Optimizing Thermal Comfort for Office Room Using CFD Analysis.

Rajnish Kumar Gautam and Neeraj Agarwal

April 6, 2021

Optimizing Thermal Comfort for Office Room Using CFD Analysis

Rajnish Kumar Gautam	Neeraj Agarwal
IES College of Technology,Bhopal	IES College of Technology,Bhopal
rajnish343gautam@gmail.com	neeraj.bhopal@gmail.com

Abstract: **Thermal comfort is a vital consideration that can be taken into account in the building design process that can have a positive effect on the facets of social and economic activity of persons using these sites. Comfortable climates can encourage the human for relaxation and can thus increase the commercial activity. In the present work computational fluid dynamics analyses have been performed for an office room using ANSYS fluent to investigate the effects of better thermal comfort by changing of four-way cassette AC inlet position. For that four CAD model of office room is designed using the CATIA software with approximate dimension. The four models created were Single AC inlet in office room from top, Double AC inlet in office room from top, Double AC inlet in office room from its side wall and Double AC inlet in office room one from top and other from side wall. Results show that the design-3 of the office room with double AC inlet from its side wall has lest time (44.93% compared with model-1 Single AC inlet and 5% as compared with model-2 Double AC inlet) to achieve the comfort temperature inside office room.**

Keywords: **CAD, CATIA, ANSYS, Temperature.**

I. INTRODUCTION

Thermal comfort is a fundamental aspect to be taken into consideration in buildings/offices configuration measure which can valuably affect the social and monetary conduct parts of individuals utilizing these spots. Comfortable environments can energize the human for unwinding and would thus be able to build the business movement.

As indicated by gökhan güngör (2015) Hoppe's investigation (1998) the greater part of individuals living in metropolitan zones are spending over 90% of their time in cooled indoor spaces. Same examination likewise proposes that assessed expenses of un-ideal thermal climate are higher than the energy cost which would be spent to improve the conditions to the ideal guidelines.

Thermal comfort can be concentrates severally. Hypothetical examinations were generally founded on energy conditions which are worked in the middle of human and climate and required broad numerical work. Useful investigations then again were finished by testing people under different thermal conditions, which were tedious and could be deluding a result of the sincere beliefs of individuals in regards to comfort. Looked at those two techniques, estimating thermal comfort by utilizing reproduction programming gave the advantage of both by having the option to recreate numerous conditions without a moment's delay and doing the colossal numerical work by the guide of computer.

Various meanings can be found in the ASHRAE handbook.

- The state of being happy with the thermal climate.
- Minimum attention requirement for the regulation of internal body temperature.
- Conditions for the comfort of the thermal environment.

.

II. LITERATURE REVIEW

Jéssica Kuntz Maykot at el. [1] inspects the impacts of gender on thermal comfort necessities in places of business. The information comes from 83 field considers directed in 2014 of every three places of business in Florianopolis, in southern Brazil. One of the structures is completely cooled and the other two utilize a blended mode system. H. Change from cooling to characteristic ventilation. The information were estimated by microclimate stations and the detainees' suppositions were dictated by a survey.

Jindal et al.[2] performed a field analysis was during the storm and winter of 2015-2016 to discuss the thermal climate and thermal comfort in the ventilated indoor homerooms of a school Ambala, India. The outcomes show that the warmth resilience of understudies is very high. In India, there are no thermal comfort guidelines for study halls in one of the environment zones. The aftereffects of this examination ought to give rules to Indian thermal comfort principles so that schools can utilize energy effectively.

Földváry V at el. [3] In 2014, the ASHRAE Global Thermal Comfort II task was led by the University of California at the Berkeley Center for the Built Environment and inside the University of Sydney's Environmental Quality Laboratory (IEQ). The operation began with a detailed variety and harmonization of crude data performed around the world from the last twenty years of thermal comfort examinations.

M.T.H. Derks bei el. [4] In the exploration room, the thermal comfort of clinic wellbeing laborers is a subject that has been little investigated. This brought about in some way or another unsuitable thermal comfort and a slight hindrance to approved work execution. The outcomes additionally showed that the ideal impression of warmth for medical attendants - in view of their requirements for thermal comfort and work execution - is preferably cold over unbiased. A useful way to deal with partitioning the hospital into isolated thermal zones with various degrees of comfort for the patient and the guardian is by all accounts the ideal arrangement that decidedly affects the workplace while making approaches to set aside cash.

III. Objective

The main objective of this research to examine the effects of better thermal comfort of an office room by changing its position which is exposed in the same thermal conditions. There are following objective have been set for this work.

1. Study of air conditioning system for thermal comfort.
2. Prepare the different model of office room and placement of AC outlet.
3. Perform the computational fluid dynamic analysis for all above models
4. Compare the various results such as temperature distribution and velocity distribution inside the office room and present the best model for better thermal comfort.

IV. Methodology

A. Computational fluid dynamics (CFD) analysis:

Computer analysis of fluid dynamics is performed with Ansys Fluent for Office Room. The input parameters were taken from the base card. To perform this computational analysis, authoritative equations such as the continuity equation, the momentum equation, the energy equations, the K equation and the ε equations are used.

