九州大学学術情報リポジトリ
Kyushu University Institutional Repository

Simulating the Fluid Flow via Ceiling Diffuser Using OpenFOAM

Manoj Kumar Gupta
Department of Mechanical Engineering, Hemvati Nandan Bahuguna Garhwal University

https://doi.org/10.5109/6781100

出版情報：Evergreen. 10 (1), pp.404-411, 2023-03. 九州大学グリーンテクノロジー研究教育センター
バージョン：
権利関係：Creative Commons Attribution-NonCommercial 4.0 International

Simulating the Fluid Flow via Ceiling Diffuser Using OpenFOAM

Manoj Kumar Gupta
Department of Mechanical Engineering, Hemvati Nandan Bahuguna Garhwal University, Srinagar, Uttarakhand, India.

*Author to whom correspondence should be addressed:
E-mail: gupta_291@rediffmail.com

(Received July 12, 2022; Revised January 24, 2023; accepted January 24, 2023).

Abstract: HVAC (heating ventilation and air conditioning) deals with cooling and heating buildings and providing clean conditioned air for indoor environments such as commercial office spaces, cinemas, and hotels to name a few. Diffusers help to effectively distribute air coming from HVAC ducts into indoor environments. Understanding the flow distribution from the diffuser is of great significance as it helps in locating the optimal location for the diffuser as well helps in designing a better diffuser. With the advent of high-performance computing, numerical simulation is mainly being used to predict airflow via Computational fluid dynamics (CFD). OpenFOAM is an open-source CFD solver written in C++ for solving fluid dynamics problems. In this work, a free architecture design tool called PCON was used to create the geometry of the office/home along with the furniture. OpenFOAM was used to create a mesh and simulate the flow through a square ceiling diffuser and understand the Coanda effect which is created by such a diffuser. Open source code called Paraview was used to analyze the CFD simulations and the results were reported in this paper. In central A/C systems, Freon has been used for decades. Freon, just like R12, has been linked to environmental damage. This work shows how diffusers can help with the efficient distribution of conditioned air in closed spaces. With the efficient distribution of air, Freon would be used less and in turn, more efficiently in a given space. This could prevent environmental damage to some extent.

Keywords: CFD , Diffuser , OpenFOAM, snappyHexMesh

1. Introduction

An Air diffuser is one of the visible parts of an air conditioning system. In HVAC the air travels through the duct and enters the room via diffusers. Diffuser's primary use is to ensure even distribution of conditioned air throughout a room. The diffuser may be of Round diffusers, Square ceiling Diffuser, and Swirl Diffuser as shown in Fig.1.The round diffusers are also known as circular ceiling diffusers. These diffusers provide horizontal and uniform airflow patterns. Some of the traits of round diffusers are high aspiration rate, large volumes of air that can be handled, not too noisy, and preferable to be installed in open areas like supermarkets, airport lobbies, etc. The square ceiling Diffuser is mainly used in low and medium-pressure ventilation systems. It can be used in supply, return and exhaust airflow cases. This type of diffuser discharges air horizontally and it travels along with the ceiling (Coanda effect). This results in proper mixing of room air with the conditioned air coming out of the diffuser. This type of diffuser allows the handling of large quantities of air at high-temperature differentials. It allows 1, 2, 3 or 4 ways air flow. The swirl diffuser diffusers create a swirl of air. This airflow results in reduced air velocity. The temperature difference between the supply air and the room air will also be reduced. These diffusers provide good mixing of air and proper room ventilation with their large volume flow rates.

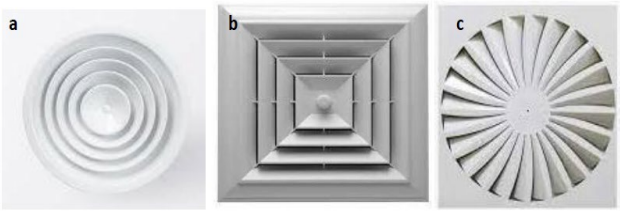

Fig. 1: (a) Round ceiling diffuser (b) Square ceiling Diffuser (c) Swirl ceiling diffuser

ASHRAE [1)] (The American Society of Heating, Refrigerating and Air – Conditioning Engineers) stated the requirement of a proper air distribution process inside

