International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 5.22 (SJIF-2017), e-ISSN: 2455-2585
Volume 4, Issue 6, June-2018

Energy conservation and thermal environment analysis of room air conditioner with different locations of Supply Air flow

Abhinandan Kumar[1], V. N. Bartaria[2]

[1]*Mechanical Engineering Department, LNCT Bhopal*
[2]*Mechanical Engineering Department, LNCT Bhopal*

Abstract- Air-conditioning contributes significantly to high energy consumption in commercial buildings. It is essential to monitor and evaluate the performance of air-conditioning systems to avoid unnecessary energy wastage.This paper presents the results of astudy carried out to evaluate the performance of an actual building air-conditioning system. The results of the air-conditioning system'sperformance and thermal comfort analysis are presented and a comparison is made to the different locations of supply air flow.In this investigation various locations used for Air Conditioning air flow supply. Room designing and analysis performed in ANSYS CFD tool. Three rooms used with same dimensions at different locations of Air conditioner unit in room. Thermal conditions of room optimized with ANSYS CFD tool.

Keywords: air-conditioning control,thermal comfort, FEM, CFD Analysis.

I. Introduction

Refrigeration is a process of maintaining system temperature lower than that of surrounding temperature. The refrigeration cycle describes the changes that take place in the refrigerant as it alternately absorbs and rejects heat as it circulates through the refrigeration cycle. Heat naturally flows from hot to cold. Work is applied to cool a living space or storage volume by pumping heat from a lower temperature heat source into a higher temperature heat sink. Insulation is used to reduce the work and energy needed to achieve and maintain a lower temperature in the cooled space.The air conditioning is the process of simultaneous control of temperature, humidity, flow of air and purity of air. It needs the controlling the properties of atmospheric air and distributing through the space to be conditioned. Air conditioner which works on vapour compression refrigeration cycle consists of compressor, condenser (heat exchanger), expansion valve and evaporator (heat exchanger). Refrigerant is a working medium in refrigeration cycle used for heat transfer; it picks up heat (latent or Sensible heat) from the source at lower temperature and pressure and gives up heat (latent or Sensible heat) at higher temperature and pressure. Selecting refrigerant for a particular purpose its characteristics must be considered and the selection must be made on the basis of its compatibility with the system.

II. Need for Air Cooling

One of the main purposes of a building is to provide a comfortable environment for its occupants. In a modern industrial society, man spends the greater part of his life in indoors. During recent decades, this has resulted in higher interest in studying the influence of indoor climate. A reasonable thermal climate can be given by cooling the indoor air by some means which increase the productivity of the occupants.

Higher air temperature inside a building is found to reduce the productivity of the occupants and increase the heat stress inside the buildings. In poultry houses, higher temperature reduces feed consumption, weight gain, and egg yield. All these aspects demand, cooling of indoor air in hot climate zones. Air cooling is a method of achieving a comfortable indoor climate by reducing the air temperature, especially in the arid region of the world. In General, 25°C, 60 % RH is recommended comfort conditions. This can be achieved by using air conditioning system. Use of adiabatic air coolers is also very common.

III. Air conditioning system

Air conditioning is the process of conditioning the indoor air for the specified thermal comfort. An air conditioner is an appliance, system or machine, designed to stabilize the air temperature and humidity within an area. It is used for cooling as well as heating, depending on the air temperature at a given time, typically using refrigeration cycle. Sometimes, evaporative cooling is used for comfort cooling in building and motor vehicles. Air conditioning mainly is used for comfort and process application. Comfort application aims to provide a building indoor environment that means relatively constant in a range, preferred by humans, despite the changes in external weather conditions or in internal heat loads. Comfort applications for various building types are quite different and may be categorized as low- rise building, high- rise building, commercial building, institutional building and industrial space.Process applications aim to provide a suitable environment for a process being carried out, regardless of internal heat and humidity loads and external weather conditions. Process applications include Data Centers, Textile Industries, Chemical and Biological Laboratories, Mines and others. Air-conditioning in hot and arid environments, is an essential requirement for supporting of daily human activities.

IV. Finite Element Method

The finite element method is a powerful tool to obtain the numerical solution of wide range of engineering problem. The method is general enough to handle any complex shape or geometry, for any material under different boundary and loading conditions. The generality of the finite element method fits the analysis requirement of today's complex engineering systems and designs where closed form solutions of governing equilibrium equations are usually not available. In addition, it is an efficient design tool by which designers can perform parametric design studies by considering various design cases, (different shapes, materials, loads, etc.) and analyze them to choose the optimum design.

