ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

COMPARISON BETWEEN SIMULATION AND EXPERIMENTAL TECHNIQUE INSIDE AN AIR CONDITIONED ROOM TO ANALYZE THE PERFORMANCE AND FLOW CHARACTERISTICS OF CEILING SWIRL DIFFUSER UNDER DIFFERENT OPERATING AND FLOW PARAMETERS

Rahul Pandey*, Prof. Anil Kumar Rao, Prof. Vinay Yadav

PG Student, of Thermal Engineering (Mechanical Engineering Department) AISECT UNIVERSITY Bhopal, (M.P), India

Head and Professor, Department of Mechanical Engineering, AISECT UNIVERSITY Bhopal,(M.P), India

ABSTRACT

In this dissertation a detailed comparision between the simulation (CFD) and experimental technique was done in an air conditioned room in order to analyze the incoming air flow pattern and to improve the quality of air which is coming inside the room. An experiment is performed in the workshop of LNCT Bhopal, to find out the air flow pattern and its distribution through different swirl diffuser having different slot angle, installed at the ceiling in an air conditioned room. It is basically performed to evaluate the thermal comfort produced in a room equipped with a heat load of 1500W load capacity with different swirl diffusers and compare their performance graphically. My project is to do the same project but with a different technique i.e CFD (Computational Fluid Dynamics) and to compare the results which has come out from the experimental with the results that is coming out from the CFD. A set up mainly consist of a wooden room of size 4ft X 4ft X 5ft with different models of swirl diffusers installed at the ceiling level. The air supplied from an air conditioner through a swirl diffuser which is placed at the bottom. There are six different location at the floor inside the room where reading is to be taken of temperature and there variation with respect to height. There are six locations at floor i.e X1,X2,X3,Y1,Y2 and Y3. The result from this desertation work is that the simulation technique (CFD) gives more accurate and consistent result as compare to the experimental technique. Velocity distribution inside the room and the graphs are plotted at six different locations inside the room, which shows the comparision between Experimental [24] and CFD technique. In this dissertation work it is analyzed that the experimental technique is time consuming and also the results are also not consistent so it is beneficial to use the simulation technique in case of flow analysis.

KEYWORDS: Conventional Room Air-Conditioner, swirl diffusior, CFD Performance.

INTRODUCTION

Air conditioning is the process of changing the propertiesof air (primarily temperature and humidity) to provide favorable and comfort conditions. Most generally, air conditioning refer to cooling, heating, ventilation, or disinfection that changes the condition of air. An air conditioner is a major home appliance system, designed to change the air temperature and humidity within an area (used for cooling and sometimes heating depending on the air properties at a given time). The cooling is usually done by using simple refrigeration, but sometimes evaporation is used, commonly for cooling in buildings and motor vehicles to provide comfortable enviorment. In construction, a complete system of heating, ventilation and air conditioning to provide thermal comfort is referred to as "HVAC". Air conditioning is provided by a simple cooling process which uses pumps to circulate a coolant (typically water or a glycol mix) from a cold source, which in turn acts as a heat sink for the energy which is removed from the cooled space. Free cooling systems can have efficiencies very high and are sometimes combined with seasonal thermal energy storage (STES) so that the cold in winter can be used for summer air conditioning. Common storage mediums are deep aquifers or natural underground rock mass accessed via a cluster of small-diameter, heat exchanger equipped boreholes etc. Some systems with small storages are hybrids, using free cooling early in the cooling season, and later used as a heat pump to chill the circulation coming out from the storage. The heat pump is added because the temperature of the storage gradually increases during the cooling season, there by declining in efficiency. Free cooling and hybrid systems are mature technology for cooling..It is the process of controlling the temperature, humidity, cleanliness and air motion inside the room simultenously. In case of the machine components, along with temperature, moisture content in the air must also be controlled to provide a comfort enviorment to the human being along with these two important parameters, air motion and

METHODOLOGY

CFD ANALYSIS OF SET UP:-

Preprocessing:

CAD Model: Generation of 3D CAD model of set up using Unigraphics . Then import the CAD model into Ansys Design modeler in parasolid format (.xtl or .xt).

