# PREDICTION OF ROOM AIR DIFFUSION

# FOR REDUCED DIFFUSER FLOW RATES

A Thesis

by

KAVITA GANGISETTI

Submitted to the Office of Graduate Studies of
Texas A&M University
in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE

December 2010

Major Subject: Mechanical Engineering

# PREDICTION OF ROOM AIR DIFFUSION

# FOR REDUCED DIFFUSER FLOW RATES

A Thesis

by

KAVITA GANGISETTI

Submitted to the Office of Graduate Studies of
Texas A&M University
in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE

Approved by:

| | |
|---|---|
| Co-Chairs of Committee, | David Claridge |
| | Michael Pate |
| Committee Members, | Hamn- Ching Chen |
| | Jelena Srebric |
| Head of Department, | Dennis O'Neal |

December 2010

Major Subject: Mechanical Engineering

# ABSTRACT

Prediction of Room Air Diffusion

for Reduced Diffuser Flow Rates. (December 2010)

Kavita Gangisetti,

B.Tech. Jawaharlal Nehru Technological University, Hyderabad, India

Co-Chairs of Advisory Committee: Dr. David E. Claridge
Dr. Michael Pate

With the ever-increasing availability of high performance computing facilities, numerical simulation through Computational Fluid Dynamics (CFD) is increasingly used to predict the room air distribution. CFD is becoming an important design and analytical tool for investigating ventilation inside the system and thus to increase thermal comfort and improve indoor air quality.

The room air supply diffuser flow rates can be reduced for less loading with the help of a variable air volume unit. The reduction in supply flow rate reduces the energy consumption for the unoccupied and reduced load conditions. The present research is to study the comfort consequences for reduced diffuser flow rates and loading and to identify the hot and cold spots inside a room.

A small office room with ceiling based room air distribution method is considered for CFD analysis. The CFD results are validated with experimental measured data for the designed diffuser flow rate. A parametric study on different turbulence models,

namely, low Reynolds number modification of standard k-epsilon model, re-normalization group k-epsilon model, transition k-kl-w model and Reynolds stress model is carried out, and simulation results in terms of velocity and temperature profiles are compared against the measured data. Other important parameters such as diffuser jet inlet angle and radiation effect are also considered on the benchmark case to validate the results and to recommend the best fit parameters for room air simulations.

Analysis has been carried out for a range of flow rates and heat loads. The jet momentum, draft and temperature distribution inside the room are studied for the impact of reduced flow rates and loading.

The thermal comfort is quantified in terms of vertical temperature distribution and percentage dissatisfied index.

From the research it is found that, for the studied room setup and air distribution method, the diffuser flow rate can be reduced up to 30 % of the design flow rate, without experiencing a considerable effect on the room air temperature distribution. Also, based on thermal comfort and room air temperature distribution, several recommendations for occupant spacing in a room are suggested for reduced diffuser flow rates.

# DEDICATION

This work is dedicated to my parents, Mr. G.V.Satyanarayana and Mrs. G. Surya, and my sister, G.Harita, who have been an immense support and encouragement to me throughout my research.

# ACKNOWLEDGEMENTS

I would like to express my sincere gratitude to my committee chair, Dr. David Claridge, for his invaluable advice and encouragement throughout this research. He has been a great source of inspiration for me.

I also want to extend my deepest gratitude to Dr. Jelena Srebric, for her kind and invaluable guidance throughout my research.

I wish to thank my committee members, Dr. Michael Pate, and Dr. Hamn-Ching Chen, for their guidance and feedback on the research work.

Thanks also go to my friends and colleagues at Energy Systems Laboratory and the department faculty and staff for making my time at Texas A&M University a great experience.

Finally, thanks to my mother and father for their encouragement and support.

# NOMENCLATURE

| | |
|---|---|
| A | area, $m^2$ |
| $A_0$ | effective diffuser area or area factor, $m^2$ |
| ACH | air change per hour, $h^{-1}$ |
| $c_p$ | specific heat at constant pressure, J/(kg.K) |
| h | convective heat transfer coefficient, $W/(m^2.K)$ |
| $H_0$ | effective width of a linear diffuser, m |
| k | thermal conductivity, W/(m.K) |
| $k$ | turbulence kinetic energy, $m^2/s^2$ |
| $K_1$ | jet centerline velocity decay constant, - |
| $K_2$ | jet centerline temperature decay constant, - |
| M | jet momentum at distance x from the jet source, $(kg.m)/s^2$ |
| $M_x$ | initial jet momentum, $(kg.m)/s^2$ |
| P | diffuser static pressure drop, Pa |
| PD | percentage dissatisfied, - |
| q | convective heat flux, W/m2 |
| qc | heat load, W |
| Q | volume flow rate, $m^3/s$ |
| R | Reynolds number, - |
| $t_a$ | temperature of air , $^0C$ |
| T | temperature, $^0C$, K, or $^0F$ |

| | |
|---|---|
| *Tu* | turbulence intensity, - |
| $T_m$ | centerline or maximum temperature velocity, $^0C$, or $^0F$ |
| $T_r$ | average room temperature, $^0C$, or $^0F$ |
| u | velocity component in x direction, m/s |
| $u_i$, $u_j$ | index notation of velocity components, m/s |
| $u_0$ | initial jet velocity, m/s |
| $u_m$ | centerline or maximum jet velocity, m/s |
| v | velocity component in y direction, m/s |
| *V* | magnitude of velocity, m/s |
| V | volume, $m^3$ |
| x, y, z | Cartesian coordinates in three directions, m |
| $x_i$, $x_j$ | index notation of Cartesian coordinates, m |
| $y_{1/2Um}$ | half-width of the jet spread, m |
| w | velocity component in z direction, m |

**Greek symbols**

| | |
|---|---|
| β | volumetric thermal expansion coefficient, $K^{-1}$ |
| *Γ* | diffusion coefficient, kg(m.s) |
| μ | viscosity of fluid, Pa.s |
| ρ | density of fluid, $kg/m^3$ |
| Δ | difference, - |
| ε | rate of dissipation of turbulent kinetic energy per unit mass, J/(kg.s |

# TABLE OF CONTENTS

# LIST OF FIGURES

FIGURE Page

FIGURE Page

FIGURE Page

FIGURE Page

# LIST OF TABLES

# 1. INTRODUCTION

## *1.1 Background*

The movement of air through interior spaces of a building is called space or room air diffusion. Air is introduced in the room/space through a supply outlet in order to increase thermal comfort and improve indoor air quality. The numerical prediction of air flow patterns has been a research object for several decades. By 1980, it was possible to predict the flow field in large domains with relatively small openings using Computational Fluid Dynamics (CFD).