The method originated in the aerospace industry as a tool to study stress in a complex airframe structures. It grows out of what was called the matrix analysis method used in aircraft design. The method has gained increased popularity among both researchers and practitioners. The basic concept of finite element method is that a body or structure may be divided into small elements of finite dimensions called "finite elements". The original body or the structure is then considered, as an assemblage of these elements connected at a finite number of joints called nodes or nodal points.

V. Computational Fluid Dynamics

CFD is an area in which the governing equations for fluid flows are solved in discrete form on computers by simulating the fluid flow problem. This helps in reducing the time and effort required in narrowing down on the design configurations of various engineering components. In the present work a three dimensional numerical simulation was carried out to study the effects of different Ac rooms by using various locations of AC duct. The simulation of flow in internal combustion engines is a complex process involving fluid dynamics and turbulent motion. Any physical fluid flow problem can be solved either experimentally or numerically. The numerical simulation is more suitable for parametric studies and it also gives accurate results by solving governing equations in each and every cell of the fluid domain. In the recent years there were tremendous improvement in the field of numerical techniques, which made a great impact on the evaluation of complex flow problems and achieving their solution. Computational Fluid Dynamics (CFD) has grown from a mathematical curiosity to an essential tool in almost every branch of fluid dynamics. CFD is considered as a bridge between the pure experimental fluid dynamics and pure theoretical fluid dynamics. Until recently, researchers had to mainly depend on time consuming and costly experimentations for studying complex flow problems. With the advent of powerful digital computers and numerical simulation techniques, the extent of experimentation required for analyzing complex engineering problems has been reduced considerably.

VI. ANSYS CFD (Fluid flow) Workbench Environment

The ANSYS CFX (fluid flow) software is fully integrated into the ANSYS Workbench environment, the framework for the entire suite of engineering simulation solutions. Its adaptive architecture enables users to easily set up anything from standard fluid flow analyses of complex interacting systems with simple drag-and-drop operations. Fig 5.1 shows project schematic with ANSYS system of fluid flow CFX. Users can easily assess performance at multiple design points or compare several alternative designs. Within the ANSYS Workbench environment, applications from multiple simulation disciplines can access tools common to all, such as geometry and meshing tools.Most important physical models available in ANSYS CFX software useful to us are:

- **Heat Transfer and Radiation:** Beyond solving the convective transport of energy by fluid flow, ANSYS CFX software includes a conjugate heat transfer (CHT) capability to solve the thermal conduction in solids. It also incorporates a wealth of models to capture all types of radiative heat exchange in and between fluids and solids, whether these are fully or semitransparent to radiation or opaque.

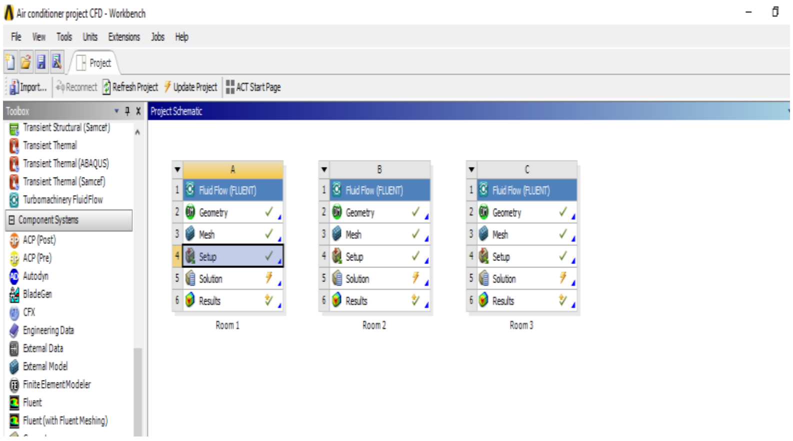


Figure 1: ANSYS workbench project schematic with fluid flow (CFD)

VII. Create a Geometry

All engineering simulations start with geometry to represent the design, be it a solid component of a structural analysis or the air volume of a fluid or electromagnetic field. The engineer either has geometry that has been created in a CAD (computer-aided design) system or builds the geometry from scratch. The ANSYS Design Modeler is a gateway to geometry handling for an ANSYS analysis. Geometry created using ANSYS Design Modeler software which is specifically designed for the creation and preparation of geometry for simulation. In engineering simulations, the geometry includes details not needed for simulation. Only the physics involved is to be included, simulating such a fully detailed model will increase solver run times. It can be more efficient to spend a short time removing these details to reduce the total run time by hours or days. Fig 4.2, 4.3 and 4.4 shows the geometry created of A. C. Room with duct location and various features of Design modeler and outline tree with Air conditioning room with dimension of room are explained in table.

