Research Inventy: International Journal of Engineering And Science
Vol.5, Issue 7 (July 2015), PP -63-71
Issn (e): 2278-4721, Issn (p):2319-6483, www.researchinventy.com

Numerical Simulation Effects on Return Air Vent with Indoor Thermal Environment

Dr. Swaminathan[1], C.Chendramohan[2], G.paremesh[3]
[1]*Professor, Department of Mechanical, Faculty of HITS ,Hyderabad ,India.*
[2]*Professor, Department of Mechanical, Faculty of HITS, Hyderabad,India.*
[3]*Asso. Professor, Department of Mechanical, Faculty of HITS, Hyderabad India.*

ABSTRACT:- *The air distribution & ventilation method, position of return air vent and air change rate having significant role on thermal environment, human comfort and energy consumption as well for an air conditioned space. In order to improve thermal environment of space, study of temperature distribution as well as air distribution is required. In this paper, author has numerically simulated the temperature and velocity contour of space in two different position of return air vent in air conditioning system. These positions are return air vent on the same wall as inlet wall and another at opposite wall. In present work computational fluid dynamics software (ANSYS-FLUENT 6.3.26 Solver) has been used for simulation. Air flow (ANSYS-FLUENT) module has been used in present work. Air flow (ANSYS-FLUENT) module predicts the temperature and velocity at three test locations which are 1 meter apart from both side walls. The Standard k-ε model is used in this numerical simulation with SIMPLE approach of pressure velocity compounding. The convergence of solution of the continuity, energy and momentum equation is obtained within the set standard criterion. The Standard k-ε model is used in this numerical simulation with SIMPLE approach of pressure velocity compounding. The convergence of solution of the continuity, energy and momentum equation is obtained within the set standard criterion. From the result it has been found that the location of the return air position in the same wall gives better human comfort, however the location of the return air position in the opposite wall gives better energy savings.*

Key words: *Computational Fluid Dynamics (CFD), Numerical Simulation, Thermal Environment.*

I. INTRODUCTION

In the present scenario, the requirement and demand of Air-conditioning is goes on increasing due to increase in average temperature of the environment due to reason of greenhouse effect. Due to this, energy consumption is also increases and for this concern, energy saving gets importance. Therefore there is requirement of better air distribution and air conditioning effectiveness.

The ventilation has a major impact on indoor air quality and human comfort of air conditioning space [1]. It also has a highly significant on the energy use in space. The influence of many variables has been explored separately such as air flow control [2, 3, 4], heat recovery [5, 6], building air tightness [7] and humidity control [8]. In previous few years, many studies have been done about various parameter affecting air distribution & its effectiveness, human comfort, indoor environment, thermal environment and indoor air quality. The inlet and outlet locations are very important for evaluating indoor thermal comfort [9]. The measured air temperature and air velocities were within the limits of thermal comfort standards, although temperature and relative humidity were located at the extreme of the limits [10]. The throw height from a diffuser, diffuser number, supply air temperature, total flow rate, cooling load and heating and cooling mode can have a major impact on air distribution effectiveness [11].

In order to improve indoor environment in air conditioned space is done by changing some basic factor like air distribution method, humidity level, lighting condition, air temperature, and air velocity distribution and ventilation system. From these factors last two factor i.e. ventilation system and air temperature & velocity distribution have important role in indoor thermal environment. In this paper for manipulating the ventilation system return air vent location is different (in same and opposite wall) and to get trend of air temperature & velocity distribution, there are few locations at some specific distance gap the space model.

