney

The mass flow rate of air is observed in three different planes of HVAC system. Here in this mass flow rate contour result, it was noticed clearly that for a normal conventional room when it integrated with solar chimney its mass flow rate increases due to chimney draught effect.

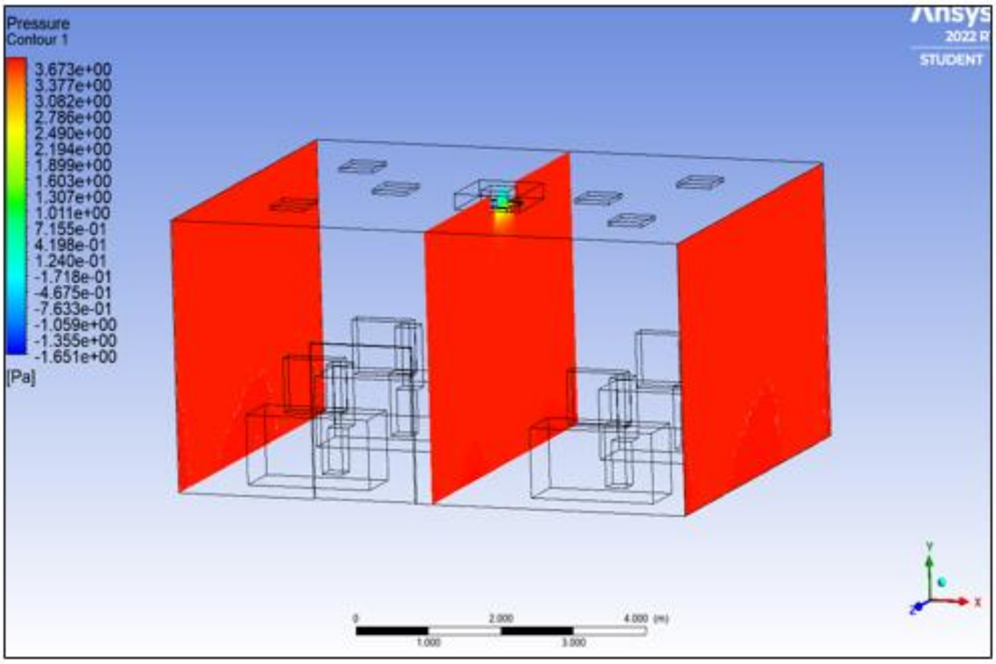

Fig 2: pressure contour of HVAC System without Solar Chimney

CONCLUSION

In this work, natural ventilation flow through a three-dimensional but real sized room is investigated numerically, using commercial CFD software ANSYS. An energy-based analysis in rooms or passive systems can be conducted by using computational fluid dynamics (CFD). Numerical study of the performance of the solar chimney in the HVAC system was done. The main parameters of study are the temperature distribution, pressure distribution, and velocity of air within the system and chimney.

In the pressure contours results, it was observed clearly that for a normal conventional room when it integrated with solar chimney its pressure reduces from 3.67 Pa to 1.19 Pa and its temperature increases from 301 K to 308 K because of chimney draught effect. In the Velocity contour results, it was clearly observed that HVAC system without having solar chimney its fluid velocity is very slow as compared to the HVAC system when it is integrated with solar chimney. And also at inlet of chimney fluid enters into the chimney with wider angle due to chimney effect. Velocity remains zero due to no slip conditions at the walls.

References:

Enteria N, Mizutani K. The role of the thermally activated desiccant cooling technologies in the issue of energy and environment. Renew Sustain Energy Rev 2011;15:2095–122.

Haqqani, Mohammed Hameeduddin, & Md Azizuddin. "Ranque-Hilsch Vortex Tube and its usage in cooling system–A Review." International Journal of Research [Online], 6.10 (2019): 636-646. International Journal of Research; Vol 6, No 10 (2019): Vol-6-Issue-10-September-2019.

G L Ranque, Method and apparatus for obtaining from a fluid under pressure two currents of fluids at different temperatures, U S Patent No. 1952281,1934.

Cardemil JM, Colle S. A general model for evaluation of vapor ejectors performance for application in refrigeration. Energy Convers Manage 2012;64:79–86.

Md. Absar Alam, Rajan Kumar, Anil Singh Yadav, Ranjeet Kumar Arya, V.P. Singh, Recent developments trends in HVAC (heating, ventilation, and air-conditioning) systems: A comprehensive review, Energy Conversion and Management, 77, 2014,738-754.

Wandong Zheng, Jingfan Hu, Zhaoying Wang, Jinbo Li, Zheng Fu, Han Li, Jakub Jurasz, S.K. Cho u, Jinyu Yan, COVID-19 Impact on Operation and Energy Consumption of Heating, Ventilation and Air-Conditioning (HVAC) Systems, Advances in Applied Energy,3, 2021, 100040.

Joint Technical Committee CH-026, Safety in Laboratories, Council of Standards Australia and Council of Standards New Zealand, Australia/New Zealand standard TM safety in laboratories, part 3: Microbiologic safety and containment: AS/NZS 2243.3: 2010. https://store.standards. org.au/product/as-nzs-2243-3-2010, 2022.

World Health Organization, Laboratory biosafety manual fourth edition. https:// apps.who.int/iris/ rest/bitstreams/1323419/retrieve, 2022.

World Health Organization, Laboratory design and maintenance. https://apps. who.int/iris/rest/ bitstreams/1323436/retrieve, 2022.

European Committee For Standardization, Biotechnology-Large-scale process and production-Plant building according to the degree of hazard: BS EN 1620:1997, British Standards Institution, 389 Chiswick High Road London W4 4AL, 1997.

General Session of the European Commission for the Control of Foot-and-mouth Disease (EuFMD) at the Food and Agriculture Organization (FAO) of the United Nations, Minimum standards for laboratories working with FMDV in vitro and in vivo. http://www.fao.org/ag/ againfo/ commissions/docs/genses38/Appendix_ 10.pdf, 2022 (accessed 5 May 2022).

G.P. Cao, G.Q. Cao, Y.G. Zhang, Z.G. Chen, P. Gao, B.B. Liu, H.Q. Li, Comparison of test requirements of biosafety laboratory standard systems at home and abroad, J. HV&AC 52 (2022) 16–22.

H. Xia, Q. Han, Z.M. Yuan, Worldwide developmental situation of arthropod containment and its inspiration, China Trop. Med 21 (2021) 804–808.

M.S. Ramirez, M.E. Tolmasky, Aminoglycoside modifying enzymes, Drug Resist. Updat. 13 (6) (2010) 151–171.

E. van Duijkeren, C. Schwarz, D. Bouchard, B. Catry, C. Pomba, K.E. Baptiste, M.A. Moreno, M. Rantala, M. Ružauskas, P. Sanders, C. Teale, A.L. Wester, K. Ignate, Z. Kunsagi, H. Jukes, The use of aminoglycosides in animals within the EU: development of resistance in animals and possible impact on human and animal health: A review, J. Antimicrob. Chemother. 74 (9) (2019) 2480–2496.

C.J. von Wintersdorff, J. Penders, J.M. van Niekerk, N.D. Mills, S. Majumder, L.B. van Alphen, P.H. Savelkoul, P.F. Wolffs, Dissemination of antimicrobial resistance in microbial ecosystems through Horizontal Gene Transfer, Front. Microbiol. 7 (2016) 173.