Table 1: Dimension Parameters

Parameters	Room dimensions (meter)	Dimension of Duct(meter)
Length	5	0.5
Width	3.65	1.2
Height	3	0.4

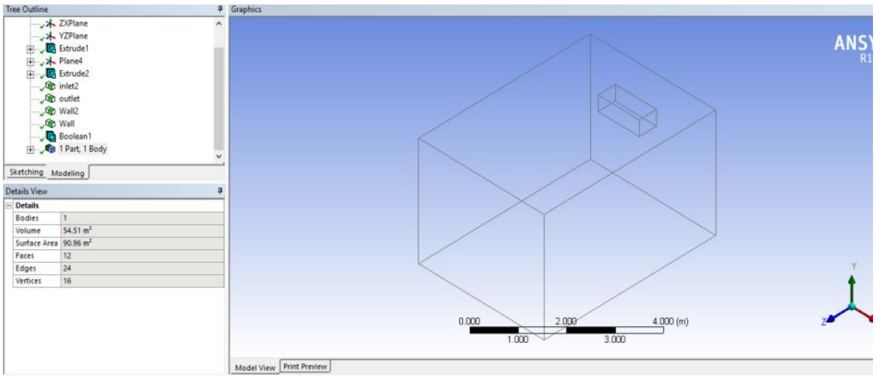


Figure 2: Geometry of Room 1 with A. C. Dust

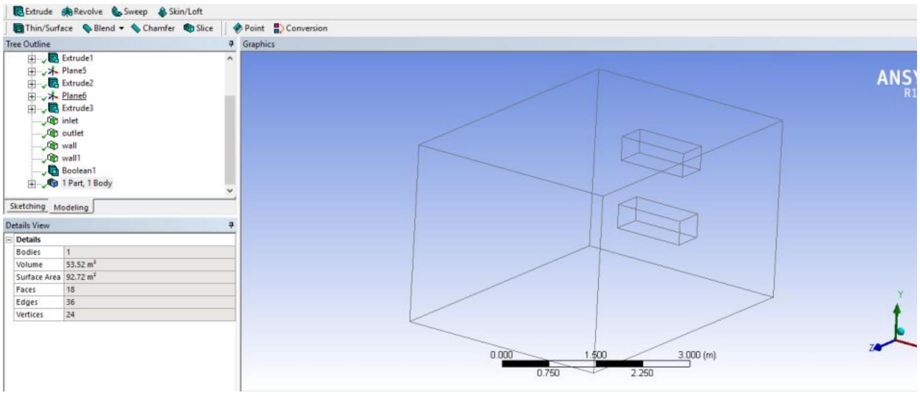


Figure 3: Geometry of Room 2 with A. C. Dust

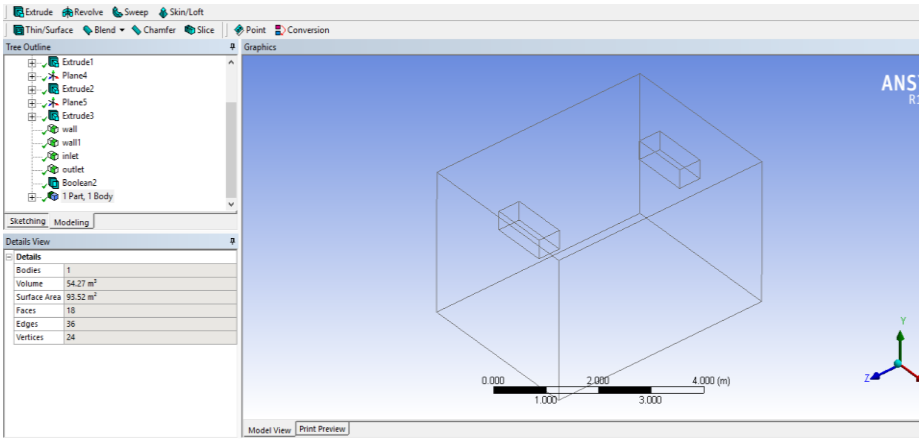


Figure 4: Geometry of Room 3 with A. C. Dust

- **Meshing of Geometry (discretization of the computational domain)**

One of the tedious and time taking event of the CFD simulation is the discretization of the computational domain. It is also famous as Mesh generation. Analytical solutions to the Navier-Stokes equations exist for only the simplest of flows under ideal conditions. To obtain solutions for real flows, a numerical approach must be adopted whereby the equations are replaced by algebraic approximations which may be solved using a numerical method, by using the approach of 'Discretization of the Governing Equations' involves fragmenting the spatial domain into small finite control volumes using a mesh. The governing equations are integrated over each control volume, such that the relevant quantity (mass, momentum, energy, etc.) is conserved in a discrete sense for each control volume.