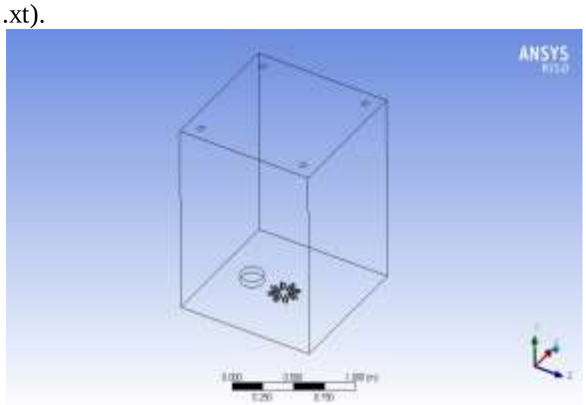

FIG 2.1:- CAD model of the set up

Mesh**:** Generate the mesh of ACC in the Ansys Mesh software .

Mesh Type: tetrahedral

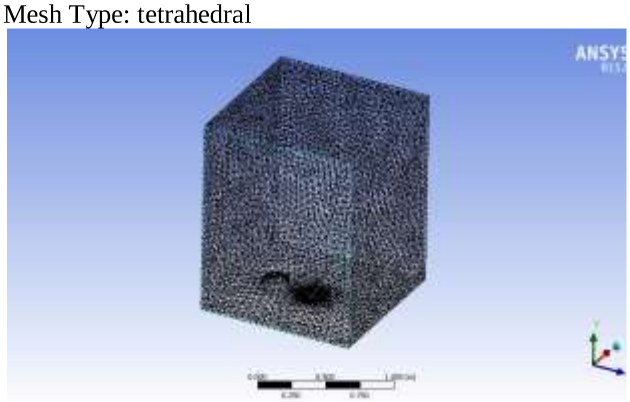

FIG 2.2 Meshing of set up

Element Edge Length =2.5 mm

No. of Nodes = 28125

No. of Element = 146404

Fluent setup: After mesh generation define the following setup in the Ansys fluent.

cleanliness also play a vital role. Air conditioning, therefore, is a very broader aspect which looks into the immediate control all of mechanical parameters which are necessary for the comfort of human beings or animals or for the proper presentation of some industrial or systematic process. In some applications, even the control of air pressure falls under the purview of air conditioning. It is to be noted that refrigeration that is control of temperature is the most important aspect of air conditioning.

- **Problem Type : 3D**
- **Type of Solver: Pressure-based solver.**
- **Physical model: Viscous: K, e two equation turbulence model.**
- **Material Property: Flowing fluid is air**

  Density of air = 1.225 kg/m^3
  Viscosity = 1.7894e-05

- **Boundary Condition:**

  Operating Condition: Pressure = 101325 Pa

  Inlet: Velocity inlet – Velocity = 2 m/s
  Turbulent intensity = 1%
  Hydraulic Dia. = 280mm
  Heat flux = 325 W/m^2

  Outlet: Pressure outlet: Define the same outlet condition for all the vent outlet
  Gauge pressure = 101325 Pa
  Turbulent intensity = 1%
  Hydraulic Length. = 50mm

Solution:

- **Solution Method :**
  Pressure- velocity coupling – Scheme -SIMPLE
  Pressure – Standard
  Momentum – Second order
  Turbulent Kinetic Energy (k) – First order
  Turbulent Dissipation Rate (e) - First order
- **Solution Initialization: Initialized the solution to get the initial solution for the problem.**

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

- **Run Solution: Run the solution by giving 2000 no of iteration for solution to converge.**

2.3 Post processing.

- **Post Processing:** For viewing and interpretation of Result. The result can be viewed in various formats: graph, value, animation etc.