B. Algorithm used for CFD analysis:

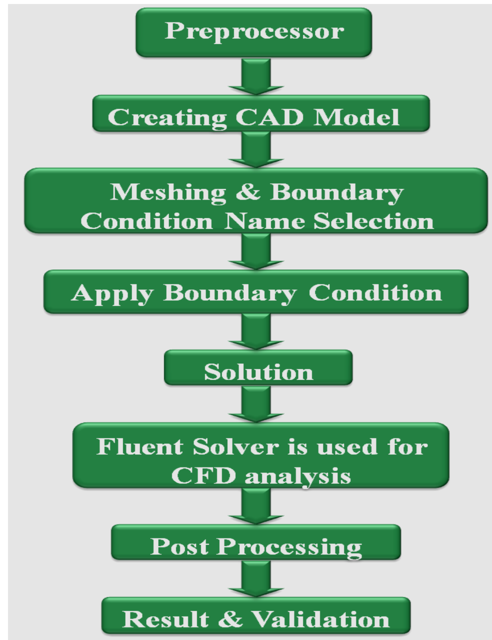

Fig. 1: Algorithm used for CFD analysis

In this work, three CDA office space models are designed using CATIA design software. The approximate size of the office space was taken into consideration when creating the model. The three models created were

1. Single AC inlet in office room from top.
2. Double AC inlet in office room from top.
3. Double AC inlet in office room from its side wall
4. Double AC inlet in office room one from top and other from side wall

C. Governing Equations

1 Conservation of mass or continuity equation:

The continuity equation, is written as following:

$$\frac{\partial \rho}{\partial t} + \nabla.\left(\rho \vec{v}\right) = S_m$$

Where S_m = mass added to the continuous phase or any user defined sources.

For 2D axisymmetric geometries, the continuity equation is given by

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}\left(\rho v_x\right) + \frac{\partial}{\partial r}\left(\rho v_r\right) + \frac{\rho v_r}{r} = S_m$$

Where x is the axial coordinate, r is the radial coordinate, v_x is the axial velocity, and v_r is the radial velocity.

2 Momentum Conservation Equations

Conservation of momentum in an inertial reference frame is described by

$$\frac{\partial}{\partial t}\left(\rho\vec{v}\right) + \nabla.\left(\rho\vec{v}\vec{v}\right) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F}$$

Where p= static pressure

$\bar{\bar{\tau}}$ = stress tensor,

$\rho\vec{g}$ = gravitational body force and

$\vec{F}$ = external body forces

The stress tensor $\bar{\bar{\tau}}$ is given by

$$\bar{\bar{\tau}} = \mu\left[\left(\nabla\vec{v} + \nabla\vec{v}^{T}\right) - \frac{2}{3}\nabla.\vec{v}I\right]$$

where μ = molecular viscosity

I = unit tensor,

For 2D axisymmetric geometries, the axial and radial momentum conservation equations are given by

$$\frac{\partial}{\partial x}\left(\rho v_x\right) + \frac{1}{r}\frac{\partial}{\partial x}\left(r\rho v_x v_x\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(r\rho v_r v_r\right) = -\frac{\partial p}{\partial x} +$$

And

$$\frac{\partial}{\partial t}\left(\rho v_r\right) + \frac{1}{r}\frac{\partial}{\partial x}\left(r\rho v_x v_r\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(r\rho v_r v_r\right) = -\frac{\partial p}{\partial r} + \cdot$$

Where

$$\nabla.\vec{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_r}{\partial r} + \frac{v_r}{r}$$

Where v_x = Axial velocity

v_r = Radial velocity

v_z = swirl velocity

4.3.3 Energy Equation:

The energy equation for the mixture takes the following form:

$$\frac{\partial}{\partial t}\sum_{k=1}^{n}\left(\alpha_k\rho_k E_k\right) + \nabla.\sum_{k=1}^{n}\left(\alpha_k\vec{v}_k\left(\rho_k E_k + p\right)\right) = \nabla.\left(k_{eff}\nabla T\right.$$

where k_{eff} = effective conductivity

S_E = volumetric heat sources

$$E_k = h_k - \frac{p}{\rho k} + \frac{v_k^2}{2}$$

Where

$E_k = h_k$ for an incompressible phase and h_k = sensible enthalpy for phase k

$k - \in model$:

The turbulence kinetic energy, k, and its rate of dissipation, $\in$, are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}\left(\rho k v_i\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\in - Y_M + S_k$$

And

$$\frac{\partial}{\partial t}(\rho\in) + \frac{\partial}{\partial x_i}\left(\rho\in v_i\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\in}\right)\frac{\partial\in}{\partial x_j}\right] + C_{1\in}\frac{\in}{k}\left(G_k + C_{3\in}G_b\right) - C_{2\in}\rho$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients,

G_b is the generation of turbulence kinetic energy due to buoyancy,

Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate,

$C_{1\in}$, $C_{2\in}$, and $C_{3\in}$ are constant. σ_k and $\sigma_\in$ are turbulent Prandtl numbers for k and$\in$,

S_k And $S_\in$ are user-defined source terms.