In recent year application of CFD grow rapidly which has led to expansion of research field related to air conditioning and analysing and optimising the different parameter of it. This approach of research is also recognised well because it is faster, viable and economical method to predict the result compare to experimental method. After the developments of computer hardware and software, numerical simulation methodology and advance mathematical models, there are easiness to carryout critical investigation on effect of air vent and their position on different parameters like temperature, velocity etc. in different locations. Computational fluid dynamics (CFD) predictions were performed in this paper two dimensional numerical study is performed and purpose is to determine distribution of temperature and velocity at different locations with respect to vertical dimension (height). CFD prediction shown that by changing location of return air vent affects different parameter like velocity characteristics & temperature gradient with respect to vertical dimension i.e. height. For the analysis of these cases, the numerical technique Computational Fluid Dynamics (CFD) has been used. The geometrical model for the three cases is prepared using GAMBIT 2.4.6 and numerical solution is obtained using ANSYS-FLUENT 6.3.26.

II. SOLUTION DOMAIN/ TEST MODEL

Numerical simulation is done by using CFD analysis. A two-dimensional solution domain is generated. A room model is prepared as a plane consisting of supply air location and return air location. The dimension of the room or conditioned space in.Numerical Simulation of Effect of Return Air Vent on Indoor Thermal Environment the present problem is taken as 4m (Length) X 3m (Height). Two cases have been taken in this study as follows

Case 1: The location of the supply air position is near the ceiling and below 0.55 meters from the ceiling and return air position is at 0.3 meters below the ceiling on the same wall.
Case 2: The location of the supply air position is near the ceiling and below 0.55 meters from the ceiling and return air position is at 0.3 meters below the ceiling on the opposite walls.

Two cases are summarised in the following table 1:

Table 1 Position and location of different air vent.

Case	Centre of Supply air Vent (0.1 m size)Location	Centre of Return air vent (0.2 m wide) Location
I	0.55 meters below the ceiling	0.3 meter below the ceiling on the same wall
II	0.55 meters below the ceiling	0.3 meter below the ceiling on the opposite wall

The assumptions which are imposed for the numerical analysis are following.

- The flow is steady and two-dimensional.
- The heat transfer is negligible from surrounding.
- The change in potential energy of the system is zero.

2.1. CFD Simulation

A commercial software computational fluid dynamics (CFD) code was used to predict the effect of location of return air vent on indoor thermal environment. A CFD solver package, ANSYS-FLUENT 6.3.26 was used to perform all the CFD computations. In the prescribed criterion the solution were obtained through iteration process. The Standard k-ε model is used in this numerical simulation with SIMPLE approach of pressure velocity compounding. The convergence of solution of the continuity, energy and momentum equation is obtained within the set standard criterion.

2.2. MESHING

From the GAMBIT 2.4.6 meshing is done in that domain in a non-uniform mesh with very fine mesh size is used. The following grid size of density have been analysed in this investigation (a) 1,80,000, (b) 1,40,000, (c) 1,20,000 and negligible difference in results are found, the complete analysis with the grid density of 1,20,000 is performed. Due to analysis is done with 2-dimension mesh, it saves not only computer memory but computational time as well.

2.3. BOUNDARY CONDITION

The boundary conditions used are shown in the following table 2:

Table 2 Required boundary condition.

Sr. No.	Boundary	Boundary Condition Used
1	Inlet	Velocity Inlet
2.	Outlet	Pressure outlet
3.	Room walls	Wall

In this simulation the different walls of the room are set at different temperatures and the values at the boundaries are tabulated in the following table 3:

Table 3 Value of boundary condition.

Sr. No.	Boundary	Value
1.	Velocity of supply air	0.4 m/s
2.	Temperature of Supply air	295 K
3.	Temperature of Ceiling	308 K
4.	Temperature of side walls	305 K
5.	Temperature of floor	303 K

III. RESULT AND DISCUSSION

In this work, the simulation methodology and utilizing the boundary conditions, simulations were completed to obtain following results:

- Temperature variation with respect to different height at different locations.
- Velocity variation with respect to different height at different locations.

