# CFD ANALYSIS OF ROOM WITH AIR CONDITIONER BY USING ANSYS WORKBENCH

Abhinandan Kumar[1], V. N. Bartaria[2]
[1]Mechanical Engineering Department, LNCT Bhopal
[2]Mechanical Engineering Department, LNCT Bhopal

**Abstract: -** The experimental analysis has been performed within two equal varieties of room for air conditioning and studied distinct duct locations in the room and optimize minimum room cooling time. The consequences of the air-conditioning device's overall performance and thermal consolation evaluation are presented and an evaluation is made to the exceptional locations of delivering air flow. In this research, numerous locations utilized for Air Conditioning air go with the flow supply. two rooms designed with the same dimensions at different locations of the Air conditioner unit in the room. Room designing and evaluation achieved in ANSYS CFD tool. Thermal situations of room optimized with ANSYS CFD Simulation tool.

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

## I. Introduction

Air conditioning is the manner of conditioning the indoor air for the specified thermal comfort. An air conditioner is an equipment, gadget or system, designed to stabilize the air temperature and humidity inside an area. It is utilized for cooling as well as heating, relying at the air temperature at a given time, generally using refrigeration cycle. Sometimes, evaporative cooling is utilized for consolation cooling in building and motor vehicles. Air conditioning in particular is utilized for consolation and manner software. Comfort software objectives to offer a constructing indoor surroundings meaning particularly steady in a range, preferred through human beings, no matter the adjustments in outside climate conditions or in inner heat hundreds. Comfort programs for diverse building sorts are quite exceptional and may be labeled as low- upward thrust building, high- rise building, business building, institutional constructing and business space. Process programs purpose to provide a suitable surroundings for a manner being performed, irrespective of internal warmness and humidity hundreds and outside climate conditions. Process packages consist of Data Centers, Textile Industries, Chemical and Biological Laboratories, Mines and others. Air-conditioning in warm and arid environments, is an essential requirement for supporting of each day human activities.

## II. Computational Fluid Dynamics

CFD is a place wherein the governing equations for fluid flows are solved in discrete shape on computers by way of simulating the fluid glide trouble. This enables in reducing the time and effort required in narrowing down on the design configurations of diverse engineering additives. In the present work a 3 dimensional numerical simulation became carried out to have a look at the results of different Ac rooms by way of using various places of AC duct. The simulation of float in inner combustion engines is a complicated procedure concerning fluid dynamics and turbulent motion. Any physical fluid flow trouble can be solved either experimentally or numerically. The numerical simulation is greater appropriate for parametric studies and it also offers accurate consequences through solving governing equations in every and each cellular of the fluid area. In the current years there have been incredible improvement in the subject of numerical strategies, which made a remarkable effect on the evaluation of complex float issues and reaching their solution. Computational Fluid Dynamics (CFD) has grown from a mathematical curiosity to a critical tool in nearly every branch of fluid dynamics. CFD is considered as a bridge between the natural experimental fluid dynamics and pure theoretical fluid dynamics. Until currently, researchers needed to mainly rely upon time consuming and pricey experimentations for analyzing complicated float troubles. With the advent of effective virtual computers and numerical simulation strategies, the quantity of experimentation required for reading complicated engineering problems has been reduced considerably.

## III. Modelling of Room design in Ansys

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. The 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.

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 1 and 2 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 |

- **Case Study 1 of AC Room**

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 utilized 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.