D. Computational fluid dynamics analysis of office room model-1:

1. CAD geometry: CDA model for office room of model -1 is created using CATIA software. For creating the model approximate dimension of office room were considered and a three dimensional view is illustrated in fig. 2.

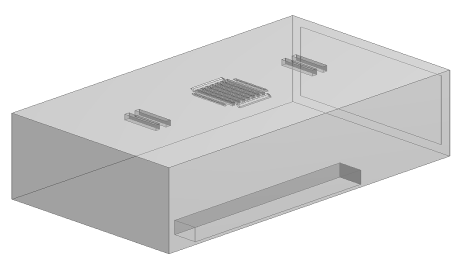

Fig. 2 CDA model of office room model -1

2. Meshing: After completing the CDA model, the office space is imported into the ANSYS workshop to perform another computer aided analysis of fluid dynamics, the next step is networking. In this process, appropriate meshing is done. The geometry of CAD is divided into a huge number of small components called meshes. There

are 69001 nodes and 358336 elements used. This is a rectangular / triangular shape with four and three nodes on each element.

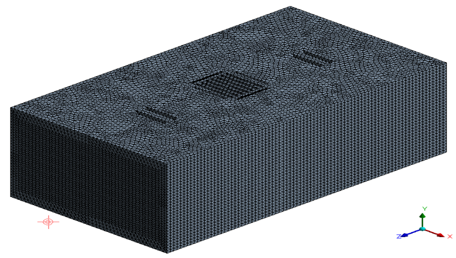

Fig. 3 meshing of office room model -1

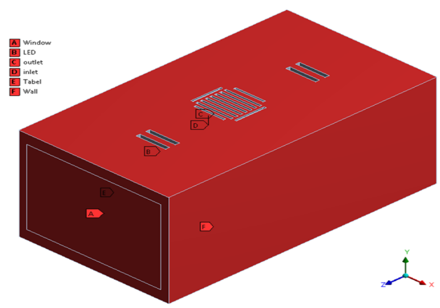

Fig. 4 Different boundaries of office room model-1

3. Boundary conditions:

Boundary conditions are assigned to create a virtual environment of the real life working of the system. The boundary conditions at different location of the office room are explained below.

1. Define the solver setting for pressure based transient and enable gravity option in y direction with the value of -9.81 m/s^2.
2. Working fluid set as air with density 1.22 kg/m3, Specific Heat 1006.43 J/kg K, Thermal conductivity 0.24 W/m^2-K
3. Set viscose model as K-epsilon realizable model with enhanced wall treatment
4. Cold air inlet inside the room at inlet velocity 1 m/sec with 16 ºC
5. To define the temperature distribution, there is requirement for the energy equation.
6. For the outlet boundary condition, the gauge pressure should set as zero.
7. Under Discretization, select standard for Pressure, and second order for Momentum and Energy equation.
8. The Fluent solver is used for CFD analysis.

E. Temperature distribution inside the room after CFD analysis office room for model-1:

After performing computational fluid dynamic transient analyses with absolute velocity formulation using pressure based solver. The temperature distribution inside the room have been analyzed and temperature contour diagram shown in below figure. Till 200 second each contour diagram shown with interval of 10 sec, then after 200 sec the contour diagrams are shown with interval of 50 sec, because first 200 seconds large variation on room temperature have been observed. Cooling time for this model is 905 sec.

F. Velocity distribution inside the room after CFD analysis office room for model-1:

After performing computational fluid dynamic transient analyses with absolute velocity formulation using pressure based solver. The velocity distribution inside the room have been analyzed and velocity contour diagram shown in below figure. Till 100 second each contour diagram shown with interval of 10 sec, then after 100 sec the contour diagrams are shown with interval of 50 sec.

G. Computational fluid dynamics analysis of office room model-2:

1. CAD geometry: CDA model for office room of model -2 is created using CATIA software. For creating the model approximate dimension of office room were considered and a three dimensional view is shown in figure No. 5.

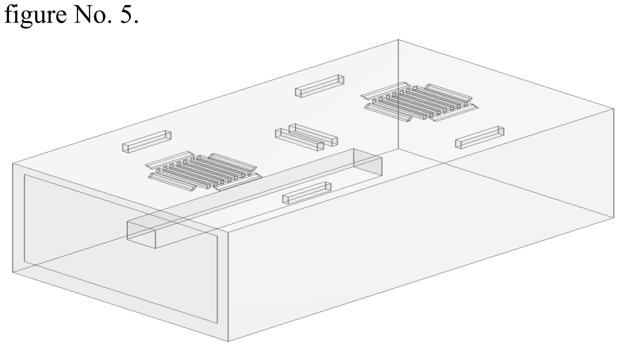

Fig. 5 CDA model of office room model -1

2. Meshing: After completing the CDA model, the office space is imported into the ANSYS workshop to perform another computer aided analysis of fluid dynamics, the next step is networking. In this process, meshing is an important operation in FEA. The geometry of CAD is divided into a huge number of small components called meshes. There are 113,410 nodes and 609,741 elements created. The types of elements used are rectangular and triangular. This is a rectangular / triangular shape with four and three nodes on each element.