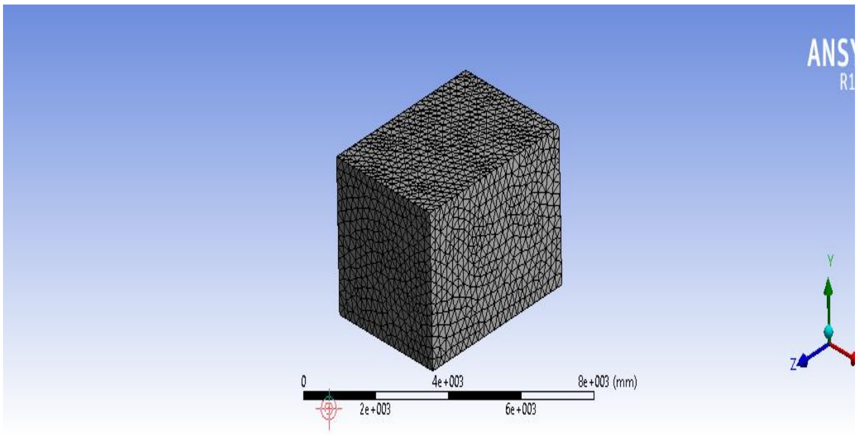


Figure 5 shows the separate view of the meshed AC room

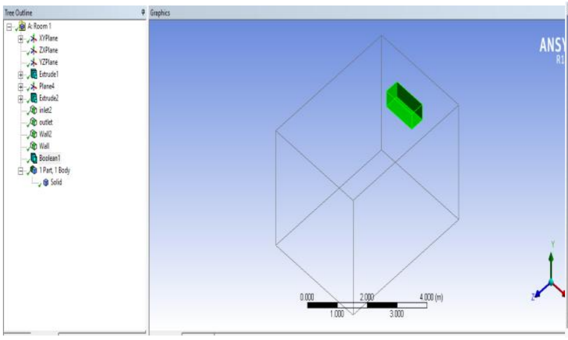
VIII. Air Conditioning Room designing in ANSYS

Computer simulations offer an exciting opportunity for validating aspects of ventilation, smoke movement, natural air flow and thermal comfort right at the design stage. In this work CFD simulation of forced ventilation on A.C. room was carried and thermal comfort of the human being is studied using ANSYS Fluent. Results shows that for Air conditioning room having variations in ac duct locations.

same room dimensions 5 m x 3.65 m x 3 m for all room designing used and room temperature is approximately 313K (40^{0}C) which is higher than comfort temperature due to the flow stagnant zone back side of the room. three type cases used for study and room dimensions were same during analysis, in case 1 AC dust is located 9 feet from base and 1 tonne air conditioning unit used in this study, temperature of air at inlet point of AC dust is 291K (18^{0}C) with velocity of Air is 0.39 m/s.

- **Case 1**

In case 1 Room designed with dimensions of 5 m x 3.65 m x 3 m and AC duct is located at 9 feet from base and Room temperature is 313K and temperature of Air at inlet point of AC duct is 291K, Air velocity is 0.39 m/s. Figure shows the arrangement of AC room with single duct.



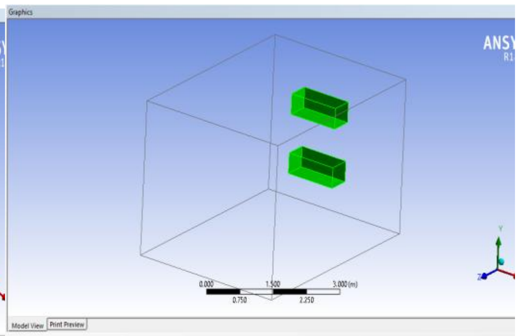


Figure 6: Case 1 of AC room Figure 7: Case 2 of AC room

- **Case 2**

In this case Room designed with 5 m x 3.65 m x 3 m and duct is divided here in two part; each duct is used with 0.19 m/s air flow velocity. first duct is placed 3 feet from base and second duct placed 6 feet above from the first duct.Room temperature is 313K and temperature of Air at inlet point of AC duct is 291K, over all Air velocity of both duct is 0.39 m/s. Figure shows the arrangement of AC room with double duct.

