Proceedings of the National Conference on Computational Modeling of Fluid Dynamics Problems (CMFDP-2019)
NIT Warangal, India – Jan 18-20, 2019
Paper No. XXX (The number assigned when the papaer is accepted)

HVAC CFD Analysis for a meeting Room: A Case Study

B U Rahul Shetty[a] , Manoj Yadav BG[a], Rohan R[a], Suhail K T[a], Priyabrata Adhikary[a],

[a]*Department of Mechanical Engineering, New Horizon college of engineering, Bangalore – 560103, Karnataka, India*

*Corresponding author Email: ________________rahulshetty0746@gmail.com________________

Received: ate? Accepted: Date?

This case study deals with the CFD based analysis of air conditioning in a conference room. The flow of air can be induced through mechanical means (air conditioner). Where air behaves as a fluid and naturally air flows from a higher pressure to lower pressure. Air flow can be disambiguated using CFD.so here by using FLoEFD commercial code. To perform analysis on HVAC model (meeting room) 1 TR two air conditioner on the same wall and 1TR two air conditioners on opposite walls by saving cost-time-effort.

1. Introduction

Airflow, or air flow is the movement of air from one area to another. The primary cause of airflow is the existence of pressure gradients. Air behaves as a fluid, meaning particles naturally flow from areas of higher pressure to where the pressure is low. Atmospheric air pressure is directly proportional to altitude, temperature, and composition. Airflow is a measurement of the amount of air per unit of time flowing through a particular room.

The flow of air can be induced through mechanical means (such as by operating an electric or manual fan) or can take place randomly, as a function of pressure differentials present in the surroundings. Like any fluid, air will exhibit laminar and turbulent flow patterns. Laminar flow occurs when air flows smoothly and exhibits a parabolic velocity profile; turbulent flow occurs when there is an irregularity, which alters the direction of movement. Turbulent flow exhibits a flat velocity profile.

2. Methodology Adopted

The speed at which a fluid flows against an object varies with distance from the surface of the liquid. The region surrounding an object where the air speed approaches zero is known as the boundary layer. It is here that surface friction most affects flow; irregularities in surfaces may affect thickness of boundary, and hence act to disrupt flow. Air flow can be disambiguated using Computational Fluid Dynamics (CFD) modelling or observed experimentally through the operation of a wind tunnel. This may be used to predict airflow patterns around automobiles, aircraft, and marine craft, as well as air penetration of a building envelope. In this case, the governing system of equations can be written as follows:

$$\frac{\partial \rho}{\partial t} = -\nabla \cdot (\rho\, \mathbf{u}) \qquad (1)$$

$$\rho\left(\frac{\partial}{\partial t}+\mathbf{u}\cdot\nabla\right)\mathbf{u} = -\nabla P + \rho\mathbf{g} + \frac{1}{c}\mathbf{J}\times\mathbf{B} \tag{2}$$

$$\rho\left(\frac{\partial}{\partial t}+\mathbf{u}\cdot\nabla\right)e = -P\nabla\cdot\mathbf{u} + \rho\mathbf{u}\cdot\mathbf{g} + \frac{1}{\sigma}\mathbf{J}^2 \tag{3}$$

3. Theory and Calculation

The modified ***k*-ε** turbulence model (as used here through FloEFD commercial code for saving time-cost-effort) with damping functions proposed by Lam and Bremhorst (1981) describes laminar, turbulent, and transitional flows of homogeneous fluids consisting of the following turbulence conservation laws

$$\frac{\partial\rho k}{\partial t}+\frac{\partial\rho k u_i}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\left[\mu+\frac{\mu_t}{\sigma_k}\right]\frac{\partial k}{\partial x_i}\right]+\tau_{ij}^R\frac{\partial u_i}{\partial x_j}-\rho\mathcal{E}P_B,$$

$$\frac{\partial\rho\mathcal{E}}{\partial t}+\frac{\partial\rho\mathcal{E}u_i}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\left[\mu+\frac{\mu_t}{\sigma_\mathcal{E}}\right]\frac{\partial\mathcal{E}}{\partial x_i}\right]+C_{\mathcal{E}1}\frac{\mathcal{E}}{k}\left[f_1\tau_{ij}^R\frac{\partial u_i}{\partial x_j}+C_B\mu_t P_B\right]-f_2C_{\mathcal{E}2}\frac{\rho\mathcal{E}^2}{k}k$$