In recent years, CFD has gained in popularity as a design and analytical tool, because CFD offers an engineer with the ability to visualize the air movement inside the room. It enables one to predict velocity and temperature values and distribution in a space. It allows an engineer to change parameters, such as flow rates, inlet temperature, heat loading, occupancy, etc and thus helps to predict the impact of these design changes on indoor climate and energy management in real buildings.

---

This thesis follows the style of ASHRAE Transactions.

### *1.2 Purpose and objectives*

The purpose of this research is to study the air temperature distribution in a space with reduced diffuser flow rates and heat loads.

The objectives of this research include:

1. To carry out computational fluid dynamics (CFD) analysis for room air distribution at the design flow rate of the diffuser and validate it with experimental data.
2. To carry out CFD analysis of room air distribution for reduced flow rates and heat loads.
3. To study the temperature distribution for a range of flow rates and heat loads to provide guidance on comfort consequences of low flow settings.

# 2. LITERATURE REVIEW

## *2.1 Room air diffusion*

### *2.1.1 Airflow patterns*

The airflow field within a space is called its airflow pattern. Typically, velocity vectors or path lines of air particles are drawn to depict the airflow pattern. These patterns provide us an understanding of the airflow behavior. There can be many types of airflow patterns (as shown in Figure 1) depending upon diffuser type, location, jet direction, jet momentum, etc. Three distinct airflow patterns are described below (Chapter 20, ASHRAE.2009):

- Perfect mixing – Perfect mixing is the theoretical condition where the air is instantly and evenly diffused into the room. The temperature, concentration and velocity of the air will be more or less constant throughout the inlet, exhaust and within the space. It is also known as complete, uniform, or well stirred mixing.
- Entrainment flow – Entrainment flows are the real flow patterns that develop in spaces when jet flows are present. The entrainment flow develops via a confined jet. In many large spaces, multiple jets are present; hence the flow is very complex. For a ceiling based diffuser, entrainment flow is often called conventional mixing.
- Displacement flow – This is also called piston or plug type flow. In this flow pattern, the dominant air flow direction is from side to side, ceiling to floor, or from floor to ceiling. The goal in creating the displacement flow is to have as little mixing as possible. For floor to ceiling flow, air is introduced in the room via floor mounted

outlets, and then flows up through the space and into the ceiling. This arrangement is called raised floor or dropped ceiling.

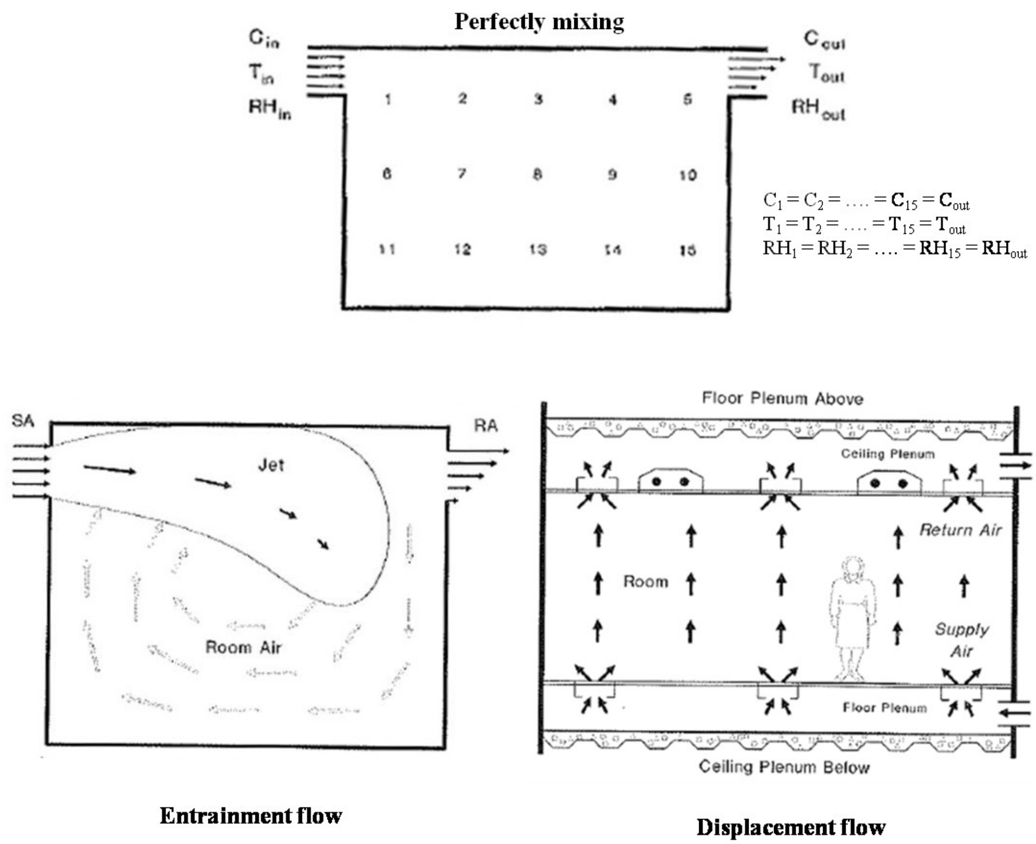



**Figure 1 Airflow patterns - Perfect mixing, entrainment flow and displacement flow (Rock and Zhu, 2002)**

### *2.1.2 Descriptor of air outlet performance*

***Occupied zone***

According to ANSI/ASHRAE Standard 55-2004, the occupied zone of a room is defined as being 2ft (0.6 m) from the walls. For our case, the occupied zone of the room is taken as 6ft above the floor. It is assumed that it is not necessary to provide thermal comfort in the portions of the spaces that are normally not occupied.

***Primary, secondary and total air***

The supply air that is emitted by an air outlet is known as primary air. The air that is entrained into the jet is called secondary air. At any cross section, perpendicular to the main direction of the jet, the combined primary and secondary air is known as total air The amount of total air increases as the jet proceeds from the outlet. The *entrainment ratio* compares the total to primary air ratio.

***Maximum and terminal velocities***

The maximum velocity is the peak velocity in an imaginary cross section through a real jet perpendicular to its primary direction. At a point downstream, the peak velocity will fall to some target value of interest called the terminal velocity. Commonly used values for this terminal velocity are 150, 100 or 50 fpm. In general, it is desired that the conventional ceiling based air jets slow to or below the terminal velocity before entering the occupied zone.