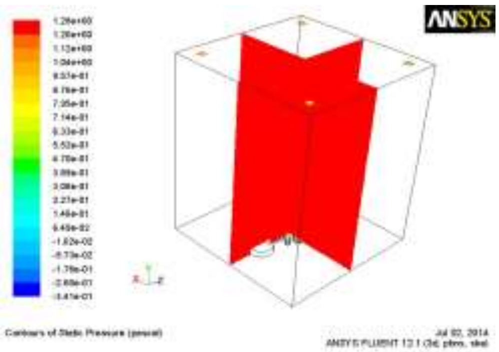

Fig 2.3 Pressure Contour At 7^0 Swirl Angle

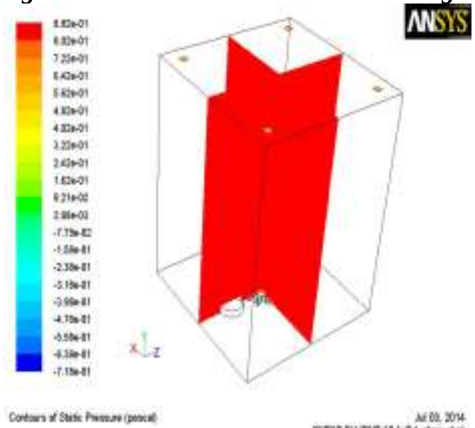

Fig 2.4 Pressure Contour At 8^0 Swirl Angle

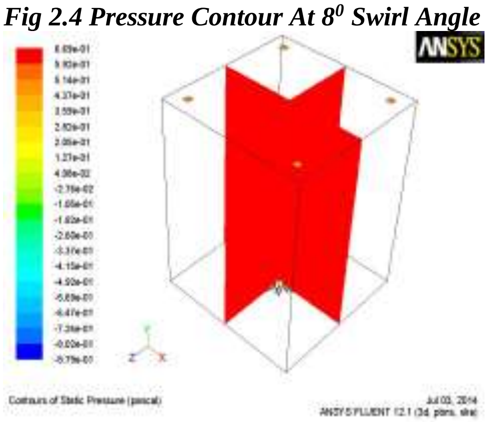

Fig 2.5 Pressure Contour At 9^0 Swirl Angle

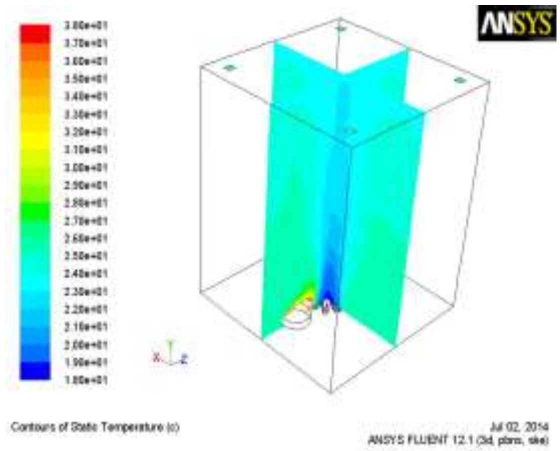

Fig 2.6 Tempreture Contour At 7^0 Swirl Angle

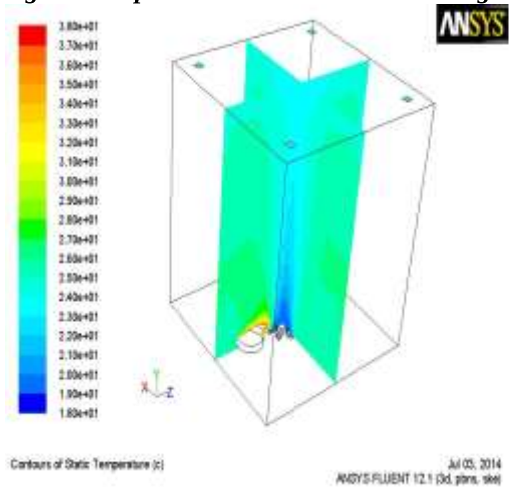

Fig 2.7 Temperature Contour At 8^0 Swirl Angle