The simulation results are obtained in the form of contours and plots of temperature and velocity. For analysis of the temperature and velocity within the conditioned space, three test locations as L1, L2 and L3 are considered at 1 meter apart from both the side walls. The results are shown below for the above mentioned three cases as under:

Case 1: The location of the supply air position is near the ceiling and below 0.55 meters from the ceiling and return air position is at 0.3 meters below the ceiling on the same wall. (Fig.1)

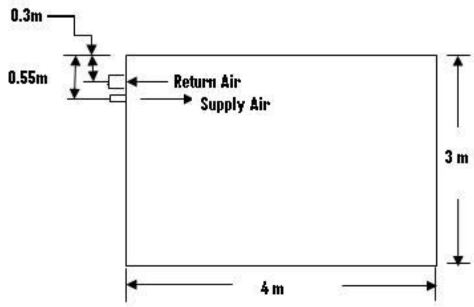

Figure 1 Figure showing position of different air vent when location of air vent in the same wall.

The three locations are represented (Fig.2) here following:

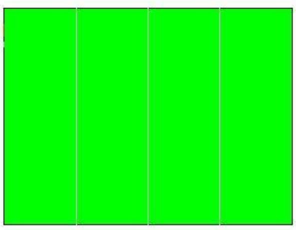

Figure 2 Contour showing different location of the model.

Temperature Variation

Temperature Contours

The contour is a curved shape outline or boundary of some property of some specific constant value. The temperature contour is the curve in which temperature value is same within the curve. The temperature contour is represented (Fig.3) as following:

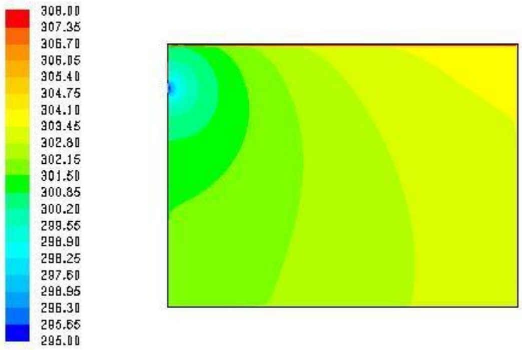

Figure 3 Temperature contour of the model when location of return air vent in the same wall.

Temperature Plots

The temperature plots with respect to different locations are following:

All the three of first case of temperature plots are represented (Fig.4) in a plot as following. In this plot, all the curves is following nearly similar fashion, but location 1's curve is a little more curvy, location 2's curve is moderate and last location 3's curve is least.

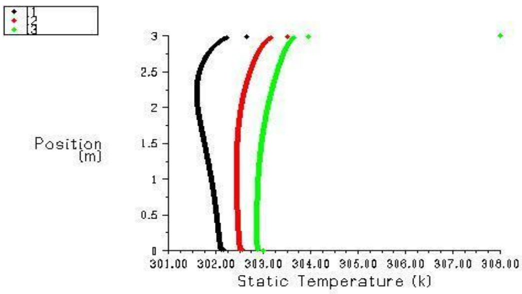

Figure 4 Temperature plot at different location when location of return air vent in the same wall.

Velocity Variation

Velocity Contours

The velocity contour is represented (Fig.5) as following:

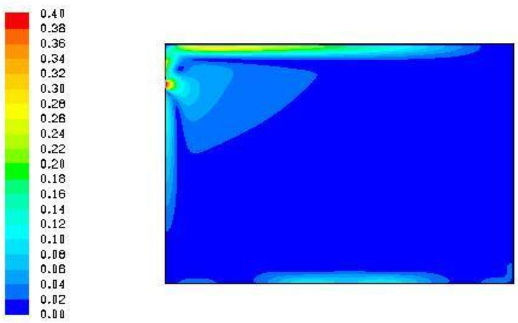

Figure 5 Velocity contour of the model when location of return air vent in the same wall.

Velocity Plots

The velocity plots with respect to different locations are following:

All the three of first case of velocity plots are represented (Fig.6) in a plot as following. In this plot, all the curves is following nearly similar fashion, but location 1's curve is a little more curvy and both other curve does not significantly change.