$$\tau_{ij}=\mu s_{ij},\quad \tau_{ij}^R=\mu_t s_{ij}-\frac{2}{3}\rho k\delta_{ij},\quad s_{ij}=\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k},$$

$$P_B=-\frac{g_i}{\sigma_B}\frac{1}{\rho}\frac{\partial\rho}{\partial x_i},$$

where $C_\mu=0.09,\ C_{\mathcal{E}1}=1.44,\ C_{\mathcal{E}2}=1.92,\ \sigma_k=1,\ \sigma_\mathcal{E}=1.3,\ \sigma_B=0.9,\ C_B=1$

if $P_B>0,\ C_B=0\ if\ P_B<0.$ The turbulent viscosity is determined from:

$$\mu_t=f_\mu.\frac{C_\mu\rho k^2}{\mathcal{E}},$$

Lam and Bremhorst's damping function f_μ is determined from:

where

$$R_y=\frac{\rho k^2}{\mu\mathcal{E}}$$

y is the distance from point to the wall and Lam and Bremhorst's damping functions $f_1\ and\ f_2$ are determined from:

$$f_1=1+\left[\frac{0.05}{f_\mu}\right]^3,\quad f_2=1-e^{R_t^2}.$$

Lam and Bremhorst's damping function f_μ , f_1 , f_2 decrease turbulent viscosity and turbulence energy and increase the turbulence dissipation rate when the Reynolds number R_y based on the average velocity of fluctuations and distance from the wall becomes too small. When $f_\mu=1$, $f_1=1$, $f_2=1$ the approach obtains the original -***k*-Ɛ** model.

Result and Discussion:

Case-1(1TR * 2 No.AC – opposite Wall): In this case where we get the required parameters, by installing 2tr one air conditioner in a system, where after performing flow simulation.