a room to provide good thermal conditions and comfort criteria for occupants. Einberg et.al [2], discussed the results of CFD simulation modeling of multi-cone diffusers for industrial places. Data from experimental procedures were compared with CFD simulations, with air velocity measured using ultrasonic sensors. This study showed that with the standard k-ε model, the CFD simulations accurately predicted the non-isothermal airflow around the diffuser. Sun et.al [3], created CFD models of a room and a supply air diffuser. Effects of various factors on temperature distributions and airflow patterns were examined. Simulations were done for different outdoor and indoor conditions to evaluate the cooling and heating of the room. The results of the 2-D diffuser model showed that the discharge air angle and airflow patterns in the near-flow region are determined by the offset and edge of the diffuser. Buoyancy also plays an important role in judging airflow patterns in a room. The Coanda effect allows air to flow along the ceiling, provided that the heating element has small exit angles. It was concluded that one diffuser provides better air temperature uniformity in the room than two operating diffusers. Halim et. al [4] discussed the impact of a face bypass damper on an oversized air conditioning system using building energy simulation software.They found that an oversized air conditioning system increases relative humidity. The simulation result provides an optimum-sized air conditioning system. The incorporation face bypass damper in the air condition system reduces the overall relative humidity. Kokash et.al [5] performed 3-D CFD modeling using different diffuser designs to evaluate temperature and air velocity distributions in room space of 4m (W) x 4m (L) x 3m (H). Both heating and cooling conditions were considered. Multi holes diffuser was compared with a Multi-cone diffuser (30° cone angle) and a simple opening in the ceiling with no diffuser case. It was found that a multi-cone diffuser has a higher maximum velocity at the diffuser exit due to its smaller cross-section area. The temperature fields indicate that a Multi-hole diffuser having 40% reduced outlet surface area has the best cooling results while 60% reduced outlet surface area has the best results for heating. On the other hand, Multi-cone diffusers with a 30° cone angle provided better uniform temperature for cooling while 40% of Multi-hole diffusers had better air mixing. Although the Multi-hole diffuser ejecting horizontal jets, is desirable for cooling due to the Coanda effect, it is not ideal for heating because of stratification due to hot air being stuck to the ceiling. This implies that diffusers with large inclination angles towards the ceiling should be avoided due to poor mixing results. The ADPI for cooling was calculated and the results showed that a Multi-cone diffuser is a better choice for cooling. The VTAH results on the other hand showed that Multi-hole diffusers lead to better air mixing through the Coanda effect. However, on changing the outlet flow angle from 0° to 30° in the case of a Multi-hole diffuser, HVAC results worsened as the diffuser lost its Coanda effect and leads to poorer air mixing. The computational simulations for different inlet velocities (0.5 and 4 m/s) were performed. The results showed that with lower inlet velocities, cooling performance deformed for both Multi-cone and Multi-hole diffusers. It also, increase inlet velocity led to better uniform temperature for both Multi-hole and multi-cone diffusers. Kotani et.al [6] performed a CFD study on a multi-cone diffuser. Using the standard turbulence model, the turbulent kinetic energy, length, and velocity were measured. The airflow pattern is complicated around the multi-cone diffuser due to two varying flows in it (radial and axial). The study compared the experimental and computational findings, concluding that both were in good practical agreement with each other. Aziz et al [7] did a numerical and experimental study on the square cone, round, and cone vortex diffusers. Comfort criteria, airflow nature, and indoor air quality were examined in the study. The standard turbulence model and energy equation were solved. Compared with other diffusers, the square cone ceiling diffuser produced the maximum fresh air throw. The highest energy consumption and smallest ventilation effectiveness were also noted in the square cone ceiling diffusers. The scale precipitation in the cooling tower is one of the major issues because it decreases the cooling tower's thermal efficiency. The ozone will be injected into the cooling water to reduce the scale of precipitation growth. Alhamid et. al [8] studied the ozonation effect on cooling tower performance and water quality. The result showed that the effectiveness of the cooling tower improve by 8.74 %. Liu et.al [9] compared the impact of the design of a multi-cone diffuser with that of a complex radial vane air diffuser, based on thermal comfort inside the building. Parameters like vertical temperature and Air Diffusion Performance Index were considered in the study. The study showed that radial vane diffuser creates a better cooling effect with the help of the Coanda effect (air jets cling longer to the ceiling). Finally, it was concluded from the study that the multi-cone diffuser was a better overall performer among the two. Jaszczur et.al [10] performed a study of the air stream generated by the square ceiling diffusers. An experimental and numerical study of a single square ceiling diffuser was performed. AL-DV aluminum diffusers having fixed vanes and capable of producing a direct air outflow were designed for experimental analysis. The numerical analysis of the square ceiling diffuser's airstream was done using Star-CCM+ software. The air is considered to be Newtonian and incompressible for CFD analysis. The mesh consisted of 2.8 Million polyhedral cells. The turbulence model was considered for conducting the simulation of the turbulent character of air. It was found that the range of the stream and its shape (dispersed or focused) can be changed. The air flow velocity distribution is initially shaped by the