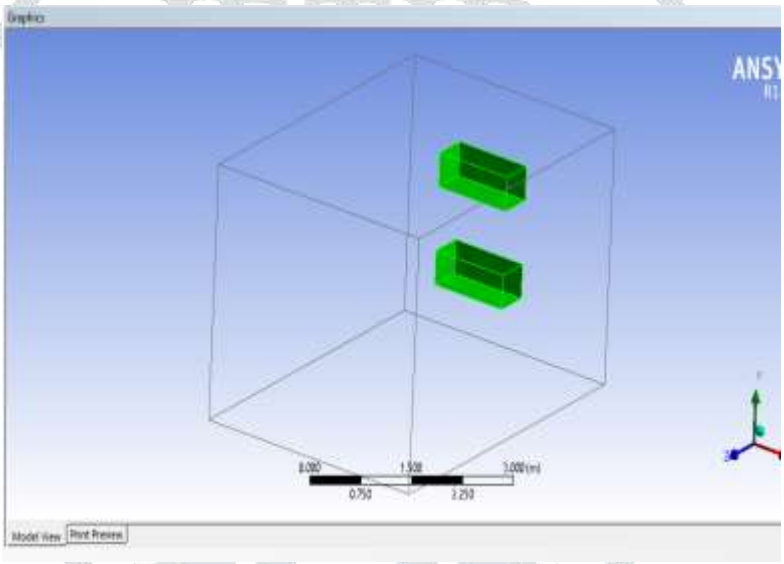

Figure 1: Geometry of Room 2 with A. C. Dust

- **Case Study 2 of AC Room**

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 utilized 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 2 shows the arrangement of AC room with double duct.