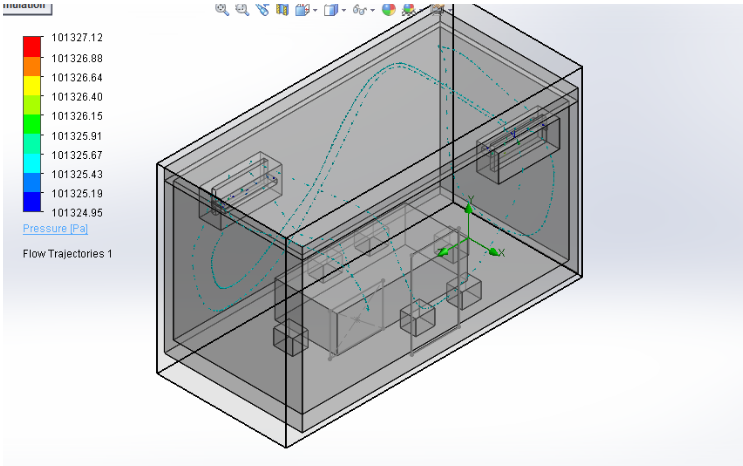

Fig. 1. Pressure- CFD analysis

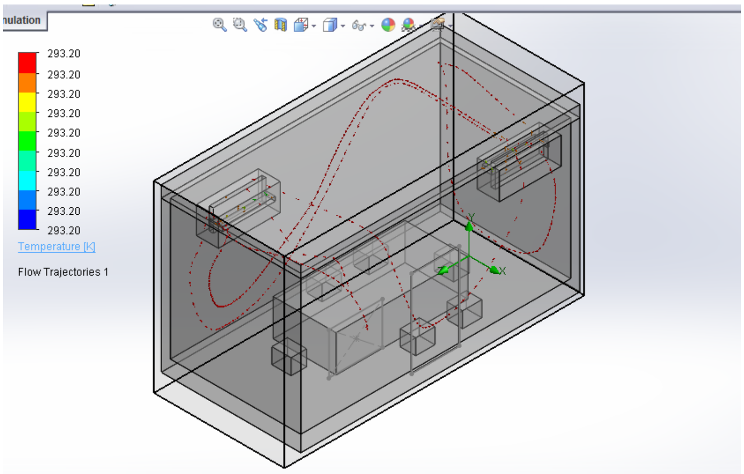

Fig. 2. Temperature- CFD analysis

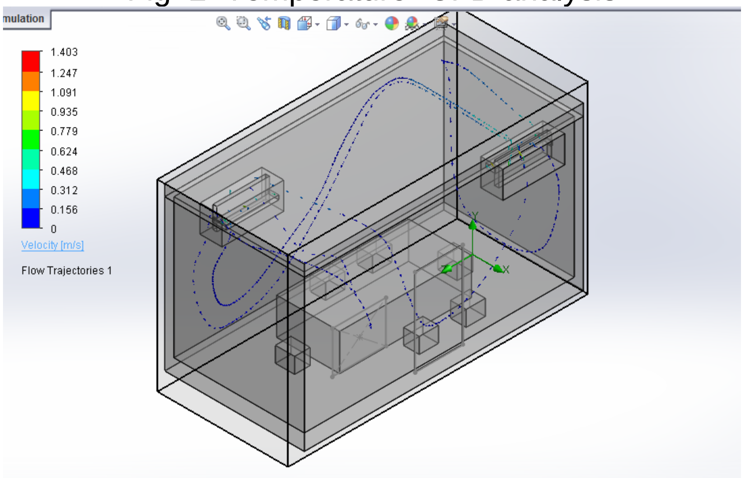

Fig. 3. Velocity- CFD analysis

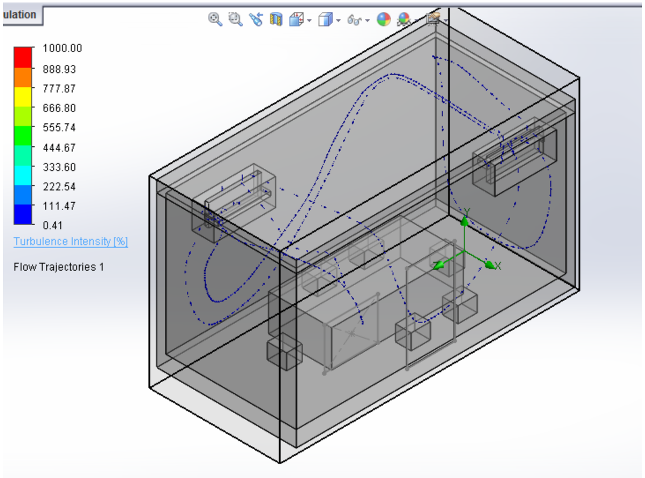

Fig. 4. Turbulence intensity- CFD analysis

Case-2(1TR * 2 No.AC – Same Wall): In this case where we get the required parameters, by installing 1tr two air conditioners in a system, by performing flow simulation.