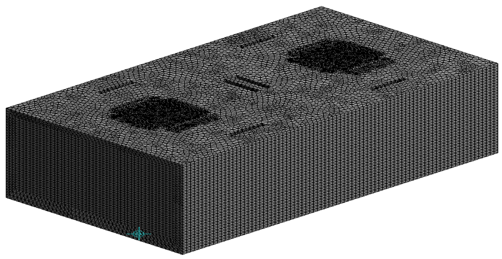

Fig. 6 Meshing of office room model -2

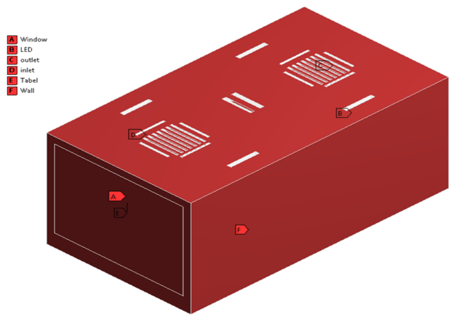

Fig. 7 Different boundaries of office room model-2

Note: Material property & Boundary condition remain same as office room model-1, hence no need to repeat again & again.

H. Temperature distribution inside the room after CFD analysis office room for model-2:

After performing computational fluid dynamic transient analyses with absolute velocity formulation using pressure based solver. The temperature distribution inside the room for model-2 have been analyzed and temperature contour diagram shown in below figure. Till 100 second each contour diagram shown with interval of 10 sec, then after 100 sec the contour diagrams are shown with interval of 50 sec, because first 100 seconds large variation on room temperature have been observed. Cooling time for this model is 602 sec.

I. Velocity distribution inside the room after CFD analysis office room for model-2:

After performing computational fluid dynamic transient analyses with absolute velocity formulation using pressure based solver. The velocity distribution inside the room have been analyzed and velocity contour diagram shown in below figure. Till 100 second each contour diagram shown with interval of 10 sec, then after 100 sec the contour diagrams are shown with interval of 50 sec.

J. Computational fluid dynamics analysis of office room model-3:

1. CAD geometry: CDA model for office room of model -3 is created using CATIA software. For creating the model approximate dimension of office room were considered and a three dimensional view is shown in figure No. 8.

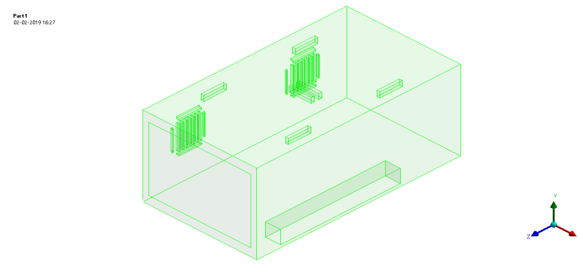

Fig. 8 CDA model of office room model -3

2. Meshing: After completing the CDA model, the office space is brought into the ANSYS workbench to perform another computer aided analysis of fluid dynamics, the next step is networking.

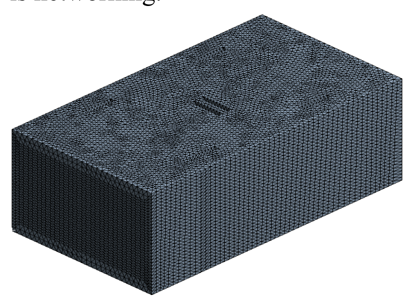

Fig. 9 Meshing of office room model -3

In this process, meshing is an important operation in finite element analysis. CAD geometry is broken down into a large number of small parts called meshes. There are 67286 nodes and 352782 elements in the model. The rectangular and triangular elements are used. This is a rectangular / triangular shape with four and three nodes on each element.

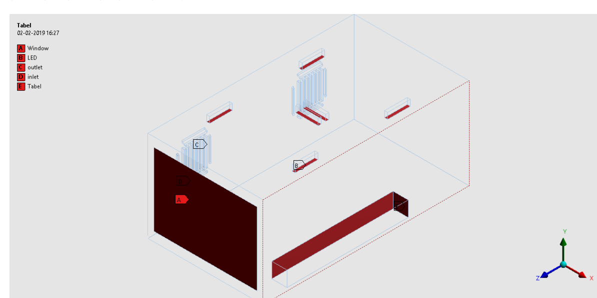

Fig. 10 Different boundaries of office room model-3

K. Temperature distribution inside the room after CFD analysis office room for model-3:

The temperature distribution inside the room for model-3 have been analyzed and temperature contour diagram shown in below figure. Till 100 sec each contour diagram shown with interval of 10 sec, then after 100 sec the contour diagrams are shown with interval of 50 sec, because first 100 seconds large variation on room temperature have been observed. Cooling time for this model is 573 sec.