diffusers themselves. They concluded that knowledge of air currents created by diffusers is important for the proper design of ventilation systems. Awwad et al. [11] laid stress on studying the characteristics of supply air outlets for understanding their effects on airflow distribution inside rooms. Ceiling air diffusers serve many purposes within a room to supply/extract airflow to/from a desired space to provide uniform diffusion patterns in any desired directions to ensure effective mixing of conditioned and entrained air within the room. Computational fluid dynamics (CFD) is used to study the behavior of dynamic fluids (fluids in motion) and solve fluid flow-related problems with the help of computational power, using numerical methods. Venas et. al [12] examined the usage of ceiling-mounted diffusers to provide a heating effect during winters. Several CFD simulations were done for the analysis. It was found that the air exchange efficiency was quite low when using excessive temperature in a situation without internal heat load. However, the ventilation efficiency was increased by adjusting the internal heat loads, which represent low-level heat sources like laptop computers, people, etc. This sort of ventilation for heating favors peak loads i.e. when the building is occupied, additional heat is possible. Farizi et. al [13] studied the effect of moisture transfer on the thermal performance of an alternating-current heat recovery ventilator through the theoretical model.The fiding of the study show that the enthalpy efficiency of alternating-current heat recovery ventilator increases as the accumulated mass of condensate increases due to the latent heat. Furthermore, the length of the thermal storage unit increases, and the airflow rate decreases [27]. Einberg [14] used a a computer model for the design of a ventilation system that takes into account all aspects of room ventilation. The aim of the work was to investigate indoor air quality, air diffusion and comfort issues using CFD. According to Gery, literature, measurements and simulations showed that locations with low air velocity conditions get contaminated with particles. This is why airflow simulations were performed to test different designs. All simulations showed that air distribution to the space is necessary. It helps in finding the concentration of contaminants, and the temperature of the occupied zone and acts as a variable for energy efficiency via the usage of airflow rate. Savanti et. al and Bahar et. al [15,28] investigated the effect of ventilation on indoor air quality and air exchange rate using SPSS software. They found that a lack of natural ventilation reduces the rate of air exchange, which affects air quality. In addition, air movement and volumetric air flow have a significant relationship with air exchange rate. Proper ventilation can reduce the temperature and relative humidity in the room. Well-designed building ventilation has a significant impact on the indoor quality of air and reduces energy consumption.

Han et.al [16] used the Bayesian Markov chain Monte Carlo algorithm for smart ventilation to reduce energy consumption in buildings. It is occupant-based demand-controlled ventilation. It controls the outdoor airflow rate in real-time according to the estimated number of occupants. Mijorski et. al [17] performed CFD based analysis of thermal comfort in a ventilated room. In the study, the thermal comfort of occupants in an office was evaluated for two different cases created by two different positions of air supply devices in the office. This study concluded that the location of the air supply device influences the thermal comfort of occupants in the room. Thermal comfort in the occupied zone of an indoor environment depends on the room airflow to a certain extent. Mogra et. al [18] evaluated the importance of thermal comfort in classrooms by assessing the air velocity and temperature with the help of the CFD technique. They described thermal comfort as a key factor that affects the productivity of people in a room. A CFD model was created for the positioning of the air conditioning system. It was originally placed in the center and simulations were done. Later, it was placed diagonally. The CFD simulation results showed that the diagonally placed AC system provided a better overall distribution of air compared to the centrally placed model. Alessandro et. al 19) stressed the importance of DCV (diffuse ceiling ventilation) to address thermal comfort requirements and internal air quality of buildings. In their study, a performance comparison between non-continuous and continuous perforation distribution was conducted using a CFD model. The CFD analysis showed that the non-continuous diffuser doesn't create any negative effect on the system. Both the perforation setups created different air distributions in the room, but draft discomfort due to airflow velocity wasn't experienced in either of the cases. Xu and Niu [20] described a method for characterizing vortex air diffusers for CFD simulation of room airflow. The simulated results showed that the velocity distribution pattern around the diffuser zone is very complicated and a vertical temperature difference exists in the room under the influence of buoyancy. Prakash et al. [21] discussed CFD analysis of flow through a conical exhaust diffuser. A diffuser with a half cone angle of 70 was found to provide maximum static pressure recovery and exhibit better performance. Uraijaree et al. [22] studied the effects of designs of ceiling diffusers with various concentric circles on airflow patterns using computational fluid dynamics simulations software. It was found that a ceiling diffuser with five concentric circles improves airflow quality at the outlet by 13.57% which reduced shortened freezing time. Rao et al. and Herlambang et al. [24, 26] discover the airflow pattern of a ceiling fan in a closed room at various rotational speeds and blade angles using CFD. The basic analysis model is comprised of models of three and four wings fans. After that, bald angles and rotational speed were varied under different blade angles. The result showed that a fan with four wings produced comfort conditions at blade angles 12°

and rotational speeds of 400 rpm. Borowski et al. [23] studied the velocity field of airflow flowing through a swirl diffuser using the Particle Image Velocimetry (PIV) method for different values of Reynolds number. The result showed that the velocity vector values decrease with the increase in Reynolds number. Its lowest velocity vector values were found at Reynolds numbers of 143,000 and above. Halibart et al. [25] analyzed the effect of plenum box (top and side) entry on the velocity profile concerning the diffuser face panel using mathematical and numerical models. Analyses show that the plenum box entry has a very significant effect on the dispersion of the air flowing out of the air diffuser. Furthermore, the use of perforations in the plenum box may help to slow down the velocity. In this work, a free architecture design tool called PCON was used to create the geometry of the office/home along with the furniture. OpenFOAM was used to create a mesh and simulate the flow through a square ceiling diffuser and understand the Coanda effect which is created by such a diffuser. Open source code called Paraview was used to analyze the CFD simulations.