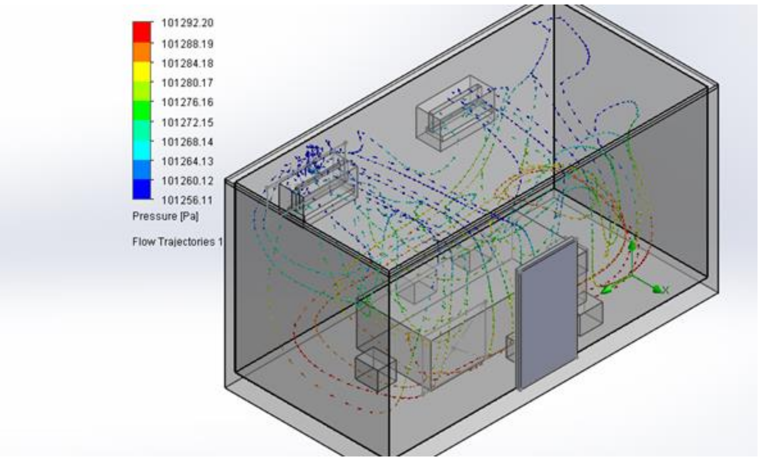

Fig. 5. Pressure- CFD analysis

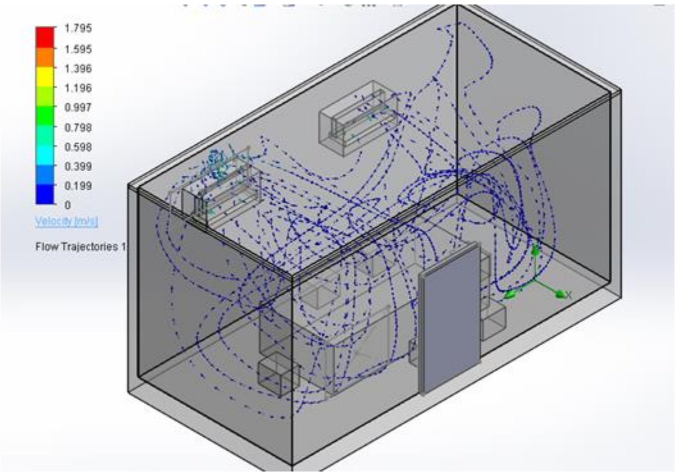

Fig. 6. Velocity- CFD analysis

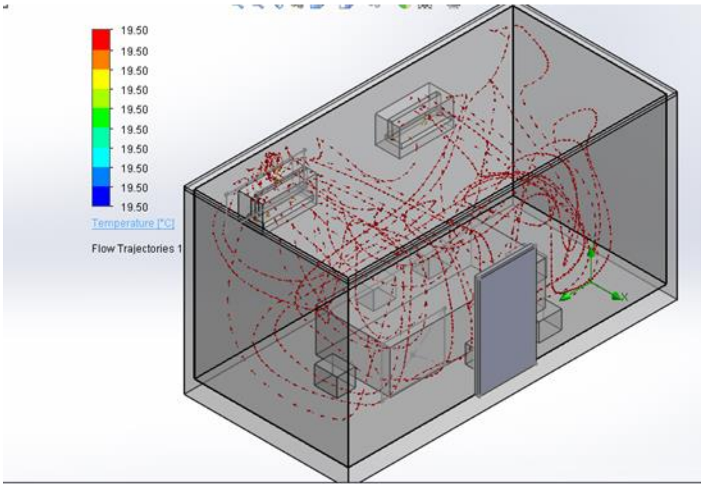

Fig. 7. Temperature-CFD analysis

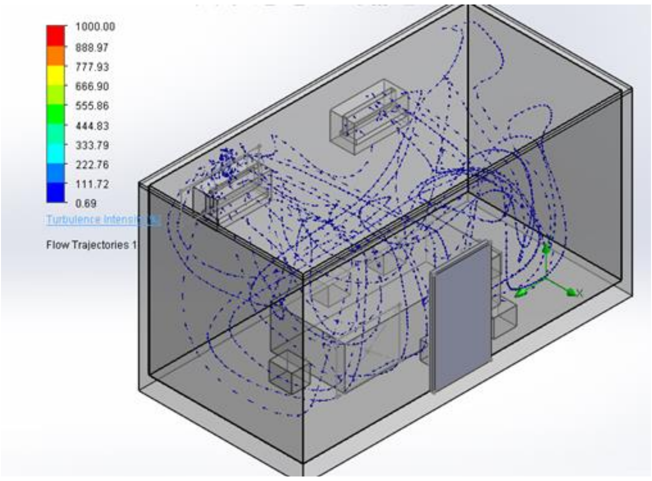

Fig. 8. Turbulence intensity- CFD analysis

4. Conclusion

By performing CFD analysis we have come to know that one ton two air conditioners fixed on the same wall perform better than one ton two air conditioner fixed on the opposite walls by saving time-cost-effort.

5. Acknowledgements

The authors wish to thank N.H.C.E. Bangalore, M.G.P.L. Bangalore and S.W.R.E., Jadavpur University, Kolkata for the valuable technical support.

References

1. P.Adhikary;Design and CFD analysis for pump impeller;2nd RSTC (WR), Science Congress 2017 (WBSSTC 2018), Paper ID: EG-P25

2. P.Adhikary;CFD analysis of concept car for improvement of aerodynamic design;2nd RSTC (WR), Science Congress 2017 (WBSSTC 2018), Paper ID: EG-P20

3. P.Adhikary;Design and CFD analysis of centrifugal pump;2nd RSTC (WR), Science Congress 2017 (WBSSTC 2018), Paper ID: EG-P11

4. P.Adhikary;12V DC pico hydro for hilly rural electrification: performance analysis by C.F.D. (Best Poster Award);2nd RSTC (WR), Science Congress 2017 (WBSSTC 2018), Paper ID: EG-P1

5. P.Adhikary;Micro Hydropower Generation from Rural Drinking W.T.P.: C.F.D. Analysis & It's Validation;FMFP 2016, International Conference, MNNIT Allahabad

6. P.Adhikary;C.F.D. ANALYSIS OF 12V 10W DC MICRO HYDRO TURBINE: A CASE STUDY;NCETPFS 2016, National Conference, Jadavpur University, Kolkata

7. P.Adhikary;Design and development of Arduino controlled self balancing duct cleaning robot: a case study;International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:121-124