L. Velocity distribution inside the room after CFD analysis office room for model-3:

After performing computational fluid dynamic transient analyses with absolute velocity formulation using pressure based solver. The velocity distribution inside the room have been analyzed and velocity contour diagram shown in below figure. Till 100 second each contour diagram shown with interval of 10 sec, then after 100 sec the contour diagrams are shown with interval of 50 sec.

M. Computational fluid dynamics analysis of office room model-4:

1. CAD geometry: CDA model for office room of model -4 is created using CATIA software. For creating the model approximate dimension of office room were considered and a three dimensional view is shown in figure No. 11.

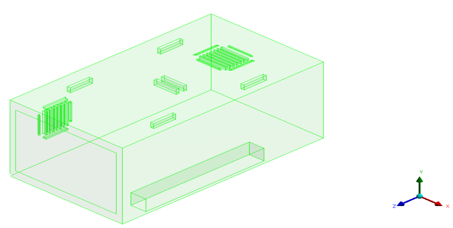

Fig. 11 CDA model of office room model -4

2. Meshing: After completing the CDA model, the office space is imported into the ANSYS workshop to perform another computer aided analysis of fluid dynamics, the next step is networking. In this process, meshing is an important operation in FEA. CAD geometry is broken into meshes. There are 351526 nodes and 67067 elements used in the study.The types of elements used are rectangular and triangular. This is a rectangular / triangular shape with four and three nodes on each element.

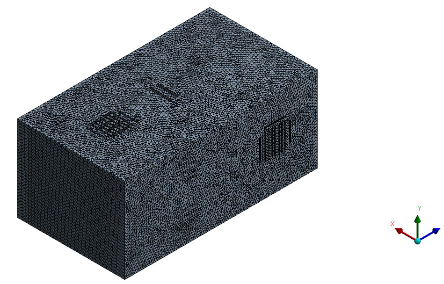

Fig. 12 Meshing of office room model -4

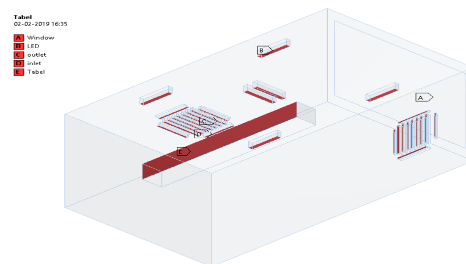

Fig. 13 Different boundaries of office room model-4

N. Temperature distribution inside the room after CFD analysis office room for model-4:

The temperature distribution inside the room for model-4 have been analyzed and temperature contour diagram shown in below figure. Till 100 sec each contour diagram shown with interval of 10 sec, then after 100 sec the contour diagrams are shown with interval of 50 sec, because first 100 seconds large variation on room temperature have been observed. Cooling time for this model is 593 sec.

O. Velocity distribution inside the room after CFD analysis office room for model-3:

After performing computational fluid dynamic transient analyses with absolute velocity formulation using pressure based solver. The velocity distribution inside the room have been analyzed and velocity contour diagram shown in below figure. Till 100 second each contour diagram shown with interval of 10 sec, then after 100 sec the contour diagrams are shown with interval of 50 sec.

V. Result And Discussion

Computational fluid dynamics analyses is performed for an office room using ANSYS fluent to investigate the effects of better thermal comfort by changing of AC inlet position. For all four designs of office room such as single AC inlet from top, double AC inlet in from top, double AC inlet from its side wall and double AC inlet one from top and other from side wall. There are following results with graphical and tabulated data have been explained.

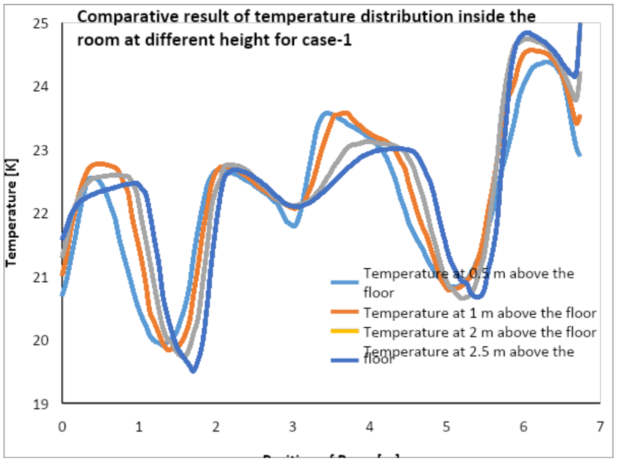

Fig. 14 Comparative result of temperature distribution inside the room at different height for case-1