- **Case 3**

In this case Room designed with 5 m x 3.65 m x 3 m and duct is divided here in two part; each duct is used with 0.19 m/s air flow velocity. Both AC Dust place in front of each other in room wall.

first duct is placed 9 feet from base on left wall of the room and second duct placed 9 feet above from the base on the right side wall.

Room temperature is 313K and temperature of Air at inlet point of AC duct is 291K, over all Air velocity of both duct is 0.39 m/s. Figure shows the arrangement of AC room with double duct. Figure shows the arrangement of AC room with double duct.

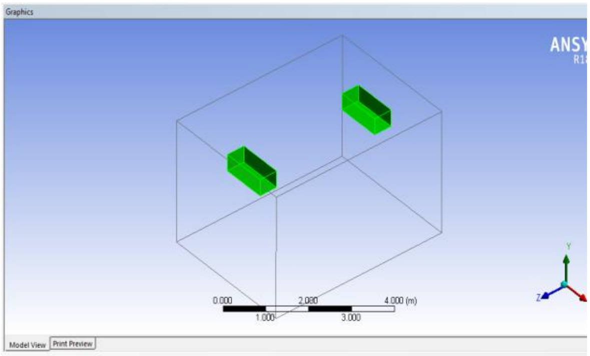


Figure 8: Case 3 of AC room

IX. Results of Air conditioning

- **Case 1 of AC Room**

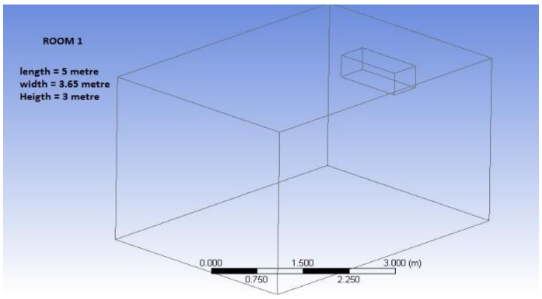


Figure 9: Room 1 Designed in ANSYS

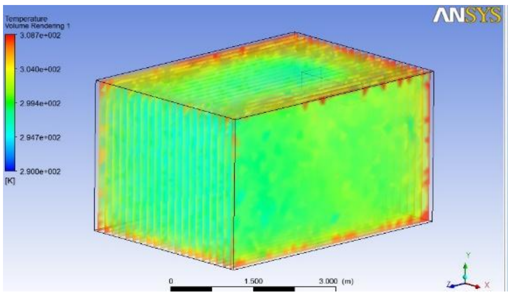


Figure10: Temperature variations in Room 1

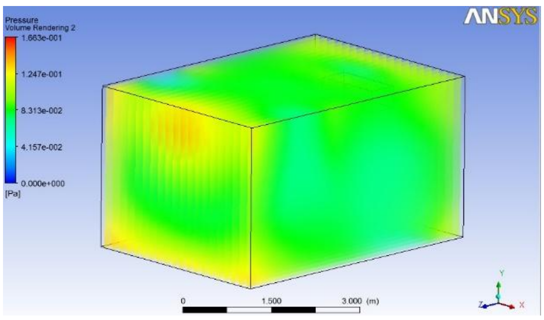


Figure 11: Pressure Generated due to Air flow in Room 1

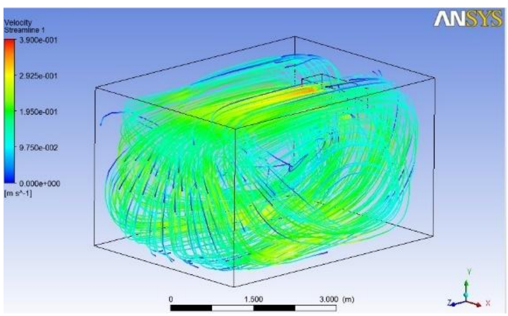