***Throw, spread and drop***

Throw is defined as the horizontal or vertical axial distance an airstream travels after leaving the air outlet before maximum stream velocity is reduced to a specified terminal velocity (e.g. 50, 100, 150 0r 200 fpm), defined by ASHRAE standard 70.The throw values are usually reported in the manufacturer's catalogue. Spread refers to the divergence of airstream in horizontal or vertical plane after it leaves an outlet.

Drop refers to vertical distance that the lower edge of a horizontally projected airstream descends between the outlet and the end of its throw. (Shown in the Figure 2)

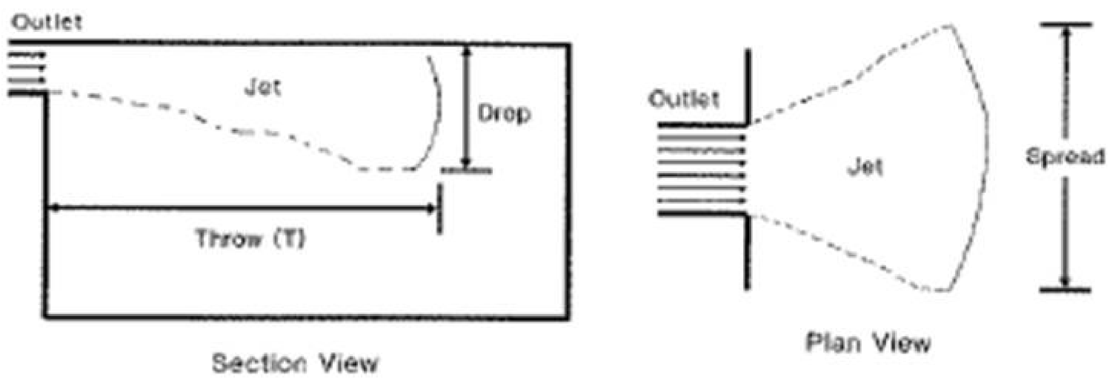



**Figure 2 Throw, spread and drop (Rock and Zhu, 2002)**

***Dumping and draft***

Draft refers to the undesired or excessive local cooling of a person caused by low temperature and air movement. Dumping occurs when the air jet directly enters into the occupied zone. As shown in Figure 3; it can occur if the supply outlet has:

a) Too long throw - The jet can be deflected into the occupied zone without slowing down due to which the high velocities are observed in the occupied zone.

This is a typical case of dumping which can be seen when the diffuser jet flow rate is increased from the designed flow rate.

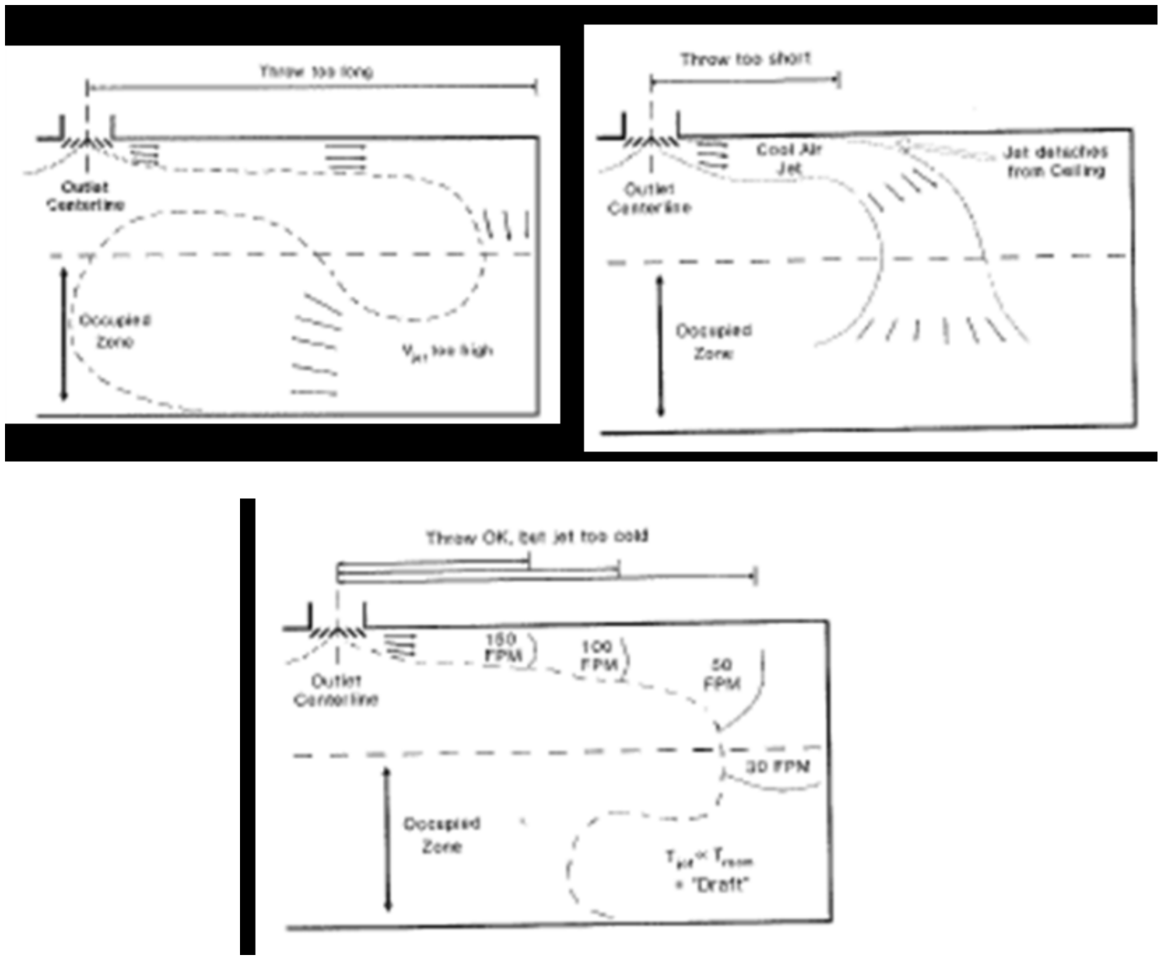



**Figure 3 Different situations of dumping (Designer's guide to ceiling based air diffusion by Brian A. Rock and Dandan Zhu, 2002)**

b) Too short throw -The negative buoyancy force of a cool air jet can exceed Coanda effect. In that case, the jet detaches itself from the ceiling and may enter the occupied zone before slowing enough.

c) If the supply temperature difference is too large - The jet can enter the occupied zone before being tempered enough even if it doesn't detach.