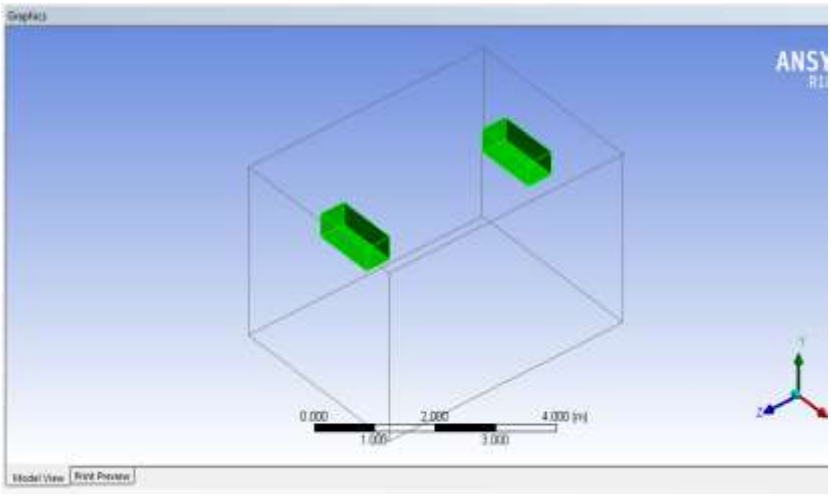

Figure 2: Geometry of Room 2 with A. C. Dust

## IV. Results and Discussion

- **Case 1 of AC room**

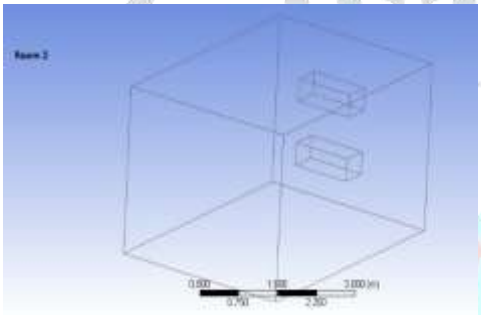

Figure 3: Room 1 Designed in ANSYS

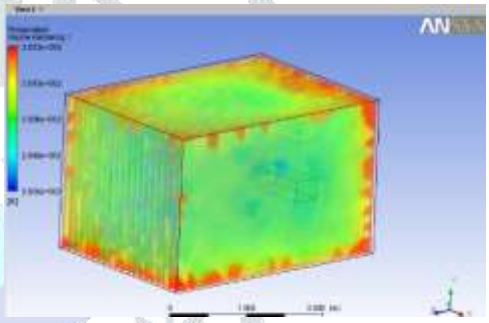

Figure 4: Temperature variations in Room 1

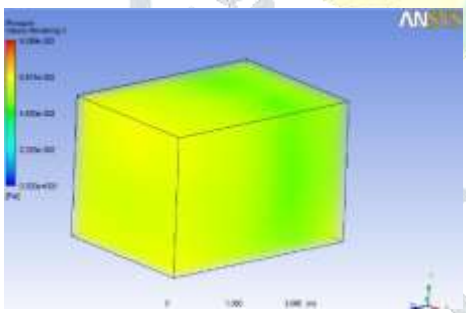

Figure 5: Pressure Generated due to Air flow in Room 1

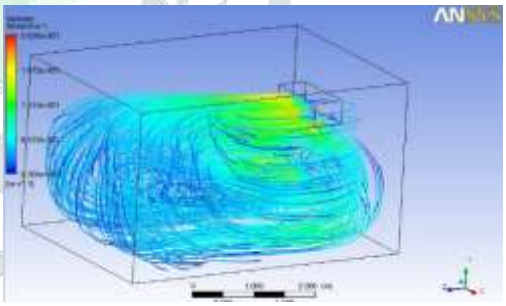

Figure 6: Streamline of Air flow in Room 1

- **Case 2 of AC Room**

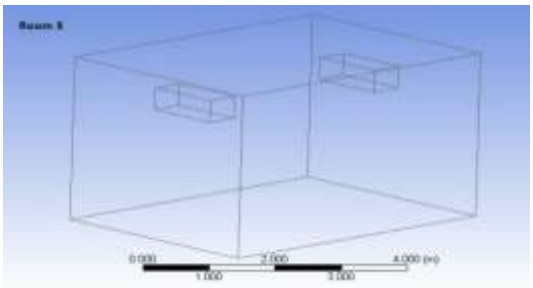

Figure 7: Room 2 Designed in ANSYS

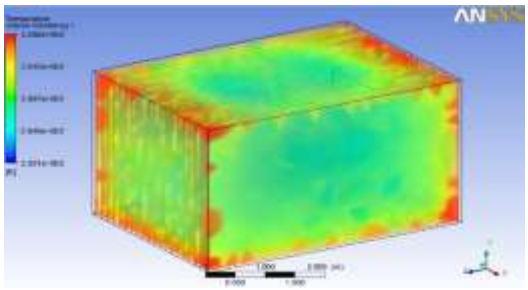

Figure 8: Temperature variations in Room 2

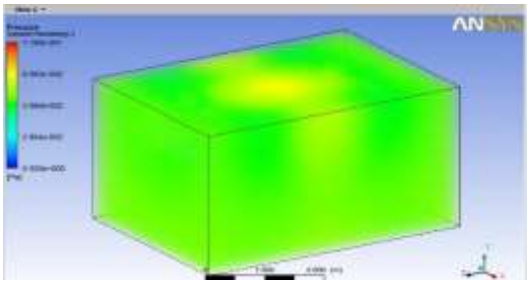

Figure 9: Pressure Generated due to Air in Room 2

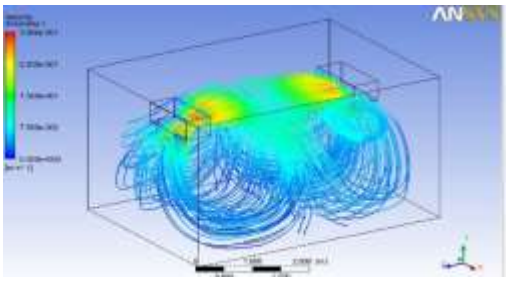

Figure 10: Streamline of Air flow in Room 2

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 1 shows the Temperature difference comparison of All two cases of room study with different Duct positions. Table 2 shows pressure difference and table 3 shows time of cooling in seconds in Room 1 and Room 2.

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

| Rooms | Minimum Temperature of Room |
|---|---|
| Room 1 | 21.8 |
| Room 2 | 21.9 |

Table 3: Pressure difference in Room 1 and Room 2

| Rooms | Pressure |
|---|---|
| Room 1 | 0.092 |
| Room 2 | 1.198 |

Table 4: Cooling Time in Room 1 and Room 2

| Rooms | Time of Cooling seconds |
|---|---|
| Room 1 | 109 |
| Room 2 | 91 |

In Room 1 conditions double duct utilized for air flow but mass flow rate of air kept constant so after study in room 1 maximum time 109 second for temperature 21.8 $^0$C found. In Room 2 double duct utilized in front of each other and maximum temperature found 21.9 and maximum time taken during temperature drop is 91 second. So case 2 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.