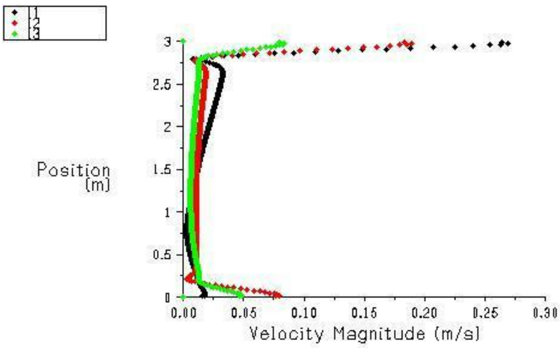

Figure 6 Velocity plot at different location when location of return air vent in the same wall.

Case 2: The location of the supply air position is near the ceiling and below 0.55 meters from the ceiling and return air position is at 0.3 meters below the ceiling on the opposite walls. (Fig.7)

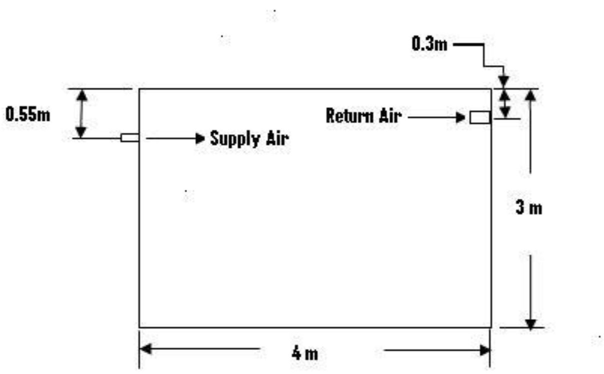

Figure 7 Figure showing position of different air vent when location of return air vent in the opposite wall

Temperature Variation

Temperature Contours

Temperature Contour at Location P1is represented (Fig.8) as following:

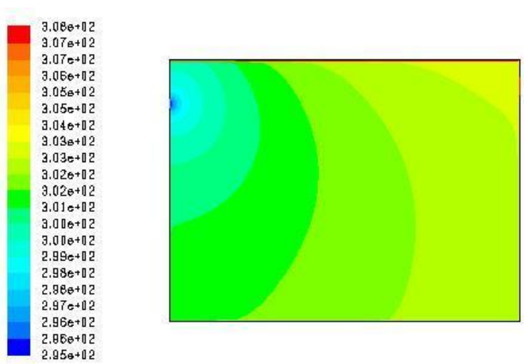

Figure 8 Temperature contour of the model when location of return air vent in the opposite wall

Temperature Plots
All the three of second case of temperature plots are represented (Fig.9) in a plot as following. In this plot, the 1st curve is curvier and move some more backward and then forward in upper strata. The other are less curvy and move more in forward direction means increased temperature in higher strata. The 2nd curve is least and 3rd is moderate curvy.

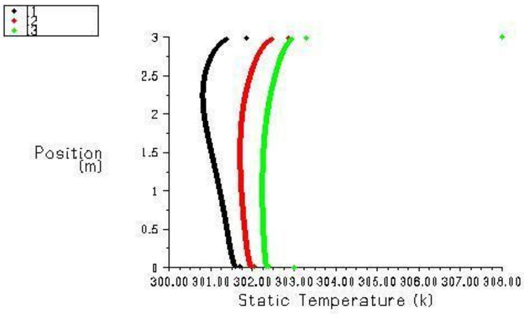

Figure 9 Temperature plot at different location when location of return air vent in the opposite wall.