### *2.1.3 Thermal comfort*

Thermal comfort is defined as that condition of mind that expresses satisfaction with the thermal environment (ASHRAE Standard 55-2004). Factors that influence comfort and are controllable by the occupant include temperature of air and the walls of the room, air humidity, air speed etc. Factors which are not controllable by the operator include activity level, clothing and personal preference.

It has been identified as the most annoying factors in the offices. Fanger and Christensen (1986) established the percentage of population feeling draft when exposed to a given mean velocity at the neck. Fanger et al. investigated the effect of turbulence intensity on sensation of draft (Air turbulence and sensation of draft, 1988). Turbulence intensity significantly affects draft sensation, as predicted by the following model. This model can be used to quantify draft risk in spaces and to develop air distribution systems with a low draft risk.

$$PD = (34 - t_a)(V - 0.05)^{0.62}(0.37VTu + 3.14)$$

where, PD is the percent dissatisfied, $t_a$ is temperature of air (deg C), V is magnitude of velocity (m/sec).

Tu is the turbulence intensity in % defined as: $Tu = \frac{\sqrt{k}}{1.1\,V}$ , where, k is the turbulence kinetic energy.

***Vertical air temperature difference***

If the gradient of temperature from floor to the head level is large, it causes local discomfort. Studies of vertical air temperature differences and their influence on thermal comfort are reported by McNair (1973), McNair and Fishman (1974), and Olesen (1979). During these tests, subjects were allowed to change the temperature level in the test room whenever they desired, but the vertical temperature difference, however, was kept unchanged. Vertical air temperature difference is an important parameter to be kept in mind while analyzing thermal comfort in a region.

### *2.2 Methods for specifying diffuser boundary conditions*

Proper specification of diffuser boundary conditions is essential for room air predictions. The major problems in simulating the diffuser are: 1) the size of the diffuser geometry, which is small, compared to the size of the room, and 2) physics of the transitional flow (flow regime between laminar and turbulent flow) with heat transfer mainly through convection. As a result, it requires a large number of nodes or grid cells to capture the flow in the region near the diffuser, which makes the problem computationally more expensive (as shown by Emvin and Davidson (1996), who simulated the complete geometry of diffuser). Thus, the specification of diffuser boundary conditions with a simple method is needed.

The extensive literature review by Srebric and Chen (2001b, 2002) showed different methods for simulating diffuser boundary conditions. Diffuser analysis methods can be grouped into two basic categories - momentum modeling at the air supply devices and momentum modeling in front of the air supply devices.

The momentum modeling approach at the air supply device imposes an initial jet momentum, $M_0$, at the supply device. It is easy to use and includes the basic model, wide slot model and the momentum method (Shown in Figure 4). The second method uses momentum, $M_x$, downstream of the diffuser. It requires measurements, jet equations and it includes the Box method, the prescribed velocity method and the diffuser specification method. (Shown in Figure 5). A complete methodology for carrying out test in order to obtain diffuser data for CFD modeling has been given by Chen and Srebric (2001a).

### 2.2.1 Basic model

The basic model proposed by Heikkinen (1991b) replaces the diffuser with a simple opening, which has the same effective area as the original diffuser geometry. The computed flow field looks similar to the one observed, but the maximum velocity is excessively low and the jet spreading is not predicted well. Ewert et al (1991) showed that the jet profiles and decay are likewise not predicted well. Another study by Chen and Moser (1991) indicated this model is not suitable for non-isothermal flow.

### 2.2.2 Wide-slot model

The wide-slot model proposed by (1991b) is a modification of the basic model. In this method, the same slot is chosen but with a different aspect ratio. The results show that the mixing in the core is over-predicted and jet penetration is higher. Thus, the wide-slot model is ineffective.

### 2.2.3 Momentum method

In the momentum method proposed by Chen and Moser (1991), momentum and mass fluxes are decoupled at the diffuser boundary. The diffuser opening has the same gross area, mass flux and momentum flux. Hence, a source for the momentum is introduced at the diffuser boundary to account for the actual discharge velocity. This method was validated with measured data for nozzle and displacement diffusers by Chen and Jiang (1996).

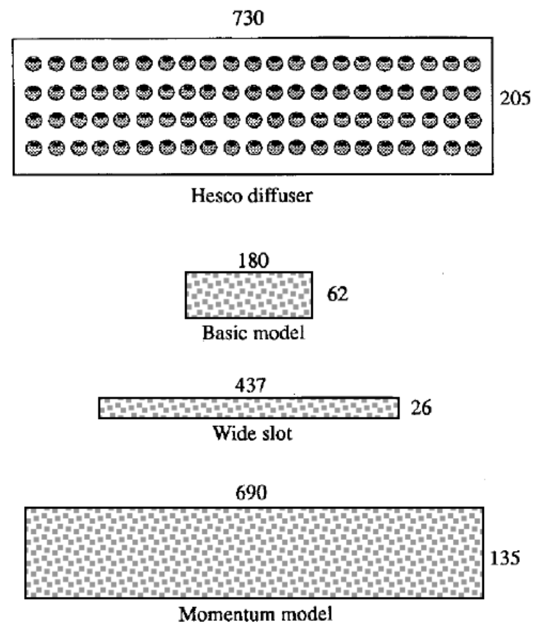



**Figure 4 The approach for momentum modeling at the air supply device (ASHRAE Research Project RP -1009, "Simplified Diffuser Boundary Conditions for Numerical Room Airflow Models, 2001)**

### *2.2.4 Box model*

Nielsen (1989, 1992) proposed the box method with an imaginary box near the diffuser. The flow field inside the box is neglected. The measured velocity profiles are given as input on one side of the box (which is front of diffuser), whereas other sides uses a free boundary condition with zero gradients for flow parameters. Nielsen (1989, 1992). Results obtained from the box method are in good agreement with the measured data.