2. Research methods

The whole methodology followed from creating the model, meshing, simulation, and post-processing is described as follows:-

- Geometry modeling using pCon.
- Mesh generation using snappyHexMesh.
- CFD: case setup with appropriate boundary condition and simulation.

2.1 OpenFOAM

OpenFOAM is a C++ toolkit used to develop custom numerical solvers and pre/post-processing tools for solving continuum mechanics problems mostly related to computational fluid dynamics. The acronym OpenFOAM stands for Open Source Field Operation and Manipulation. It is free software, having an open-source code which makes it a popular choice in both industrial and academic systems.

2.2 SnappyHexMesh

snappyHexMesh is an octree mesh generator utility within the OpenFOAM framework to create hex dominant mesh with desired minimum quality.

Key features of snappyHexMesh are-

- It starts from a predefined mesh and refines it.
- It works on .stl, .obj files having triangulated formats of geometry.
- It can work in watertight as well as non-watertight geometries.
- Can be run in parallel systems
- RL (refinement level) can be adjusted

2.3 pCon

pCon Planner is software enabling room furnishing, space planning, and furniture configuration. pCon is preferred by architectural designers and engineers because of its simplicity of use and wide range of features. It was designed by EasternGraphics GmbH in llmenau (Thuringia/Germany).

2.4 ParaView

ParaView is an open-source post-processing application that enables its users to analyze and visualize data. It was generally developed to analyze large datasets through the usage of distributed computation models. ParaView is an important tool in the field of computational fluid dynamics, which helps in improving design efficiency through CFD simulations.

2.5 Geometry modeling using pCon planner

The 3-D schematic model of a room of dimensions 6x4.8x3m^3 (length x breadth x height) is created using pCon planner software as shown in Fig. 2 and the 3-D geometry model as viewed in ParaView depicted in Fig. 3. This model is used for meshing and numerical analysis in the OpenFOAM software. The room consists of sofas, two beds, a cupboard, one table, four windows, and two doors. The inlet air diffuser is placed in the middle of the room ceiling. There is only one outlet in the geometry, placed at the ceiling. It is a watertight geometry which means that there is no escape route for air, apart from the outlet provided at the ceiling. Air is coming into the room via an inlet which is located at the center of the roof.

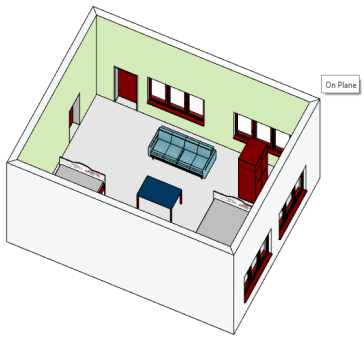

Fig.2: Geometry modeling using pCon planner (Ceiling removed for clearer interior view).

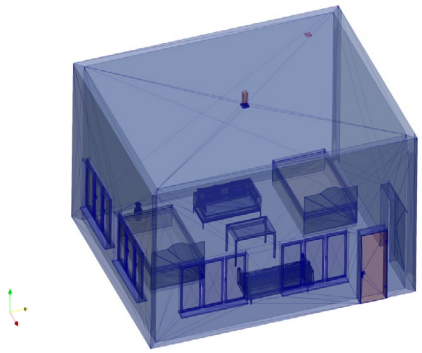

Fig.3: The 3-D geometry model as viewed in ParaView.

2.6 Mesh generation using snappyHexMesh

The steps of mesh generation are as follows:-

- The meshing problem is defined in the form of a stl or nastran file.
- The background mesh of hexahedral cells is generated using blockMesh. Fills the entire region inside the outer boundary.
- At feature edges and surface, cellsplitting is performed using castellatedMeshControls. Octree approach is used for better refinement.
- Retained cells are identified by location point within the defined region. Approximately, if half or more of the cell area lies within the region, then the cell is retained. Non-retained cells are removed.
- The points on the surface are snapped to create the required mesh.

The cut section of mesh along the x-axis of the model created as shown in Fig.3 is depicted in Fig. 4. Fig. 5 shows mesh near the diffuser region created using the castellatedMeshControls. This feature helps in surface refinement in regions where a finer mesh is required.