Figure 12: Streamline of Air flow in Room 1

- **Case 2 of AC room**

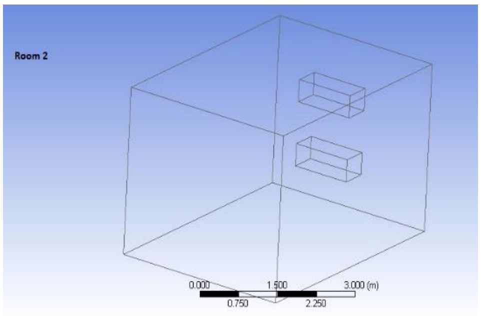


Figure 13: Room 2 Designed in ANSYS

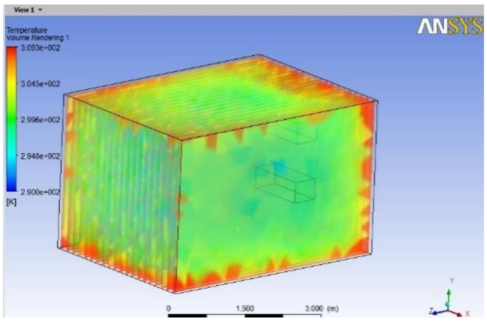


Figure 14: Temperature variations in Room 2

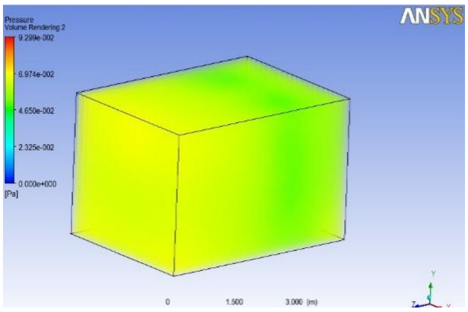


Figure 15: Pressure Generated due to Air flow

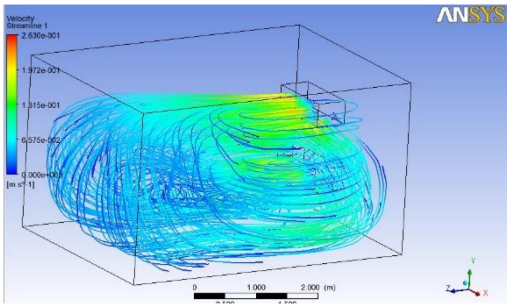


Figure 16: Streamline of Air flow

- **Case 3 of AC Room**

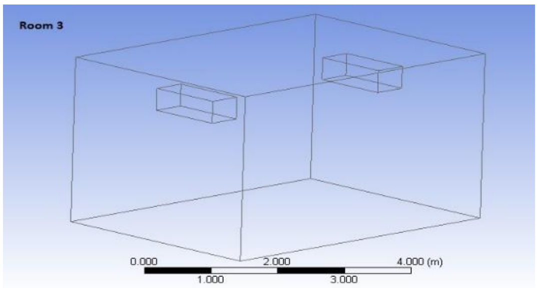


Figure 17: Room 3 Designed in ANSYS

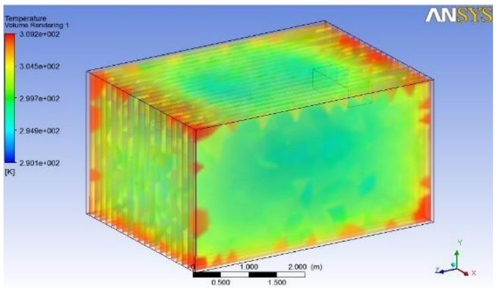


Figure 18: Temperature variations in Room 3

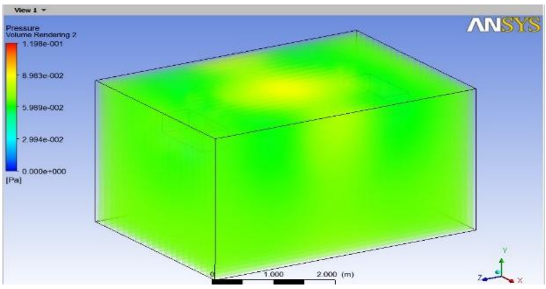


Figure 19: Pressure Generated due to Air in Room 3

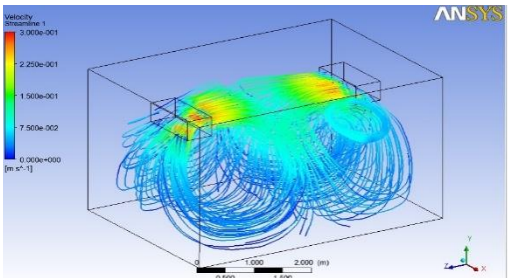


Figure 20: Streamline of Air flow in Room 3

Above graph and figures shows the variations of AC Rooms temperatures, Pressure of Air in Room and effects of room during Air circulation by AC duct. Table 5.1 shows the comparison of All three cases of room study with different Duct positions.