Velocity Variation

Velocity Contours
Velocity Contour at Location P1 is represented (Fig.10) as following

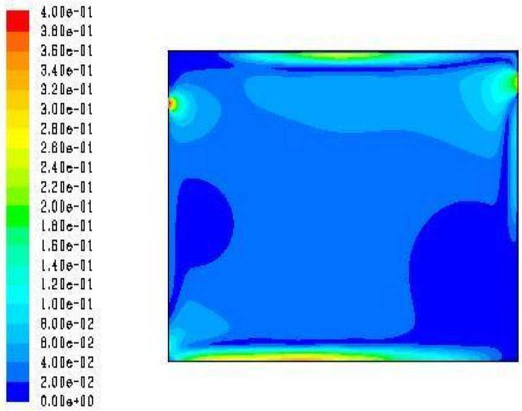

Figure 10 Velocity contour of the model when location of return air vent in the opposite wall

Velocity Plots
All the three of second case of velocity plots are represented (Fig.11) in a plot as following. In this plot, it is found that all of these three curves are intersecting at some different height and not much variation in value. But curve of location 1st and 2nd are following same pattern however curve of location 3rd is follow completely different.

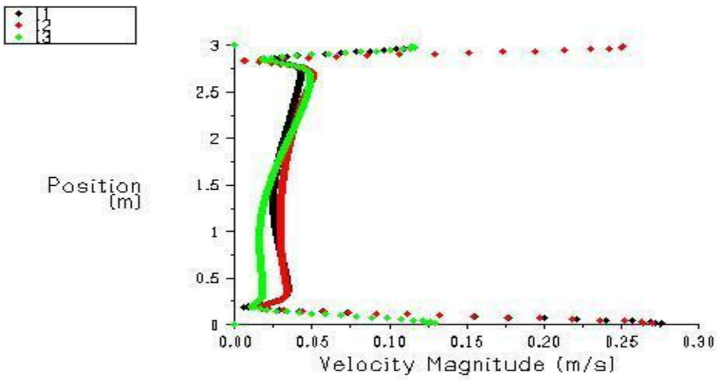

Figure 11 Velocity plot at different location when location of return air vent in the opposite wall.

Comparative plots of temperature and velocity

Temperature Plots

- Temperature Plots for the three cases at the location L1:

The following height verses static temperature plot is represented (Fig.12) in L1 location in these two cases. The black curve is shown for 1st case and red one is for 2nd case. On comparing both the curve it is found that both curve follow same fashion in this location but the second curve (means return vent in opposite wall) shows the lower temperature compare to first curve (means return vent on same wall) which shows higher temperature.

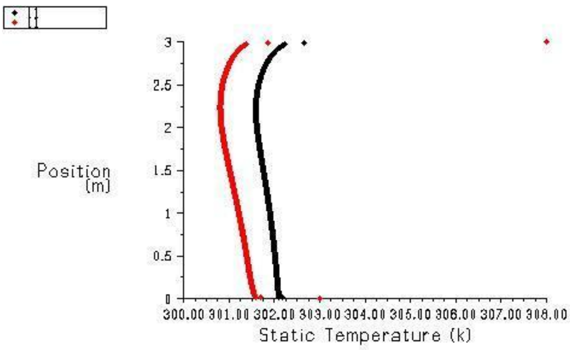

Figure 12 Temperature plot at different position (same and opposite wall) of return air vent in the location 1.

- Temperature Plots for the three cases at the location L2:

The following height verses static temperature plot is represented (Fig.13) in L2 location in these two cases. The black curve is shown for 1st case and red one is for 2nd case. It is found that both curve follow same fashion in this location and also same lower temperature for 2nd case compare to 1st case. But this case curves are little bit less curvy.

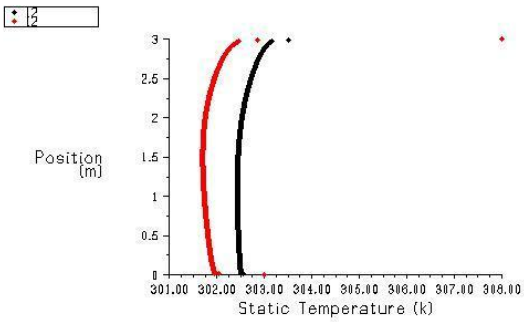

Figure 13 Temperature plot at different position (same and opposite wall) of return air vent in the location 2

- Temperature Plots for the three cases at the location L3:

For the two cases in the location L3, the following height verses static temperature plot is represented (Fig.14). The black curve is shown for 1st case and red one is for 2nd case. Similarly in this plot also both the curve follows same fashion in this location and also same lower temperature for 2nd case compare to 1st case. This time curve isn't much changing with height.