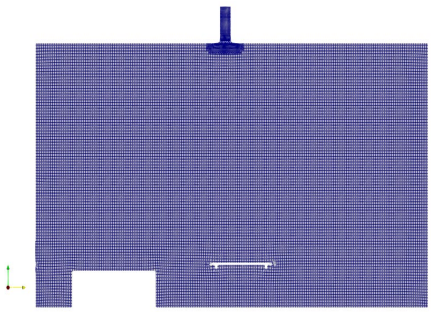

Fig.4: Cut section of mesh along the x-axis.

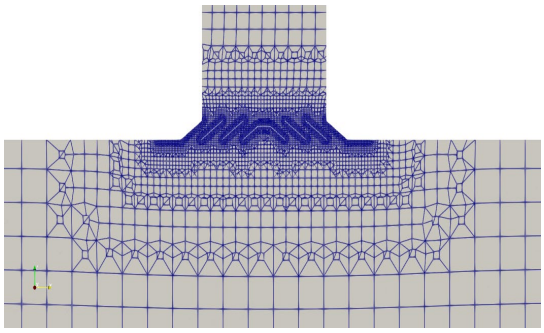

Fig.5: Diffuser mesh structure.

2.7 Simulation by applying appropriate boundary conditions in OpenFOAM

The geometry is created and mesh is formed, which forms the basis of simulation. OpenFOAM solves the governing equations and produces the results.

2.7.1 Governing equations

The conservation equations like continuity, momentum, energy, standard k-Ω SST model, etc. are used in the OpenFOAM solver to solve the airflow analysis. The equations are as follows:-

Equation of continuity

$$\nabla.V=0 \quad \text{----------(i)}$$

Equation of momentum or Navier Stoke's equation

$$-\nabla.P+\mu_{eff}\nabla^2 v+\rho g\beta(T-T_{ref})=\rho v.\nabla v \quad \text{---(ii)}$$

Where, ρ, V, P, μ_{eff}, β, T_{ref}, T, g are density, velocity and pressure, effective dynamic velocity, thermal expansion coefficient of air, reference point temperature, temperature for air, gravity acceleration of air respectively.

Equation of energy

$$\rho c_p v.\nabla T=\lambda_{eff}\nabla^2 T \quad \text{----------(iii)}$$

Where, c_p and λ_{eff} are specific heat for constant pressure and effective thermal conductivity.

Equation of standard k-Ω SST turbulent model

$$\frac{\partial}{\partial x_i}(\rho\mu_t k)=(\mu+\frac{\mu_t}{\rho_k})\nabla^2 k+\mu_t S^2-\rho\varepsilon \quad \text{-----(iv)}$$

$$\frac{\partial}{\partial x_i}(\rho\mu_t\varepsilon)=(\mu+\frac{\mu_t}{\rho_\varepsilon})\nabla^2\varepsilon+C_{\varepsilon_1}\frac{\varepsilon}{k}\mu_t S^2-C_{\varepsilon_2}\rho\frac{S^2}{K} \quad \text{-----(v)}$$

$$\mu_t=\frac{\rho C_\mu k^2}{\varepsilon} \quad \text{----------(vi)}$$

The coefficients of $C_\mu, \rho_k, \rho_\varepsilon, C_{\varepsilon 1}, C_{\varepsilon 2}$ = 0.09, 1.0, 1.3, 1.44, 1.44 respectively and S= $(S_{ij} S_{ij})^{0.5}$

2.7.2 Boundary conditions

The volumetric flow rate of 0.043 m^2/s is applied at the inlet and fixed pressure (atmospheric pressure) 101325 Pa is used. The fluid material properties and geometry dimensions are tabulated below in Table 1.

Table 1. Boundary conditions

Item	Value
Air density, ρ	Ideal gas law
Dynamic air viscosity, μ	1.7894 x10^{-5} kg/m-s
Specific heat of air, cp	1006.43 J/kg-K
Thermal conductivity of air, k	0.0242 W/m-K
Coefficient of thermal expansion of air, β	0.0034 K^{-1}
Turbulent viscosity ratio	10
Reynolds number at the inlet R_{ei} ($=\rho V_i d_i/\mu$)	23,961
Room dimensions	6x4.8x3m^3
Diffuser dimensions	165.6x165.6 mm^2
Outlet dimensions, d_i	15.28x215.28 mm^2
Inlet Temperature, T_i	-287 K (14 °C)
Average Room Temperature, T_a	296K (23 °C)

3. Result and Discussion

Air is coming into the room via an inlet which is located at the center of the roof. Air then passes through a small duct and enters the room via a ceiling diffuser (four ways). Air, as it passes through the ceiling diffuser, gets split in four directions owing to the geometry of the ceiling diffuser. The geometry of the diffuser is as shown in Fig.6. Air jets coming out of the ceiling diffuser stick to the roof and move out towards the wall. It is to be noted that air does not follow downward movement but keeps moving along the roof. The jet spreads out as can be seen in Fig.7 but remains attached to the roof. This phenomenon is called the Coanda effect. The flow velocity at the ceiling is higher compared to the rest of the domain as illustrated in Fig7. The red color denotes higher velocities and blue denotes lower velocity. It's evident from figure 7 that the velocity of air keeps decreasing as it moves further away from the diffuser toward the walls.