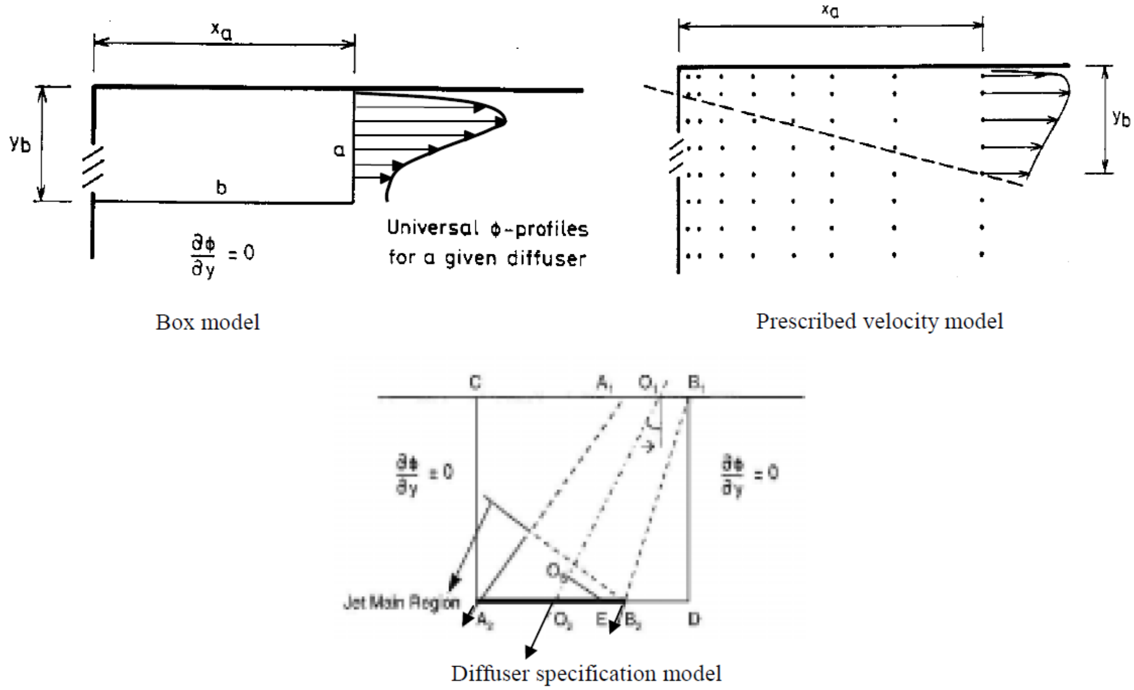



**Figure 5 Methods for momentum modeling in front of an air supply device (ASHRAE RP -1009, "Simplified Diffuser Boundary Conditions for Numerical Room Airflow Models", 2001)**

### *2.2.5 Prescribed velocity model*

The velocity model was proposed by Gosman et al. and later developed by Nielsen (1989). This method also uses jet theory or measured data, but the space between the diffuser and profile location is included in the calculation. In this method, boundary conditions are given at a simple opening (as in the basic model) and also in the flow field as in the box model. The velocities are taken either from theory or manufacturer's data. Heikkinen (1991a) showed that the jet decay was closely predicted, but the jet spread and maximum velocity were not predicted well. Skovgaard and Nielsen (1991) and Svidt (1994) showed that the results are in better agreement with the measured data than the basic model.

### *2.2.6 Diffuser specification model*

The diffuser specification model was proposed by Huo et al. (1996). This method is another modification of the box model. The model uses jet formulae for jet profiles, decay and trajectories, and thus it do not require any measurements or manufacturer's data. The use of jet formulae is advantageous when we have existing jet formulae in the literature, but it cannot be used to predict the flow distribution for several important special cases like the partially attached jet or non isothermal jets since jet formulae are not available for these cases.

***Summary of boundary condition specification for diffuser***

Most of the models reviewed above have specific advantages and disadvantages. Out of all the models, the momentum method is selected for defining diffuser boundary condition because it:

i. Does not need experimental data (velocity, temperature and concentration profiles) which is seldom available for diffusers; and-
ii. Is easy to implement and modify for different flow rates and discharge velocities.

### *2.3 Jet theory, classification and formulae*

#### *2.3.1 Jet theory*

Air jets can be classified based on several different criteria as mentioned below.

***Proximity to a surface***

If the air jet is not obstructed by walls or ceilings, it is considered a "free" jet. If the air jet is attached to a surface, it is called an "Attached" jet. This effect is also called the *Coanda Effect*, which is commonly seen when a solid surface is near to the air jet. If the jet in a room is influenced by reverse flows created by it, entraining the ambient air, it is called a "Confined" jet.

***Temperature difference between jet and ambient air***

If the temperature of the air jet is the same as the temperature of the ambient air, it is called an "isothermal" jet. Otherwise it is called a "non-isothermal" jet.

***The diffuser type***

- Compact air jets – are formed by cylindrical tubes, nozzles, square or rectangular openings with a small aspect ratio. These jets are three dimensional and axisymmetric - at least at some distance from the diffuser opening.
- Linear air jets –are formed by linear slots or rectangular openings with a large aspect ratio. These jet flows are considered two dimensional.
- Radial air jets –are formed by ceiling air diffusers with flat disks or multi diffusers that direct the air horizontally in all directions.

- Conical air jets –are formed by cone-type or regulated multi diffuser ceiling-mounted air distribution devices. They have an axis of symmetry. The air flows parallel to the conical surface with the maximum velocities in the flow cross section perpendicular to the axis.
- Incomplete radial air jets –are formed by outlets with grilles having diverging vanes and a forced angle of expansion.
- Swirling air jets –are formed by diffusers with vortex forming devices. These devices create rotation, which imparts axial, tangential and radial components to the velocity vectors.

### *2.3.2 Description of jet region*

For indoor air simulation, an air jet is divided into four zones – core, transitional, main and terminal zones (Figure 6).

- Core zone - The centerline velocity $U_m$ is equal to the initial jet velocity. It usually extends approximately four diameters from the supply opening.
- Transitional region - Its length is determined by the type of outlet, aspect ratio of the outlet, initial air flow of turbulence, etc. It can be neglected for circular or plane jets when the aspect ratio is smaller than 13.5 (Awbi 1991).
- Main region –This is a zone of fully established turbulent flow that may extend from 25 to 100 equivalent air outlet diameters. This zone is very important for HVAC engineering applications because the diffuser jet usually enters the occupied space within the main zone.