Table 2: Comparison of Results with all three cases of AC Room

Rooms	Minimum Temperature of Room	Pressure	Time of Cooling seconds
Room 1	21.7	0.166	140
Room 2	21.8	0.092	109
Room 3	21.9	1.198	91

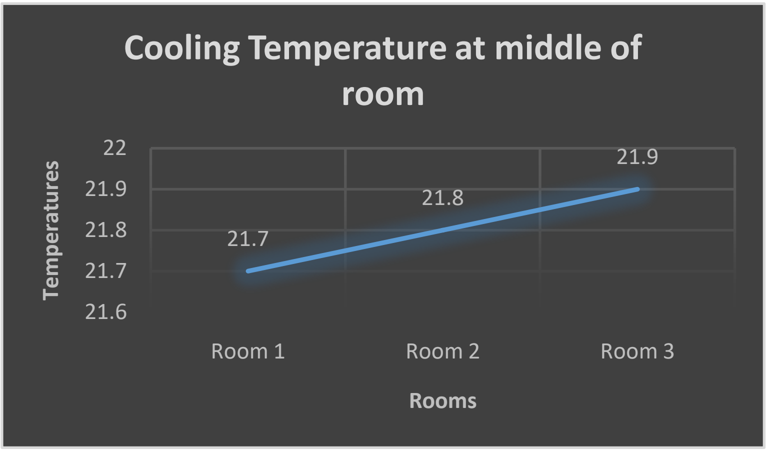


Figure 5.13: Comparison of Temperature b/w All Room with different duct locations

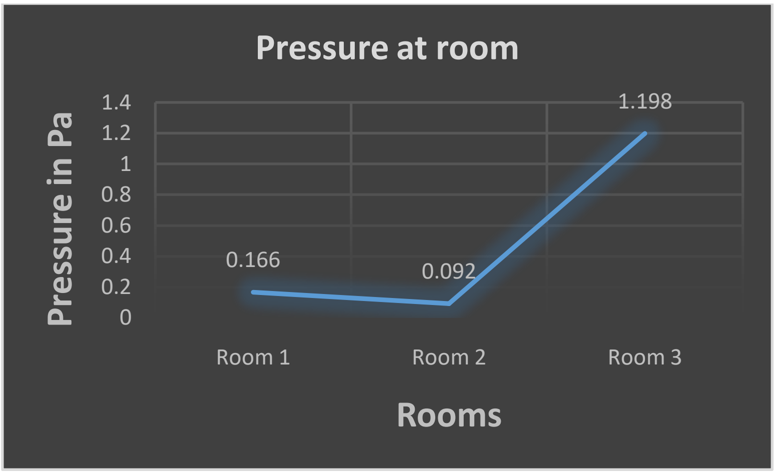


Figure 5.14: Comparison of pressure b/w All Room with different duct locations

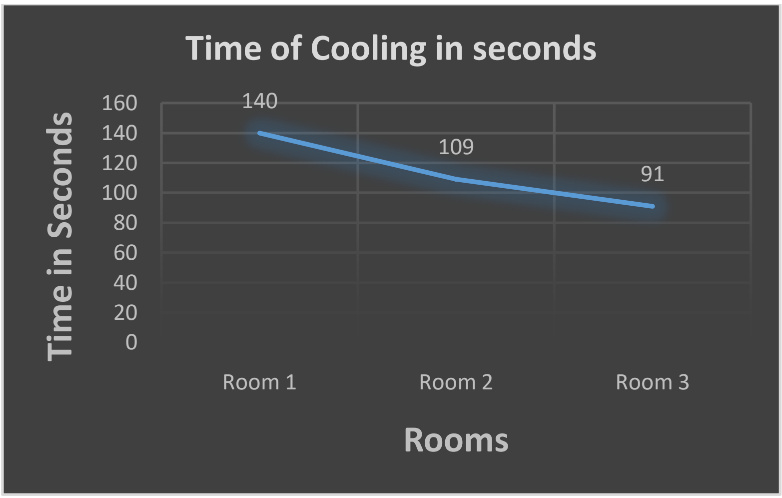


Figure 5.15: Comparison of Cooling time b/w All Room

As per Above Study it is concluded that Room1 having maximum time for cooling of room is 140 Second and temperature found that is 21.7^0C. In Room 2 conditions double duct used for air flow but mass flow rate of air kept constant so after study in room 2 maximum time 109 second for temperature 21.8^0C found. In Room 3 double duct used in front of each other and maximum temperature found 21.9 and maximum time taken during temperature drop is 91 second. So case 3 Room with double AC duct but same mass flow rate is better arrangement for early cooling system because it takes less time of cooling room.