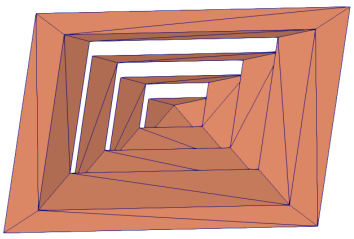

Fig. 6: Square ceiling diffuser.

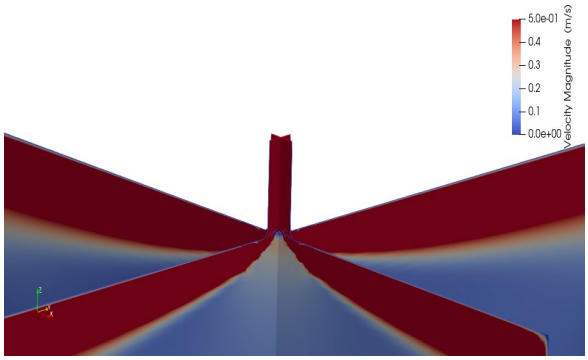

Fig.7: Four-way directional airflow coming out of diffuser.

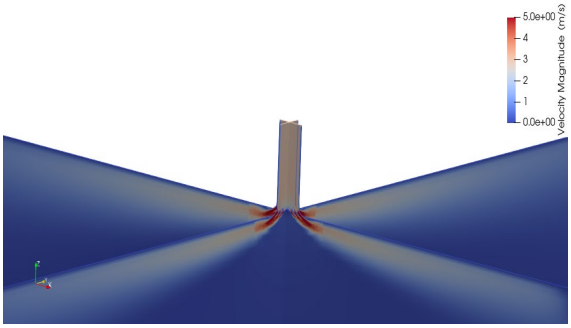

Fig.8: Airflow velocity magnitude along with the ceiling.

The airflow velocity vector is depicted in Fig.9. It can be seen in Fig. 9 that airflow gets split through diffuser slots in all four directions with uniform velocity. As the airflow moves along the ceiling, its velocity keeps on decreasing as it moves further away from the diffuser.

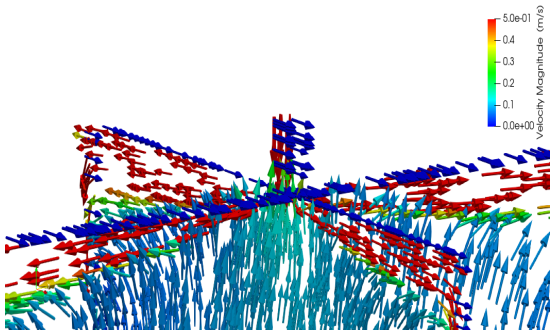

Fig.9: velocity vectors of airflow.

The velocity vector in the cut plane is shown in Fig.10. The velocity vector is colored by velocity magnitude. By the virtue of the Coanda effect, the airflow moves along the ceiling/roof equally in all four directions and the air is then directed downwards and creates four vortexes. This leads to an even distribution of conditioned air throughout the room, as shown through the velocity vectors in Fig.10. The air velocity magnitude becomes less than 0.2 m/s (shown by dark blue vectors) till the movement reaches the room occupants, which is comforting for the occupants of the room

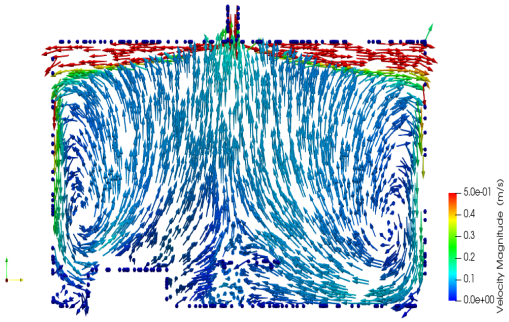

Fig.10: Cut section of the Velocity vector.

The diffuser vanes change the direction and velocity of flow. As flow passes through diffuser vanes, it is forced to bend owing to the geometry of the vanes. The change in geometry causes the flow to separate at the edge of the vanes as shown in Fig.11. According to Bernoulli's principle, an increase in fluid velocity occurs simultaneously with a decrease in static pressure. The bluish region in Fig.11 denotes an area of decreased pressure near the diffuser slots. Simultaneously, Fig. 12 shows the increased velocity of air in the same region, denoted by the red color. The increase in velocity of airflow is due to the pressure drop near the diffuser slots.