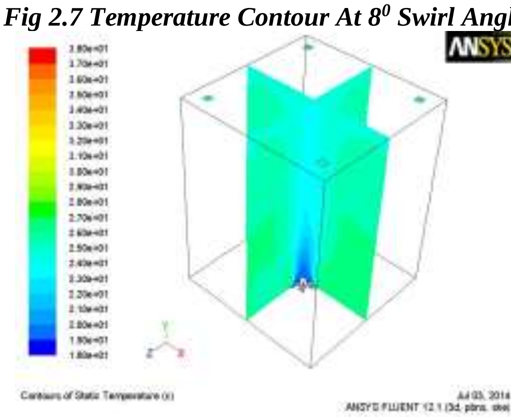

Fig 2.8 Temperature Contour At 9^0 Swirl Angle

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

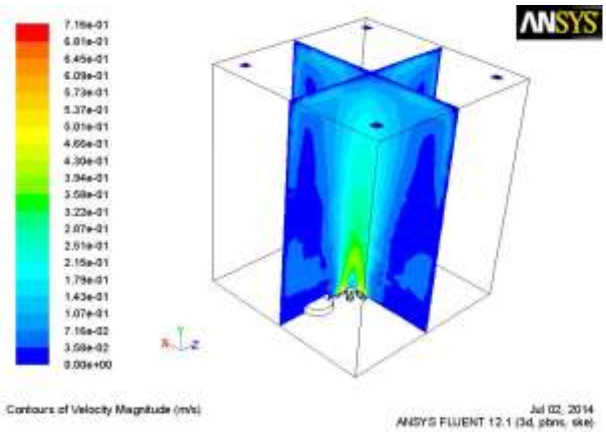

Fig 2.9 Velocity Contour At 7^0 Swirl Angle

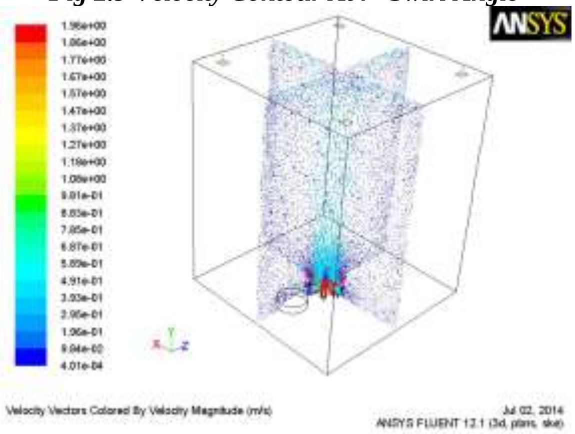

Fig 2.10 Velocity Vector At 7^0 Swirl Angle

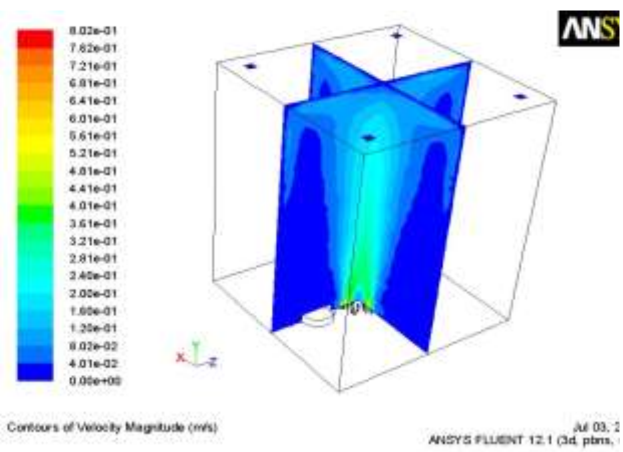

Fig 2.11 Velocity Contour At 8^0 Swirl Angle

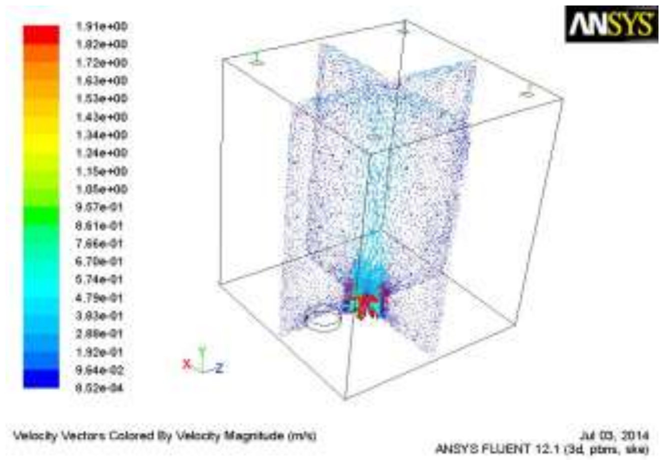

Fig 2.12 Velocity Vector At 8^0 Swirl Angle