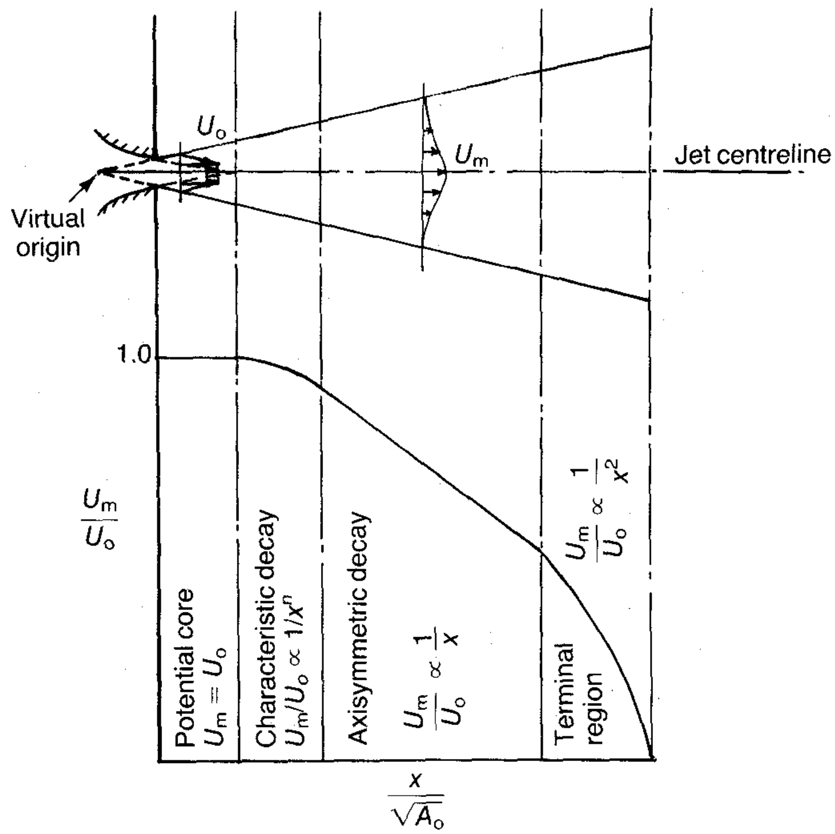



**Figure 6 Jet zones and characteristic velocity profiles (Awbi 1991)**

- Terminal zone – The jet velocity decay is rapid and the jet velocities become the same order as room airflow velocities in this zone. The theory of the terminal jet region is still under development and no formulae are available to describe the jet degradation.

### *2.3.3 Jet formulae*

The jet formulae are the empirical expressions for specifying jet velocity and temperature profiles, decay and trajectory. These equations are defined for the fully developed jet region.

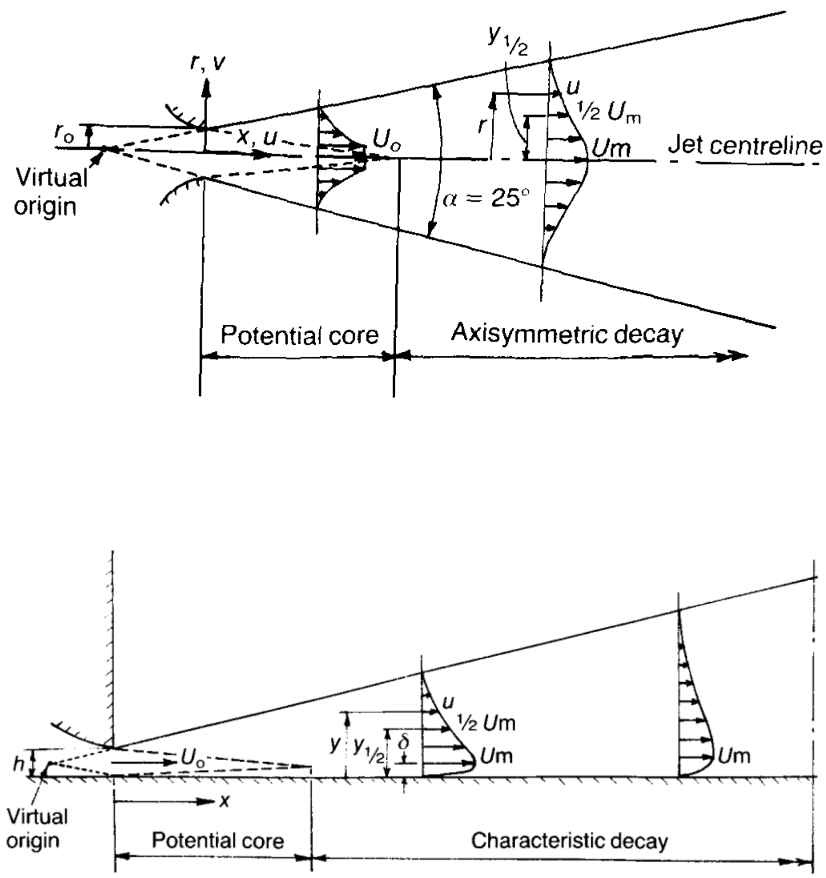



**Figure 7 Jet velocity profiles for (a) free axisymmetric jets (b) attached jets (Awbi 1991)**

***Jet profile equations***

The profile equations for the jet can be written as follows:

- Isothermal plane (linear) jet equation (Huo et al,1996)

$$\frac{u}{u_m} = exp\left[-\left(\frac{y^2}{2(0.082x)^2}\right)^2\right] = exp\left[-74.4\left(\frac{y}{x}\right)^2\right]$$

where $u_m$ is the centerline velocity, and $u$ is the velocity at any x and y (Refer to Figure 7).

- Isothermal axi-symmetric jet equation (Huo et al ,1996)

$$\frac{u}{u_m} = exp\left[-\left(\frac{y^2 + z^2}{2(0.082\,x)^2}\right)\right]$$

- Isothermal attached jet equation (Verhoff,1963)

$$\frac{u}{u_m} = 1.48\,\eta^{1/7}[1 - erf\,(0.68\eta)]$$

where $\eta = \frac{y}{y_{1/2U_m}}$ , $y_{1/2U_m} = 0.073(x + 12b)$ , (Rodi 1982)

- Non-isothermal jet equation

$$\frac{T - T_r}{T_m - T_r} = \sqrt{\frac{u}{u_m}}$$

where $T_m$ is the centerline temperature for free jets or the lowest temperature at the given jet cross section for wall jets. $T_r$ is the average room temperature. For mixing ventilation, $T_r$ equals the return air temperature.