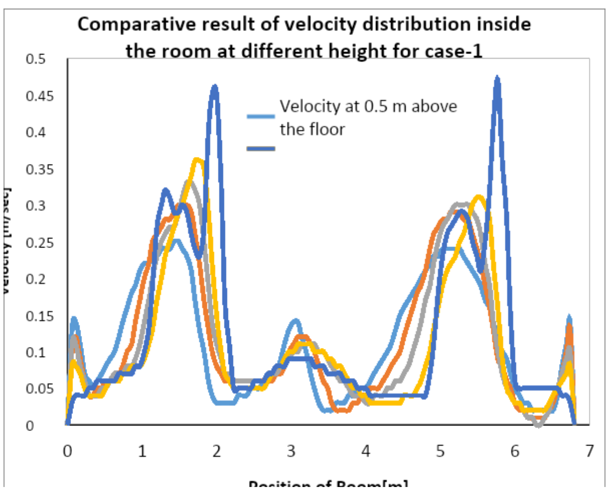

Fig. 15 Comparative result of velocity distribution inside the room at different height for case-1

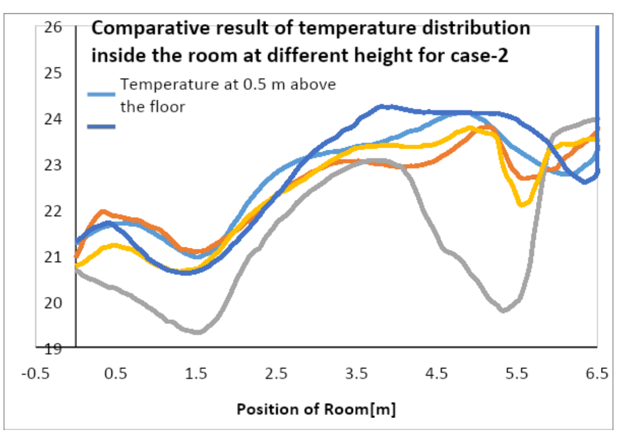

Fig. 16 Comparative result of temperature distribution inside the room at different height for case-2

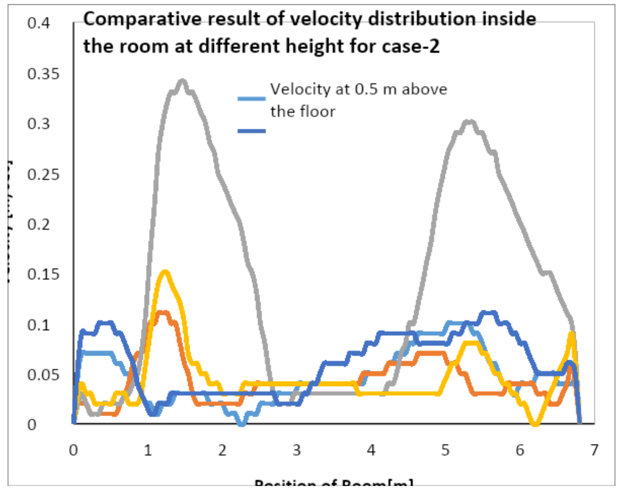

Fig. 17 Comparative result of velocity distribution inside the room at different height for case-2

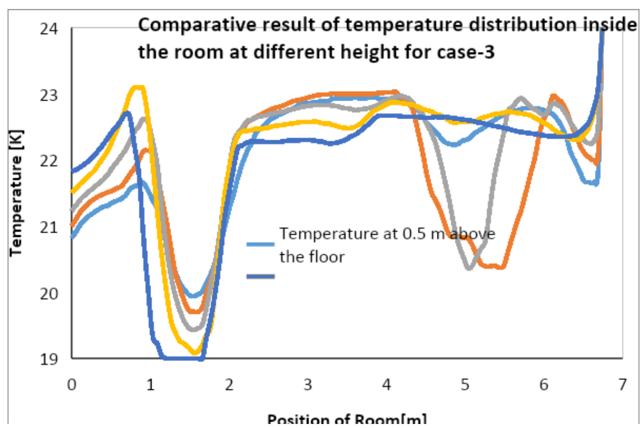

Fig. 18 Comparative result of temperature distribution inside the room at different height for case-3

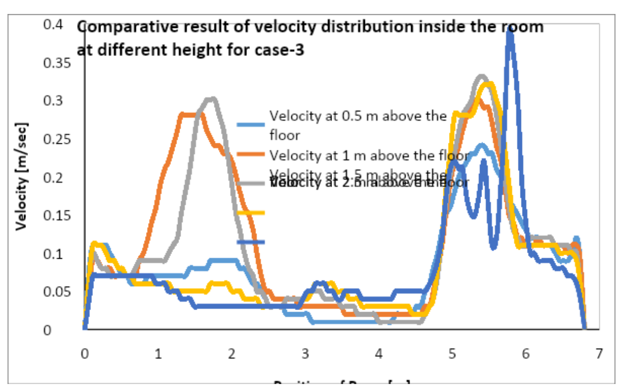

Fig. 19 Comparative result of velocity distribution inside the room at different height for case-3