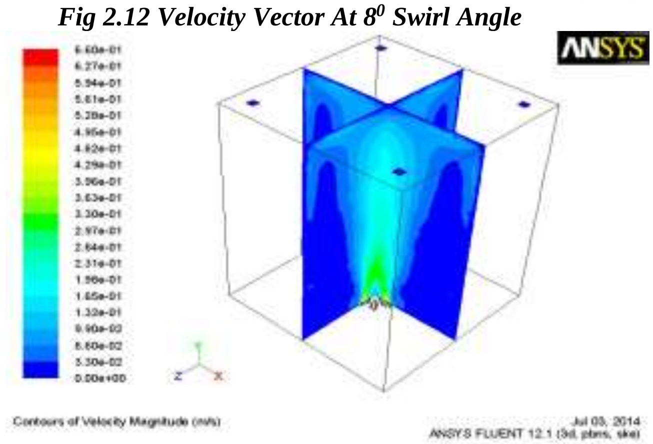

Fig 2.13 Velocity Contour At 9^0 Swirl Angle

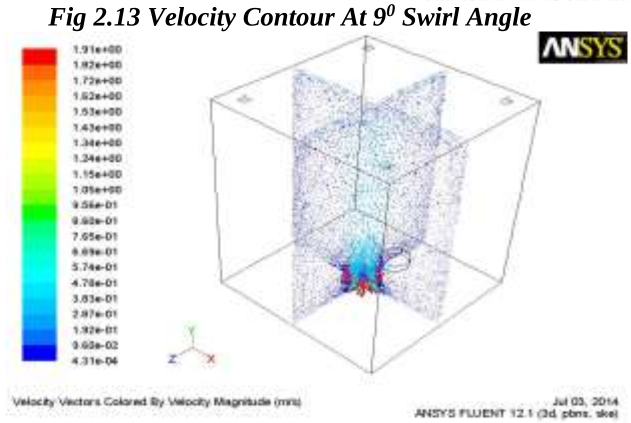

Fig 2.14 Velocity Vector At 9^0 Swirl Angle

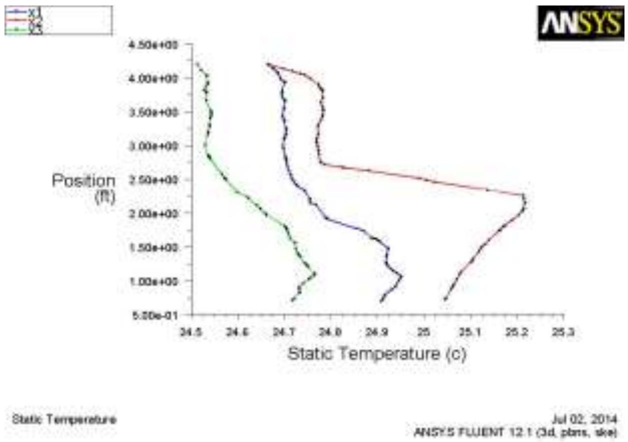

Fig 2.15 Variation Of Temperature With Respect To Height At X1, X2, X3 (7^0 Swirl Angle)

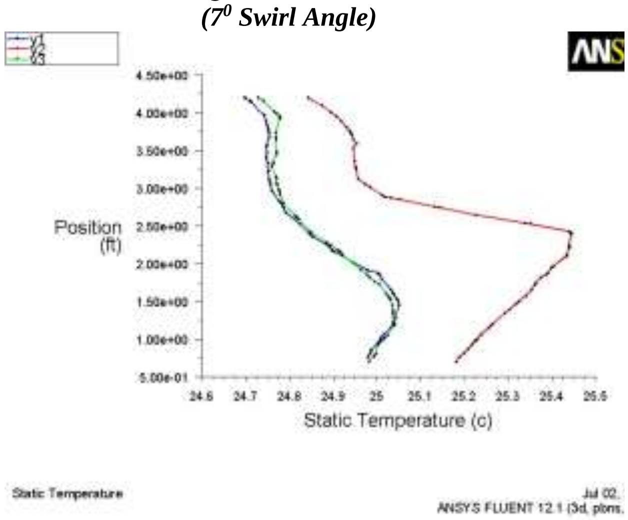

Fig 2.16 Variation Of Temperature With Respect To Height At Y1, Y2, Y3 (7^0 Swirl Angle)