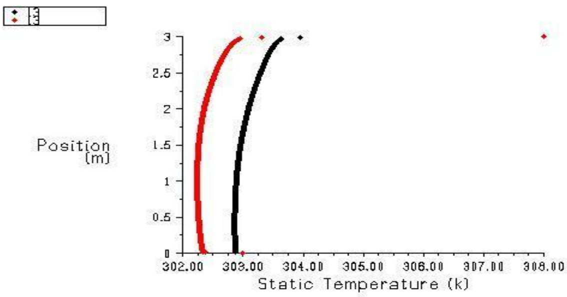

Figure 14 Temperature plot at different position (same and opposite wall) of return air vent in the location 3

Velocity Plots

- Velocity Plots for the three cases at the location L1:

The position (height) verses velocity (magnitude) plot is represented (Fig.15) is following in first location. The black curve is shown for 1st case and red one is for 2nd case. Both the curves are following similar fashion after height more than 1.5m but below that in lower strata value of velocity is different. The higher velocity is in 2nd case compare with 1st case.

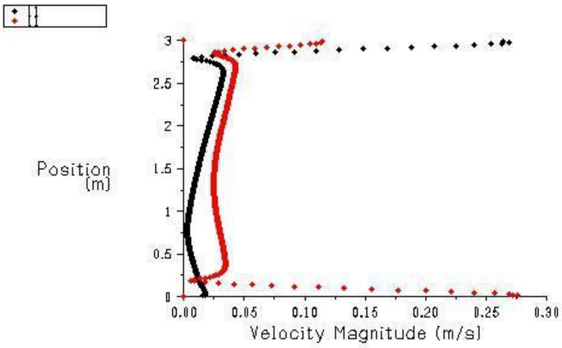

Figure 15 Velocity plot at different position (same and opposite wall) of return air vent in the location 1

- Velocity Plots for the three cases at the location L2:

In the second location, position (height) verses velocity (magnitude) plot is represented (Fig.16) is following. The black curve is shown for 1st case and red one is for 2nd case. Both the curves are following similar fashion and much varying velocity magnitude throughout the position. But the value of velocity magnitude is higher in 2nd case compare to 1st case.

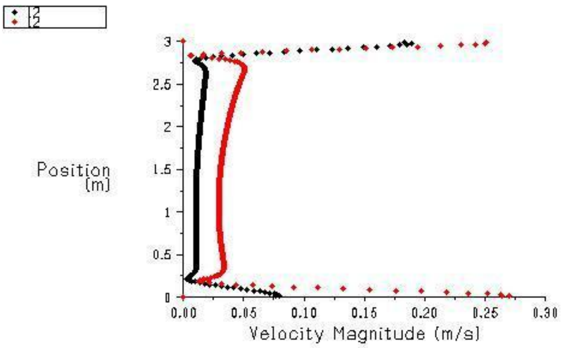

Figure 16 Velocity plot at different position (same and opposite wall) of return air vent in the location 2

- Velocity Plots for the three cases at the location L3:

The position (height) verses velocity (magnitude) plot is represented (Fig.17) is following in third location. The black curve is shown for 1st case and red one is for 2nd case. Both the curves are completely different from each other and value of velocity in this location for 2nd case is much higher in upper strata. This difference is decreasing rapidly as going in lower strata.