*2.3.4 Jet decay equations*

- Isothermal axi-symmetric free jet velocity decay

$$\frac{u_m}{u_o} = K_1 \frac{\sqrt{A_0}}{x}$$

where $u_o$ is the initial jet velocity, $A_0$ is the diffuser effective area, $x$ is the distance from diffuser, and $K_1$ is the centerline velocity decay constant.

- Isothermal plane (linear) free jet velocity decay

$$\frac{u_m}{u_o} = K_1 \frac{\sqrt{H_0}}{x}$$

where $H_0$ is an effective width of the diffuser.

- Non- isothermal axisymmetric free jet temperature decay

$$\frac{T_m - T_r}{T_0 - T_r} = K_2 \frac{\sqrt{A_0}}{x} \frac{1}{K_n}$$

where $T_m$ is the centerline jet temperature, $K_n = \left[1 \pm \frac{2.5K_2}{K_1^2} A_r \left(\frac{x}{\sqrt{A_0}}\right)^2\right]^{1/3}$, and $K_n = 1$ for horizontally projected jets (Li.et al.1993).

- Non-isothermal plane (linear) free jet temperature decay

$$\frac{T_m - T_r}{T_0 - T_r} = K_2 \frac{\sqrt{H_0}}{x} \frac{1}{K_n}$$

where $K_n = \left[1 \pm \frac{2.5K_2}{K_1^2} A_r \left(\frac{x}{\sqrt{A_0}}\right)^{3/2}\right]^{1/3}$ for vertically discharged jets, and $K_n = 1$ for horizontally projected jets (Li et al.1993). Formulae for inclined jets were developed by Zhivov (1993).

### *2.4 Numerical model*

The numerical predictions were performed by a CFD solver – FLUENT (*Ansys FLUENT User's Guide 12.0, April 2009*) that solves the following governing equations.

- Mass continuity equation

$$\frac{\partial \rho}{\partial t} + \rho \frac{\partial U_i}{\partial x_i} = 0$$

where $\rho$ is the air density, $U_i$ is the mean velocity component in the $x_i$ direction (for i=1, 2, 3, $x_i$ corresponds to three perpendicular axes).

- Momentum equation

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial \rho U_j U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \mu \frac{\partial^2 U_i}{\partial x_j \partial x_j} + f$$

The above equation is the famous Navier- Stokes (N-S) equation where, $U_i$ and $U_j$ are the velocities in $x_i$ and $x_j$ direction respectively, P is the pressure, $\mu$ is the viscosity of fluid, and $f$ is the body force which includes gravity or centrifugal forces.

- Energy equation

$$\frac{\partial \rho T}{\partial t} + \frac{\partial \rho U_j T}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\Gamma_{T,eff} \frac{\partial T}{\partial x_j}\right) + \frac{q}{c_p}$$

where $\Gamma_{T,eff}$ is the effective diffusion coefficient for T, $q$ is the thermal source and $c_p$ is the specific heat.

- Turbulence modeling

By time averaging the momentum equation, we get the Reynolds averaged Navier-Stokes equation, which is given below:

$$\frac{\partial}{\partial x_j}\left(\rho U_i U_j\right) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(-\rho \overline{u_i u_j}\right) + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right]$$

A test of four turbulence models, namely the k-epsilon model with low Reynolds number modification as proposed by Chen (1995), the renormalization-group (RNG) k-epsilon model, the Reynolds stress model (RSM) and the Transition k-kl-w model, has been performed to select the best turbulence model suited for the problem. A detailed explanation on the literature of these turbulence models can be found in the Fluent manual (*Ansys FLUENT 12.0 User's Guide, April 2009*) and textbook – "A first course in turbulence" Tennekes and Lumley (1972).

# 3. RESEARCH CONDUCTED

## *3.1 Benchmark CFD simulation for room air distribution validated with experimental data.*

A Benchmark CFD simulation for a small office space is conducted. The problem specification, i.e. room dimensions, position and size of air diffuser and the outlet are taken from the paper – "Simulation of a multiple - nozzle diffuser" by Chen and Moser (1991). The values of cooling load are taken from simulation carried by Chen (1990) – "Simulation of Test case E", IEA Annex 20 project. The simulated results, in terms of maximum velocity, distribution of velocity and temperature in the room are validated against the experimental data.

### *3.1.1 Study the effect of various parameters on the CFD simulation.*

A study of the influence of different parameters on the Benchmark Case is conducted. This study is used to determine the diffuser inlet angle, the grid size, the turbulence model and the importance of radiation in the room air simulations. The values of these parameters which provide the best fit between the simulated and measured values are used for simulations at other flow rates.

## *3.2 Conduct analysis for reduced diffuser flow rates and loading*

Once the baseline simulation is validated, analysis is carried out with reduced diffuser flow rates. The flow rates are reduced in steps from the design flow rate (100%)

to 15% of the design flow rate. The intent is to predict the jet draught, jet momentum, air velocity and temperature distribution inside the room.

### *3.3 Analyze temperature distribution in the occupied zone at reduced flow rates*

The jet draft and temperature distribution in the occupied zone are studied in detail for the impact of reduced flow rates and loading.

# 4. RESULTS OF BENCHMARK SIMULATIONS AND PARAMETER DETERMINATION

### *4.1 Domain and grid setup*

The computational domain consists of a small empty office, which is 4.2m long, 3.6 m wide and 2.5 m high, as shown in Figure 8. It consists of an air supply device (nozzle diffuser, as explained below), an outlet and a window. Due to the symmetry of the room, only half of the room is simulated. The air enters with a flow rate of 3 ACH (0.0315 $m^3/s$) at a supply temperature of 15 deg C. Heat transfers through the window and the walls of the room. The window is assumed to have a surface temperature of 30 $^0$C.The diffuser used is a 'HESCO" type diffuser, which was used in the International Energy Agency (IEA) Annex 20 project (1993): "Room air and contaminant flow, evaluation of computational methods".

The diffuser consists of 84 small round nozzles arranged in four rows in an area of 0.71m X 0.17 m. Each nozzle has a diameter of 11.8 mm and the total effective area for all the nozzles is 0.0008 $m^2$. The flow direction of all the nozzles is adjusted to $40^0$ upwards as shown in Figure 9. A non - uniform structured mesh is used with fine mesh near the walls and window as shown in Figure 10.