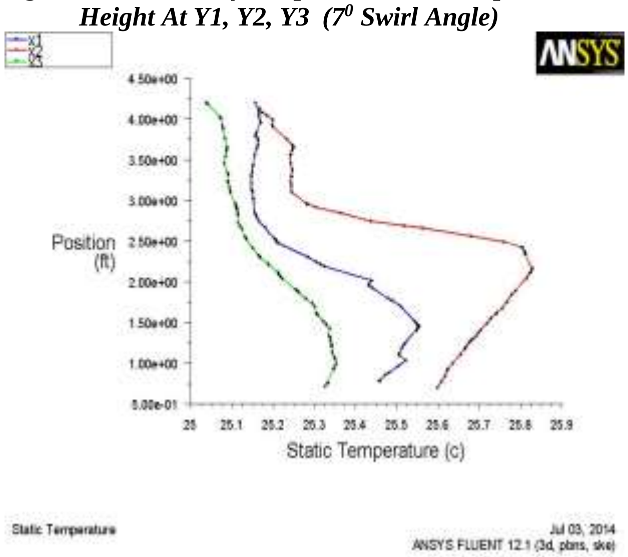

Fig 2.17 Variation Of Temperature With Respect To Height At X1, X2, X3 (8^0 Swirl Angle)

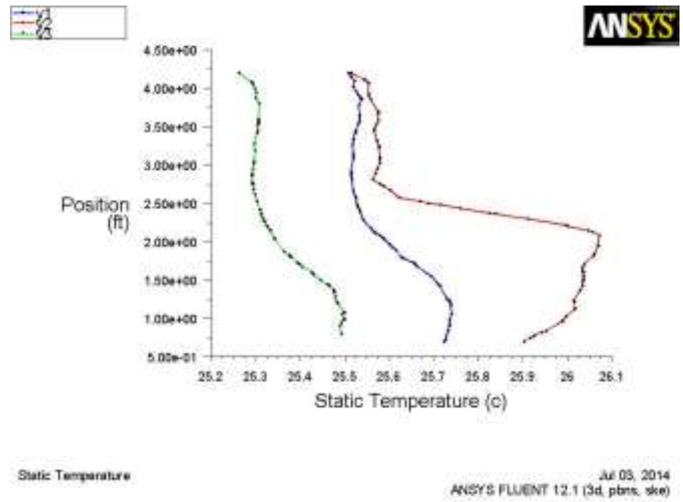

Fig 2.18 Variation Of Temperature With Respect To Height At Y1, Y2, Y3 (8^0 Swirl Angle)

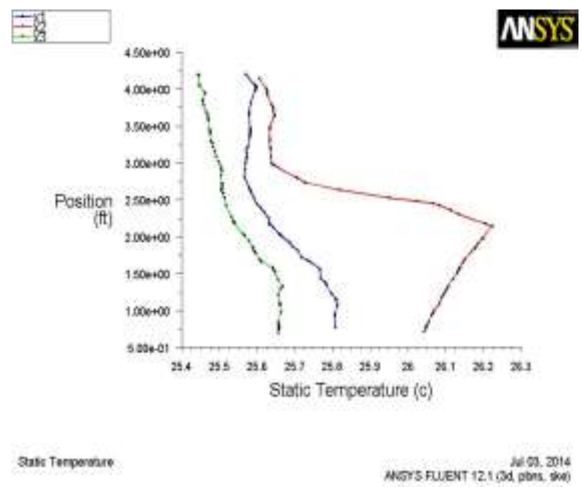

Fig 2.19 Variation Of Temperature With Respect To Height At X1, X2, X3 (9^0 Swirl Angle)

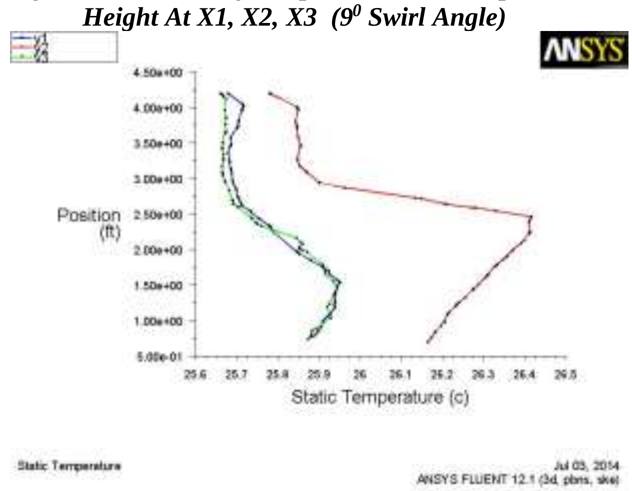

Fig 2.20 Variation Of Temperature With Respect To Height At Y1, Y2, Y3 (9^0 Swirl Angle)