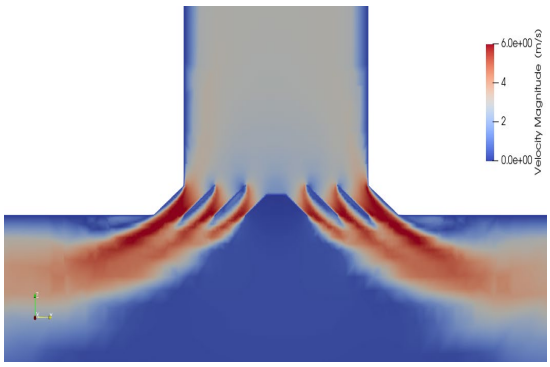

Fig.11: Airflow along with diffuser slots.

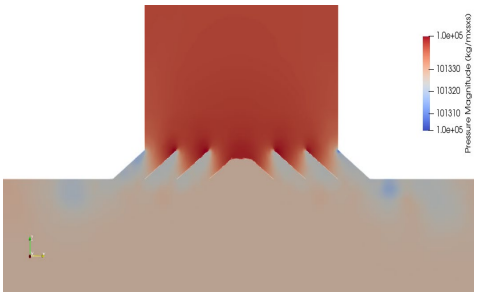

Fig.12: Pressure drop along with diffuser slots.

4. Conclusion

1. The main objective of the study was to understand the airflow patterns generated by the intake of a square ceiling diffuser placed in the center of the ceiling of the room.
2. A three-dimensional model is created using freeware called pCon and then open-source code OpenFOAM is used to generate the mesh in the geometry and to do CFD simulation. The post-processing is done in ParaView which is a freeware and open-source code.
3. Simulation was able to capture the phenomena of the Coanda effect by which flow after coming from the ceiling diffuser flows along the roof and spreads air uniformly.
4. The square ceiling diffuser throws air in all four directions.
5. The flow sticks to the ceiling and it moves towards the walls of the room, thus creating the Coanda effect.
6. Air velocity is reduced to 0.2-0.3m/s as it reaches the floor and toward the center of the room. As the velocity of air is reduced to a very small value which indicates that no possibility of the draught of air onto the room occupants is there
7. The results showed that a square ceiling diffuser distributes conditioned air properly throughout the room at a velocity that is comforting for room occupants.

References

1) ASHRAE Handbook Fundamentals, “Thermal Environmental Conditions for Human Occupancy”, *Chapter 20, Space Air Diffusion,* 55 (2013).
2) G. Einberg, K. Hagström, P. Mustakallio, H. Koskela, and S. Holmberg, “CFD modeling of an industrial air diffuser—predicting velocity and temperature in the near zone”, *Building and Environment*, 40(5) 601-615 (2005).
3) Y. Sun, and T.F. Smith, “Airflow characteristics of a room with square cone diffusers”, *Building and Environment*”, 40(5) 589-600 (2005).
4) A.Q.A Halim, B.T. Tee, W.S.W. Wan and M. Zain, “Face Bypass Damper Application to Overcome Air Conditioning Oversizing Issue”, *EVERGREEN Joint Journal of Novel Carbon Resource Sciences & Green Asia Strategy*, 8(3) 686-692(2021).
5) H. Kokash, “Optimization through CFD analysis of a Coanda effect enhanced novel HVAC diffuser” *Doctoral dissertation, University of Michigan-Flint,* (2020).
6) H. Kotani, T. Yamanaka, and Y. Momoi, (2002). “CFD simulation of airflow in the room with multi-cone ceiling diffuser using measured velocity and turbulent parameters in large space”, *In Proceedings of 8th International Conference on Air Distribution in Rooms,* 117-120 (2002).
7) M. A. Aziz, I.A.M. Gad, E.S.F.A. Mohammed, and R.H. Mohammed, “Experimental and numerical study of influence of air ceiling diffusers on room air flow characteristics”, *Energy and Buildings*, 55, 738-746 (2012),
8) M. I Alhamid, R. M Miftah, and M. ARainanda, "Analysis of Ozonation Effect on Performance and

Water Quality in Closed-System Cooling Tower", *EVERGREEN Joint Journal of Novel Carbon Resource Sciences & Green Asia Strategy*, 8(4) 896-903(2021).