X. Conclusion

In the present work, an extensive data analysis has been made to study the cooling potential for AC Duct location, using the three different location of duct in Same dimension rooms, optimize suitable method of cooling by comparing the results of This study. Following conclusions are made as per above study.

- While there are several experimental investigations performed by researchers regarding the Air conditioning of Rooms. On the basis of 3-D finite element modeling of conventional room design. The obtained data showed how room temperature drops with respect to time during CFD analysis using FLUENT was considered. The modeling, meshing, preprocessing and analysis portion of the work is performed in ANSYS V14.0.
- And the results obtained under boundary conditions taken into consideration are discussed. The results obtained during analysis are shown above, it can be concluded that the Room 3 conditions has shown better results because it take less time to make comfort temperature from 40 ^{0}C to 21.9 ^{0}C.
- it is concluded that Room1 having maximum time for cooling of room is 140 Second and temperature found that is 21.7^0C. In Room 2 conditions double duct used for air flow but mass flow rate of air kept constant so after study in room 2 maximum time 109 second for temperature 21.8^0C found. In Room 3 double duct used in front of each other and maximum temperature found 21.9 and maximum time taken during temperature drop is 91 second. So case 3 Room with double AC duct but same mass flow rate is better arrangement for early cooling system because it takes less time of cooling room.

References

1. Zhou, Zhong, Liu, "Study on the Relationship between Thermal Comfort and Air-Conditioning Energy Consumption in Different Cities", Journal of Computers, Vol. 28, No. 2, 2017, pp. 135-143.
2. Jing Ni, Bowen Jin, Bo Zhang, "Simulation of Thermal Distribution and Airflow for Efficient Energy Consumption in a Small Data Centers", Sustainability, 2017.
3. Kanga, Peng, Chenga, "Analysis of Condensation and Thermal Comfort of Two Kinds of Compound Radiant Cooling Air Conditioning Systems Based on Displacement Ventilation", 10th International Symposium on Heating, Ventilation and Air Conditioning, ISHVAC2017, October 2017.
4. Bamodu, Xia, Tang, "A numerical simulation of air distribution in an office room ventilated by 4-way cassette air conditioner", ICAE, Elsevier, 2016.
5. Ding, Guo, Chen, "Design and simulation of an air conditioning project in a hospital based on computational fluid dynamics", Archives of civil engineering, Volume 13, Issue 2, 2017.
6. DU, Lei, Chena, "The indoor thermal environment simulation and testing validation of a power plant turbine room in extreme cold area", 8th International Cold Climate HVAC Conference, Elsevier, 2016.
7. Popovicia, Hudisteanua,"Numerical simulation of HVAC system functionality in a sociocultural building", International Conference Interdisciplinary in Engineering, Elsevier, 2016.
8. Aryal, Leephakpreeda, "CFD analysis on thermal comfort and indoor air quality affected by partitions in air-conditioned building", Applied Mechanics and Materials, Vol. 836, pp 121-126, 2016.
9. Satyam, Jagtap, Archana, "Design and Development of Portable Air Conditioner", International Journal for Research in Engineering Application & Management, Volume 02, Issue 7, 2016.
10. SudhangshuSarma, O. P. Jakhar,"Computational Analysis of Impact of the Air-Conditioner Location On Temperature and Velocity Distribution inan Office-Room", International Research Journal of Engineering and Technology (IRJET), Volume: 03, Issue: 09, Sep -2016.

11. Pillai, Bhand, Shinde, "A Review on CFD Analysis in Air-Conditioning System", International Journal of Current Engineering and Technology, 2016.
12. D prakash, "Transient analysis and improvement of indoor thermal comfort for an air-conditioned room with thermal insulations", Ain Shams Engineering Journal, Elsevier, 2015.
13. Thakur,Patel, Parth, "Quantification of Air Flow Pattern in Air Conditioned Room – A Review", International Journal of Advance Engineering and Research Development Volume 1,Issue 12, December -2014.
14. Calautit, Hughes, "Wind tunnel and CFD study of the natural ventilation performance of a commercial multi-directional wind tower", Building and Environment, Volume 80, October 2014, PP 71-83.
15. Mallikarjun, Malipatil, "CFD Analysis of Air Cooled Condenser by Using Copper Tubes and Aluminum Fins", International Journal for Research in Applied Science & Engineering Technology, Volume 2 Issue X, October 2014.
16. Vladut, Sbirna, Sebastian, "CFD simulation of the airflow pattern within a three-bed hospital room with or without a portable air conditioner in use", 18th International Conference on System Theory, Control and Computing, IEEE, 2014.