RESULT AND DISSCUSSION

Table 3.1 Variation Of Temperature And Height At Location X1 At 70 Swirl Angle (Experimental And CFD)

S.NO	HEIGHT IN FT	TEMPERATURE	
		EXPERIMENTAL [24]	CFD
1	0.7	23	24.70
2	1.4	22	24.92
3	2.1	21	24.80
4	2.8	24	24.70
5	3.5	23	24.73
6	4.2	21	24.71

The result from this desertation work is that the simulation technique (CFD) gives more accurate and consistent result as compare to the experimental technique. In this desertation work it is analyzed that the experimental technique is time consuming and also the results are also not consistent so it is beneficial to use the simulation technique in case of flow analysis. This simulation and experimental technique are analysed in an air conditioned room by passing the air through a swirl diffuser having different slot angle. Due to this different slot angle flow pattern of diffused air changes. The air is circulated inside the room through different diffusers. An experiment is performated by my guide **Asst Prof Anil Kumar Rao** to find out the air flow pattern and its distribution through different swirl diffusers installed at the bottom in an air conditioned room. I have done the same analysis but with a simulation technique and compare the results with the experimental reading by taking the reference. The nature and behavior of air diffused by three different types of swirl diffuser having slots with draft angle of 7^0, 8^0, and 9^0. The graphs are plotted between temperature of diffused air inside the room and vertical height from the floor level. Pressure, Temperature, Velocity distribution inside the room and the graphs are plotted at six different locations inside the room, which shows the comparision between Experimental [24] and CFD technique. The results are described in tabulation and graphical form at all six locations and at different slot angle of the diffuser.

CONCLUSION

This section concludes the entire work which was done in this desertation. Simulation (CFD) technique has been applied to analyze the behavior of the incoming air inside an air conditioned room and to study the air flow distribution inside the room by measuring the temperature at different heights from ground at six different locations. A swirl diffuser is placed at the bottom of the room having different slot angles. This desertation concludes that by using simulation technique in comparison with the experimental technique the results are quite accurate and exact. Variation of temperature with respect to height in case of simulation technique more accurate results as compare to the experimental technique. The variation in temperature is very little with respect to height in case of simulation technique as compare to the experimental technique. So these results also conclude that simulation technique gives more accurate results as compare to the experimental technique. As we know that the experimental technique is laborious and also time consuming and does not give accurate results so it is beneficial to use CFD technique in case of flow analyses. CFD is a computer based simulation technique in order to analyze the fluid flow and heat transfer. This technique is particularly useful in case of fluid flow and heat transfer in any system. This technique employs ANSYS FLUENT software to analyze the characteristics of incoming air and also air flow distribution. It may be advantageous to use CFD over habitual experimental based analyses, since experiment has a cost which is directly proportional to the number of configurations required for testing, different with CFD, where many results can be obtained at practically no added expense. In this way, parametric study to optimise apparatus are not costly with CFD when compared to experiments. CFD technique fundamentally consist of three elements: the pre-processor, processor, and post-processor. An experiment is performed in the workshop of LNCT, Bhopal to find out the air flow pattern and its distribution through different swirl diffusers installed at the ceiling in an air conditioned room.

FUTURE SCOPE

In this analyses there are certain limitations and assumptions which are as follows:

1) ANSYS FLUENT software are very costly so the initial cost of the simulation technique is very high.
2) In simulation technique the environmental effect which comes in picture in case of experimental technique does not have any effect because it is a computer based simulation technique.
3) A heater is used in experimental technique which is of 1500W but in our analyses we can't use heater in simulation technique in place of that we consider heat flux.

4) Velocity of the incoming air is assumed to be 2 m/s.