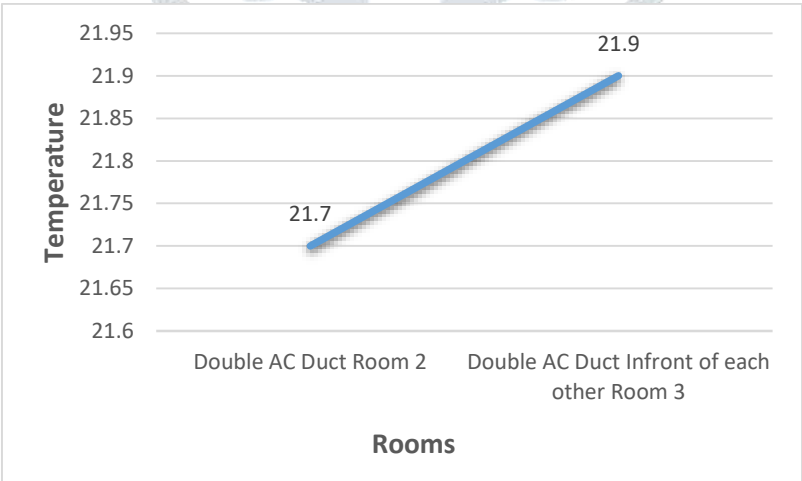


Figure 11: Temperature difference in Room 1 Vs Room 2

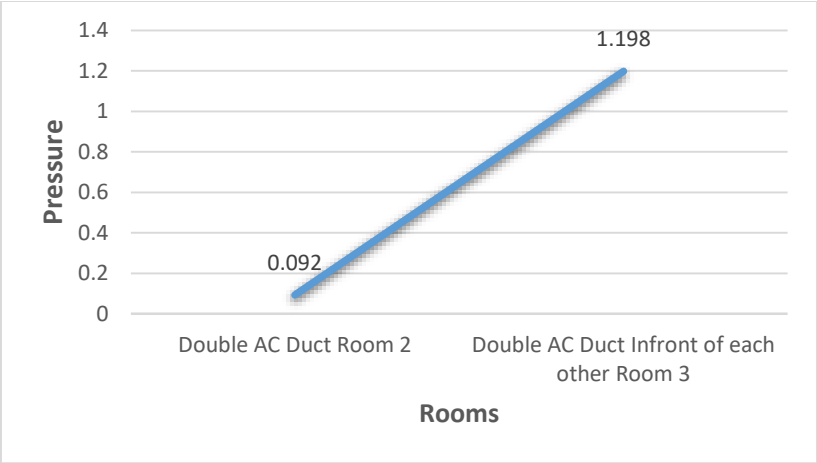


Figure 12: Pressure difference in Room 1 Vs Room 2

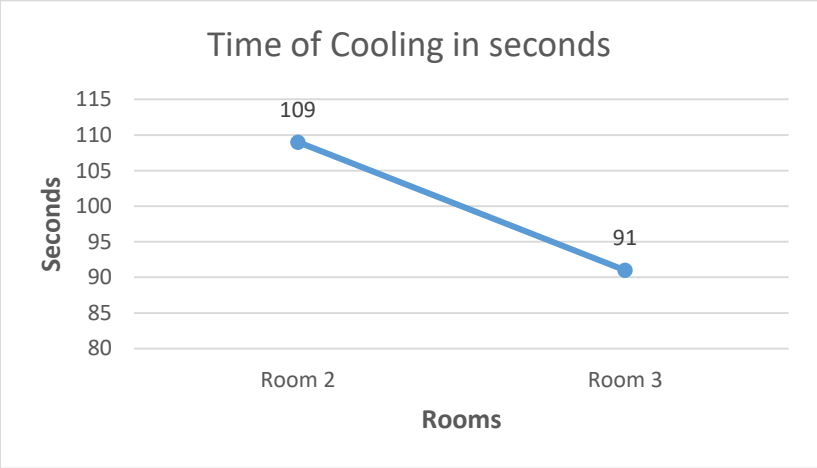


Figure 13: Colling time of Room 1 Vs Room 2

## V. 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 2 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 in Room 1 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 $^0$C found. In Room 2 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 2 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. Sudhangshu Sarma, O. P. Jakhar,"Computational Analysis of Impact of the Air-Conditioner Location On Temperature and Velocity Distribution in an 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.