9) Y.C. Liu, M.G. Burzo, S. Sier, and C. Ellis, "Improved thermal comfort of office occupants through better air diffuser designs", *In ASME International Mechanical Engineering Congress and Exposition*, 57496, (2015).
10) M. Jaszczur, P. Madejski, S.Kleszcz, M. Zych, and P. Palej, "Numerical and experimental analysis of the air stream generated by square ceiling diffusers", *In E3S Web of Conferences*, 128(08003) (2019).
11) A. Awwad, M.H. Mohamed, and M. Fatouh, "Study the effect of ceiling air diffuser blade and lip angles using CFD". *Athens Journal of Technology and Engineering*, 4(4) 309-358(2017).
12) B. Venås, T.T. Harsem, and B.A. Børresen, "CFD simulation of an office heated by a ceiling mounted diffuser", *In 35th AIVC conference, 4th TightVent Conference, 2nd Ventilcool Conference, Ventilation and airtightness in transforming the building stock to high performance, Poznan, Poland*, 24-25 (2014).
13) A. A. Farizi, H. Lee, and H. Han, "Thermal Performance of Alternating-Current Heat Recovery Ventilator in Partially Wet Conditions", *EVERGREEN Joint Journal of Novel Carbon Resource Sciences & Green Asia Strategy*, 8(1) 221-228 (2021).
14) G. Einberg, "Air diffusion and solid contaminant behaviour in room ventilation: a CFD based integrated approach", *Doctoral dissertation, KTH*, (2005).
15) F. Savanti, E. Setyowati, and G. Hardiman, "The Impact of Ventilation on Indoor Air Quality and Air Change Rate", *Joint Journal of EVERGREEN Novel Carbon Resource Sciences & Green Asia Strategy*, 9(1) 219-225 (2022).
16) H. Han, M. Hatta, and H. Rahman, "Smart Ventilation for Energy Conservation in Buildings", *EVERGREEN Joint Journal of Novel Carbon Resource Sciences & Green Asia Strategy*, 6(1) 44-51(2019).
17) S.G. Mijorski, D.G. Markov, G.T. Pichurov, P. Stankov, N.G. Ivanov, and I.S. Simova, "CFD based design of a ventilated space". *In IOP Conference Series: Materials Science and Engineering*, 618(1), 012049 (2019).
18) A. Mogra, P.K. Pandey, and K.K. Gupta, "Computational Fluid Dynamics Analysis of a Class Room for Effective Utilization of Position of Air Conditioning System", *In IOP Conference Series: Materials Science and Engineering*, 810(1), 012029 (2020).
19) A. Nocente, T. Arslan, S. Grynning, F. Goia , "Effect of Ceiling Diffuser on Diffuse Ceiling Ventilation Performance", *Proceedings of the 16th IBPSA Conference Rome, Italy* (2019).
20) H.T. Xu, and J.L. Niu, "A New Method of CFD Simulation of Airflow Characteristics of Swirling Floor Diffusers", *Eighth International IBPSA Conference*, 1429-1433(2003).
21) R. Prakash, D. Christopher, K. Kumarrathinam, "CFD Analaysis of Flow through a Conical, Exhaust Diffuser", *IJRET*, 3(11) 239-244 (2014)
22) C. Uraijaree, K. Tangchaichit, and J. Suriyawanakul, "Comparison of airflow-uniformity in different ceiling-diffuser designs with various concentric circles in shrimp freezer using CFD", *IOP Conf. Series: Earth and Environmental Science*, 113, 012205 (2018). doi :10.1088/1755-1315/113/1/ 012205
23) M. Borowski, M. Karch, R. Łuczak1, P. Życzkowski, and M. Jaszczur, "Numerical and experimental analysis of the velocity field of air flowing through swirl diffusers", *E3S Web of Conferences* 128, 05003(2019). https://doi.org/ 10.1051/ e3sconf/ 201912805003.
24) J. B. Rao, D. B. Rao, J.S. Yadav, M. Sreerama, "Simulation of Air Flow Around Ceiling Fan in an Enclosed Space by using CFD, *IJSTR*, 9(6), 1061-1064 (2020).
25) J. Halibart , K. Z. nska , M. Borowski , and M. Jaszczur , " Analysis of the Velocity Distribution in the Plenum Box with Various Entries", *Energies* 14(36302021), 1-17 (2021). https:// doi.org/10.3390/ en14123630
26) Y. D. Herlambang, Supriyo, B. Prasetiyo, A.S. Alfauzi, T, Prasetyo, Marliyati, F. Arifin, "Experimental and Simulation Investigation on Savonius Turbine: Influence of Inlet-Outlet Ratio Using a Modified Blade Shaped to Improve Performance", *EVERGREEN Joint Journal of Novel Carbon Resource Sciences & Green Asia Strategy*, 9(2) 457-464 (2022).
27) Safril, Mustofa, M. Zen, F.S.M. Wirandi, "Design of Cooling System on Brushless DC Motor to Improve Heat Transfers Efficiency, *EVERGREEN Joint Journal of Novel Carbon Resource Sciences & Green Asia Strategy*, 9(2), 584-593, (2022).
28) A. R.A. Bahar, A.S Yatim, E.P. Wijaya1, "CFD Analysis of Universitas Indonesia Psychrometric Chamber Air Loop System", Y. D. Herlambang, Supriyo, B. Prasetiyo, A.S. Alfauzi, T, Prasetyo, Marliyati, F. Arifin, "Experimental and Simulation Investigation on Savonius Turbine: Influence of Inlet-Outlet Ratio Using a Modified Blade Shaped to Improve Performance", *EVERGREEN Joint Journal of Novel Carbon Resource Sciences & Green Asia Strategy*, 9(2) 457-464 (2022).