ACKNOWLEDGEMENTS

The authors wish to acknowledge the support rendered by AISECT University Bhopal in preparation of this manuscript

REFERENCES

1) B.F. Yu, Z.B. Hu, M. Liu, H.L. Yang, Q.X. Kong, Y.H. Liu, Review of Research on air conditioning system and indoor air quality control for human health, May 2008, ELSEVIER, Science Direct.
2) Sami A, AL-Sanea, M.F Zedan, M.B. Al Harbi, Heat transfer characterstics in air conditioned room using mixing air distribution system under mixing convection system, May 2012, ELSEVIER, Science Direct.
3) Amer Abduladheem, K. S. M. Sahari, H. Hasini, Wisam Ahmed, Raed Abed Mahdi, Ventilation Air Distribution in Hospital Operating Room-Review, November 2013, International Journal Of Science and Research.
4) Tengfei (Tim) Zhang, Kisup Lee, Qingyan Chen, A simplified approach to describe complex diffusers in displacement ventilation for CFD simulation, 2009 School of Civil and Hydraulic Engineering Dalian University of Technology, Dalian, China
5) Mohammed A. Aziz, Ibrahim A. M. Gad, El Shahat F. A. Mohammed, and Ramy H. Mohammed, Experimental and Numerical Study of A/C Outlets and Its Impact on Room Airflow Characteristics, November 2012, World Academy Of Science, Engineering and Technology.
6) Josephine Lau and Qingyan Chen, Floor Supply Displacement ventilation for workshop, January 2006, ELSEVIER, Science Direct..
7) Eunsu Lim, Kazunobu Sagara, Toshio Yamanaka, Hisashi Kotani, Noriaki Mishima, Susumu Horikawa, and Tomoaki Ushio, CFD analysis of air flow characterstics in office room with task air conditioning and natural ventilation, Department of Architectural Engineering, Osaka University, Japan, The Kansai Electric Power CO. INC., Japan
8) C.J. Wu,, D.P. Liu , J. Pan, A study of the aerodynamic and acoustic performance of an indoor unit of a DC-inverter split air-conditioner, November 2011, ELSEVEIR, Science Direct.
9) Vivian Chanteloup, Pierre-Sylvain Mirade, Computational fluid dynamics (CFD) modelling of local mean age of air distribution in forced-ventilation food plants, June 2008, ELSEVEIR, Science Direct.
10) Dilek Kumlutas¸ Ziya Haktan Karadeniz, Investigation of flow and heat transfer for a split air conditioner indoor unit, September 2012, ELSEVEIR, Science Direct.
11) Jichao Hu, Junye Shi, Yuanyuan Liang, Zijiang Yang, Jiangping Chen, Numerical and Experimental Investigation on Nozzle Parameters for R410A Ejector Air Conditioning System, Institute of refrigeration and cryogenics, Shanghai Jiao Tong University, No.800 Dongchuan Road, Shanghai, 200240, China, December 2013.
12) G. Grazzini, A. Milazzo, D. Paganini, Design of an ejector cycle refrigeration system, November 2011, ELSEVEIR, Science Direct.
13) Mayurkumar S. Gandhi, Arijit A. Ganguli, Jyeshtharaj B. Joshi, Pallippattu K. Vijayan, CFD simulation for steam distribution in header and tube assemblies, 2012, ELSEVEIR, Science Direct.
14) Kwang-Chul Noh, Jae-Soo Jang, Myung-Do Oh, Thermal comfort and indoor air quality in the lecture room with 4-way cassette air-conditioner and mixing ventilation system, October 2005, ELSEVIER, Science Direct.
15) Qiong Li, Hiroshi Yoshino, Akashi Mochida, Bo Lei, Qinglin Meng, Lihua Zhao, CFD study of the thermal environment in an air-conditioned train station building, August 2008, ELSEVEIR, Science Direct.
16) Ghassem Heidarinejad, Vahid Khalajzadeh, Shahram Delfani, Performance analysis of a ground-assisted direct evaporative cooling air conditioner, May 2010, ELSEVEIR, Science Direct.
17) Ryan K. Dygert, Thong Q. Dang, Experimental validation of local exhaust strategies for improved IAQ in aircraft

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

cabins, April 2011, ELSEVEIR, Science Direct.

18) T.T Chow, Z.Lin, J.P. Liu, Effect of condensing unit layout at building re-entrants on split type air conditioner performance, July 2001, ELSEVEIR, Science Direct.
19) K.C. Ng, K. Kadirgama, E.Y.K. Ng, Response surface models for CFD predictions of air diffusion performance index in a displacement ventilated office, April 2007, ELSEVEIR, Science Direct.
20) Qi Jie Kwong, Nor Mariah Adam, B.B. Sahari, Thermal comfort assessment and potential for energy efficiency enhancement in modern tropical buildings, September 2013, ELSEVEIR, Science Direct.