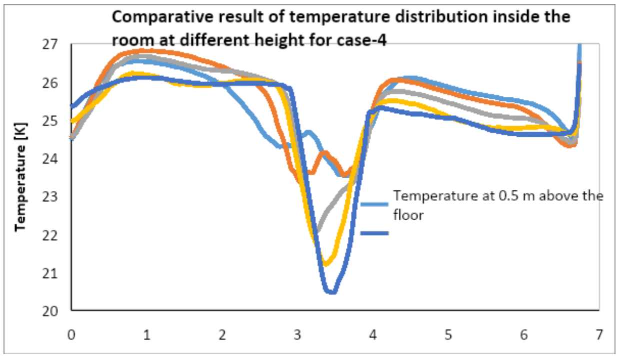

Fig. 20 Comparative result of temperature distribution inside the room at different height for case-4

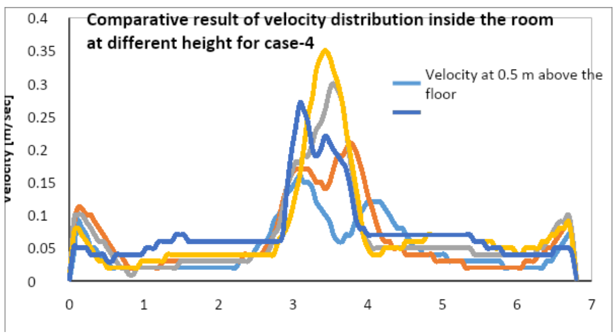

Fig. 21 Comparative result of velocity distribution inside the room at different height for case-4

Table 1 Time taken to achieved the comfort temperature in room

Models	Time in Sec
Mode-1	905
Mode-2	602
Mode-3	573
Mode-4	593

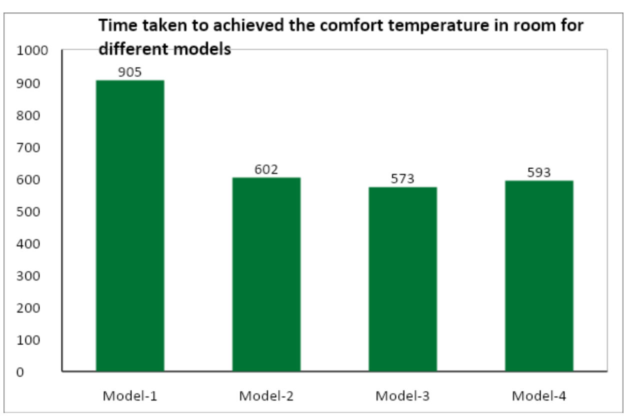

Fig. 22 Time taken to achieved the comfort temperature in room for different models

Percentage difference calculation
Model-1 and Model-2
between $V_1 = 905$ and $V_2 = 602$

$$\frac{[V_1 - V_2]}{\left[\frac{V_1+V_2}{2}\right]} \times 100$$

$$\frac{[905-602]}{\left[\frac{905+602}{2}\right]} \times 100$$

= 40.2% difference as compared with model-1

Model-1, Model-2 and Model-3

$$\frac{[905-573]}{\left[\frac{905+573}{2}\right]} \times 100$$

= 44.93% difference as compared with model-1 and 4.94 % as compared with model-2

Model-1, Model-2 and Model-4

$$\frac{[905-593]}{\left[\frac{905+593}{2}\right]} \times 100$$

= 41.66% difference as compared with model-1 and 1.5 % as compared with Model-2

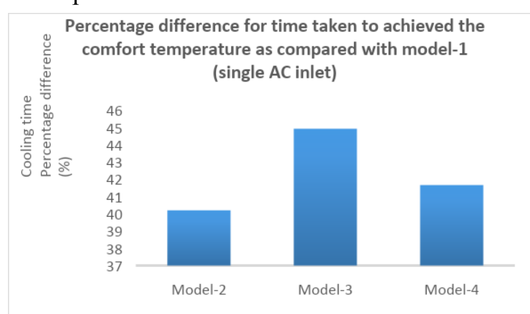

Figure 5.10: Percentage difference for time taken to achieved the comfort temperature as compared with model-1 (single AC inlet)

From the above figure it has been observed that model-3 (Double AC inlet from its side wall) take 44.93% less time to achieved the comfort temperature as compared with model-1 (Single AC inlet from top).

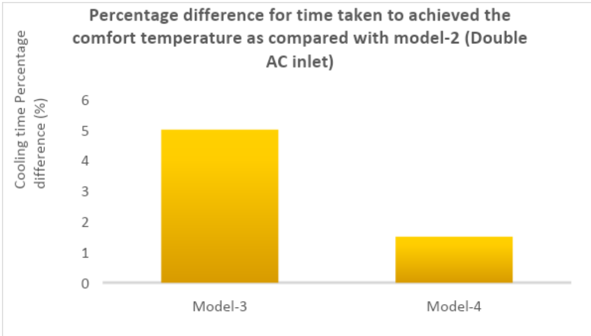

Figure 5.11: Percentage difference for time taken to achieved the comfort temperature as compared with model-2 (Double AC inlet)

From the above figure it has been observed that model-3 (Double AC inlet from its side wall) take 5% less time to achieved the comfort temperature as compared with model-2 (Double AC inlet one from top and other from side wall).