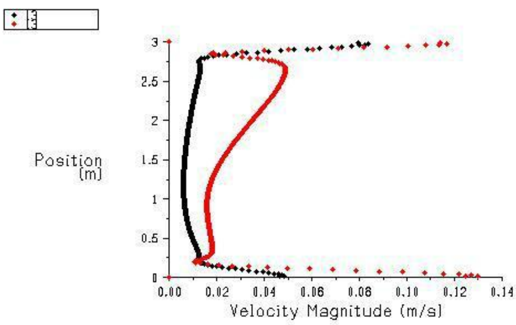

Figure 17 Velocity plot at different position (same and opposite wall) of return air vent in the location 3

IV. CONCLUSIONS

The CFD analysis is carried out in a two dimensional room model to study the temperature distribution and air velocity distribution in the space. The effect of location of return air vent on indoor thermal environment has been studied. From the results of the analysis conclusion are drawn. If the location of return air vent is in same wall as supply air vent, then it will gives a better human comfort due to its lower velocity of air which lead to lesser draft. If the location of return air vent is in opposite wall to supply air vent, then there will be more energy savings. It is due to its lower temperature in the space with the present supply, so there may be less energy is required to run the system to fulfil the requirement.

REFERENCES

[1] Sundell J., On the history of indoor air quality and health, *Indoor air 14* (2004) 51–58.

[2] Kolokotroni M., Aronis A., Cooling-energy reduction in air conditioned offices by using night ventilation, *Applied Energy* 63 (4) (1999) 241–253.

[3] Budaiwi I. M., Al-Homoud M.S., Effect of ventilation strategies on air contaminant concentrations and energy consumption in buildings, *International Journal of Energy Research* 25 (12) (2001) 1073–1089.

[4] Sherman M.H., Walker I.S., Meeting residential ventilation standard through dynamic control of ventilation systems, *Energy and Building* 43(8) (2011) 1904-1912.

[5] Roulet C.A., Heidt F.D., Foradini F., Pibiri M.C., Real heat recovery with air handling units, *Energy and Buildings* 33 (5) (2001) 495–502.

[6] Dodoo A., Gustavsson L., Sathre R., Primary energy implications of ventilation heat recovery in residential buildings. *Energy and Buildings* 43 (7) (2011) 1566-1572.

[7] Nabinger S., Persily A., Impact of airtightening retrofits on ventilation rates and energy consumption in a manufactured home, *Energy and Buildings* 43(12) (2011).

[8] Lin Z., Lee C. K., Fong K.F., Chow T.T, Comparison of annual energy performances with different ventilation method for temperature and humidity control, *Energy and Buildings* 43 (12) (2011) 3599–3608.

[9] Chung K. C., Lee C.Y., Predicting Air Flow and Thermal Comfort in an Indoor Environment under Different Air Diffusion Models, *Building and Environment*, 31(1), pp. 21–26, 1996.

[10] Cheong K.W.D, Djunaedy E.,Chua Y.L., Tham K.W., Sekhar S.C.,Wong N.H.,Ullah M.B., Thermal comfort study of an air-conditioned lecture theatre in the tropics, *Building and Environment* 38 (2003) 63 – 73.

[11] Lee K.S., Jiang Z., Chen, Q. 2009 Air distribution effectiveness with stratified air distribution systems, *ASHRAE Transactions*, 115(2).

[12] Nikam S., Bartaria V.N., Indoor Environment In Air Conditioned Spaces A Review, *International Journal of Advanced Technology & Engineering Research*, 2(4), July 2012.

[13] Rakesh Jaiswal, Anupam Raj Jha, Anush Karki, Debayan Das, Pawan Jaiswal, Saurav Rajgadia, Ankit Basnet and Rabindra Nath Barman, Structural and Thermal Analysis of Disc Brake Using Solidworks and Ansys. *International Journal of Mechanical Engineering and Technology*, 7(1), 2016, pp. 67–77.

[14] Saurav S., Bartaria V.N., CFD Analysis of Heat Transfer through Artificially Roughened Solar Duct, *International Journal of Engineering Trends and Technology* 4 (9) September 2013.

[15] Praveen Padagannavar and Manohara Bheemanna, Automotive Computational Fluid Dynamics Simulation of A Car Using Ansys. *International Journal of Mechanical Engineering and Technology*, 7(2), 2016, pp. 101–114.