VI. Conclusion

In this research computational fluid dynamics analyses is performed for an office room using ANSYS fluent to investigate the effects of better thermal comfort by changing of four-way cassette AC inlet position. The input parameters have been taken from the base paper [Oluleke Bamodu at el. 2017]. The governing equations such as continuity equation, momentum equation, energy equations, K-ε equations are used to perform this computational analysis. The Working fluid set as air with density 1.22 kg/m3, Specific Heat 1006.43 J/kg K, Thermal conductivity 0.24 W/m^2-K, Cold air inlet inside the room at inlet velocity 1.055 m/sec with 16 ^{o}C, to determine the temperature distribution used energy equation, under Discretization, select standard for Pressure, and second order for Momentum and Energy equation. The Fluent solver is used for CFD analysis. For that four CDA model of office room is designed using the CATIA software with approximate dimension of office room (6.8x3.8x3m and window size 3.4x2.4m) were considered. The four models created were Single AC inlet in office room from top, Double AC inlet in office room from top, Double AC inlet in office room from its side wall and Double AC inlet in office room one from top and other from side wall. There are following conclusive points drawn from this work.

- After performing computational fluid dynamic transient analyses with absolute velocity formulation using pressure-based solver for design-1. The temperature distributions inside the room is analyzed with interval of 10 and 50 sec which shows the total cooling time of 905 sec at the 0.8 m/sec air velocity.
- After performing computational fluid dynamic transient analyses for design-2. The temperature distributions inside the room is analyzed with interval of 10 and 50 sec which shows the total cooling time of 602 sec at the 0.8 m/sec air velocity and take 40.2% less time to achieved thermal comfort temperature as compared with model-1.
- After performing computational fluid dynamic transient analyses for design-3. The temperature distributions inside the room is analyzed with interval of 10 and 50 sec which shows the total cooling time of 573 sec at the 0.8 m/sec air velocity and take 44.93% less time to achieved thermal comfort temperature as compared with model-1 & 4.94% less time as compared with model-2.
- After performing computational fluid dynamic transient analyses for design-4. The temperature distributions inside the room is analyzed with interval of 10 and 50 sec which shows the total cooling time of 593 sec at the 0.8 m/sec air velocity and take 41.66% less time to achieved thermal comfort temperature as compared with model-1 & 1.5% less time as compared with model-2.

It has been observed from the above conclusion that the design-3 (Double AC inlet from its side wall) take 44.93% less time to achieved the comfort temperature as compared with model-1 (Single AC inlet from top) and 5% less time to achieved the comfort temperature as compared with model-2 (Double AC inlet one from top and other from side wall) has lest time to achieved the comfort temperature inside office room.

VII. Future Scope

This work focuses on improving thermal performance by changing the AC inlet position of the four-way cassette for an office space. Although the study is being conducted with great care, there is still room for improvement. The future studies may be possible.

1. Only four different models of office spaces using a four-way cassette system are used in this work, but other models can be used for future work.

2. In this work, only the input position of the four-way AC box is changed, but the output position can also change.
3. This work only focuses on optimizing the cooling time using the temperature and air velocity, but some other parameters can be used.

References

[1] Jéssica Kuntz Maykot at el. “Assessment of gender on requirements for thermal comfort in officebuildings located in the Brazilian humid subtropical climate” Contents available at Science Direct Energy and Buildings 158 (2018) 1170–1183.

[2] Jindal A “Thermal comfort study in naturally ventilated school classrooms in composite climate of India”, Building and Environment Accepted Date: 26 May 2018.

[3] Földváry V at el. “Development of the ASHRAE Global Thermal Comfort Database II”, Building and Environment Accepted Date: 12 June 2018, doi: 10.1016/j.buildenv.2018.06.022.

[4] M.T.H. Derks at el. “Understanding thermal comfort perception of nurses in a hospital ward work environment” Building and Environment 140 (2018) 119–127.

[5] Ricardo Forgiarini Rupp, Richard de Dear & Enedir Ghisi “Field study of mixed-mode office buildings in Southern Brazil using anadaptive thermal comfort framework“Energy and Buildings 158 (2018) 1475–1486.

[6] Shilei Lu, Bo Pang, Yunfang Qi & Kun Fang “Field study of thermal comfort in non-air-conditioned buildings in a tropical island climate” Applied Ergonomics 66 (2018) 89e97

[7] Elena Barbadilla Martat el. “Assessment of thermal comfort and energy savings in a Þeld study on adaptive comfort with application for mixed mode of offices, Energy & Buildings Accepted date: 16 February 2018.

[8] Vitor E.M. Cardoso at el. “A discussion about thermal comfort evaluation in a bus terminal, Energy & Buildings, Accepted date: 5 March 2018.

[9] Hye-Jin Cho & Jae-Weon Jeong “Evaluation of thermal comfort in an office building served by a liquid desiccant-assisted evaporative cooling air-conditioning system”, Energy & Buildings Accepted date: 6 May 2018.

[10] Andrei Claudiu Cosma & Rahul Simha “Thermal comfort modeling in transient conditions using real-time local body temperature extraction with a thermographic camera, Building and Environment 143 (2018) 36–47.