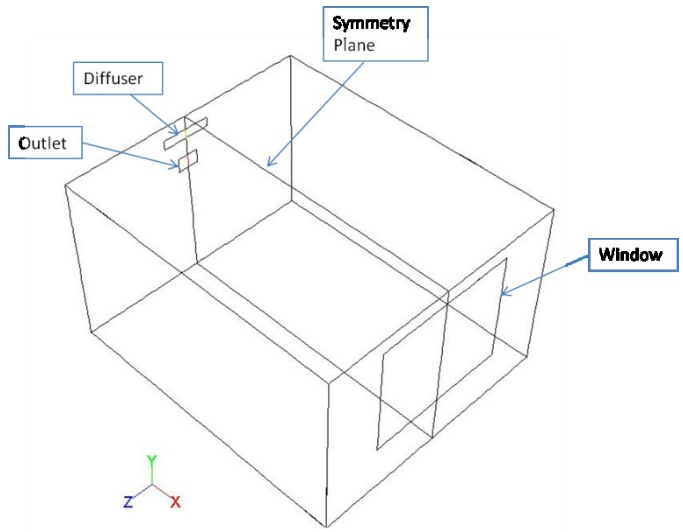



**Figure 8 Computational domain**

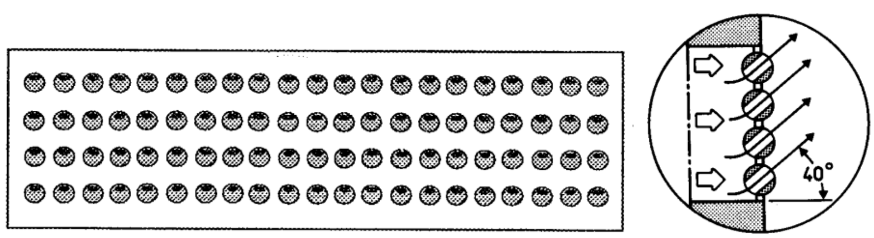



**Figure 9 Nozzle diffuser and its orientation**

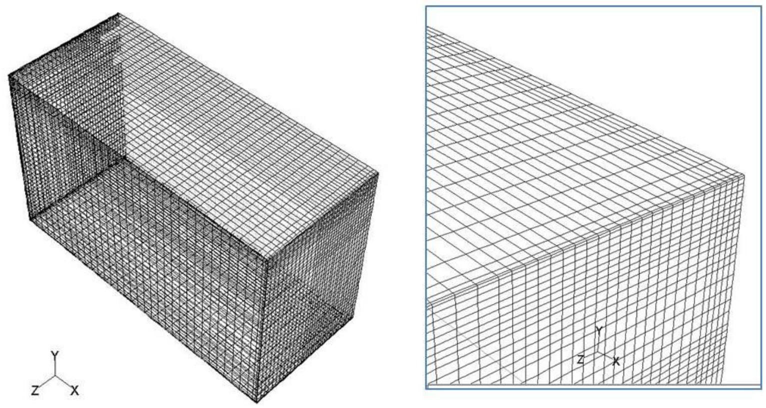



**Figure 10 Meshed domain and magnified view of mesh near walls**

### *4.2 Boundary condition and flow properties*

#### *4.2.1 Flow properties*

The working fluid is air. The fluid properties for air are held constant with air density of 1.225 kg/m$^3$ and viscosity - 1.7894e-05 kg/m-s.

#### *4.2.2 Boundary condition at inlet*

From the literature review, the momentum method is selected for specifying the diffuser boundary condition. In this method, we consider the actual area of the diffuser and decouple the mass and momentum boundary conditions. The inlet mass flow rate is given at the grille diffuser surface, and a momentum source is defined near the inlet to account for the original supply velocity. The supply velocity can be calculated from the equation:

$$V_0 = \frac{m}{\rho * Area_{eff}}$$

where, m is the mass flow rate of the diffuser,$V_0$ is the actual supply velocity, $\rho$ is the density of air, and $Area_{eff}$ is the effective area of the diffuser which is calculated from the diffuser geometry.

***Turbulence specification***

$$k_0 = \frac{3}{2}\,(Ti *\ U_0)^2$$

$$\varepsilon_0 = \frac{C_\mu^{3/4} k_0^{3/2}}{l_0}$$

where, $U_0$ is the supply velocity, $T_i$ is the turbulence intensity, which is assumed to be 10 %, $C_\mu$ is the empirical constant and $l_0 = 0.1L$ , where, L is the characteristic length of the diffuser.(Ewert, Renz, Vogl and Zeller, 1991)

### *4.2.3 Boundary condition at surfaces*

The surfaces enclosing the room are given no-slip wall boundary conditions, which indicates that the first fluid layer adjacent to the wall sticks to the wall and moves with the same velocity as the wall.

***Shear stress calculation***

For the no-slip boundary condition, the properties of flow adjacent to the wall/boundary are used to predict the shear stress on the fluid at the wall.

$$\tau = \ \mu_{eff}\,\frac{\partial U_i}{\partial x_j}$$

where, $\tau$ is the shear stress and $\mu_{eff}$ is the effective viscosity.

***Heat transfer calculation***

The surface to surface radiation model is used. In this model, it is assumed that the medium separating the walls of the room does not participate in the radiation process, which is a reasonable approximation for air.

*4.2.4 Boundary condition at outlet*

Atmospheric pressure is given at the outlet boundary condition. The backflow turbulence is specified with intensity and length scales.

***4.3 Numerical solution procedure***

The air flow equations are solved for the three dimensional Cartesian system using the SIMPLE algorithm. The convection and diffusion terms are integrated using the QUICK difference scheme (Fluent 12.0 User's Guide, 2009).

The solution is monitored at a point in the middle of the domain. Convergence is considered to be reached when the normalized residuals of each equation are below $10^{-3}$ and when the average value of the velocity magnitude at the point (which is being monitored) becomes almost constant.

### *4.4 Benchmark Case: Study of different parameters*

The following parameters were studied to determine their effect on the room air simulations.

- *Effect of different turbulence models*
- *Effect of radiation*
- *Effect of diffuser inlet angle*

A grid sensitivity analysis was performed on the mesh and the finalized mesh is used for carrying out the study on different parameters.

#### *4.4.1 Effect of different turbulence models*

The turbulence models have huge impact on room air simulations in term of room air flow and magnitudes of velocity and temperature. Different turbulence models were studied and compared by Chen (1995) and Nielsen (1998). In order to study the effect of turbulence models for predicting the air velocities and temperatures inside the room, simulations were carried out using four different turbulence models, namely, the k-epsilon model with low Reynolds number modification (proposed by Chen, 1995), the Renormalization-group (RNG) k-epsilon model, the Reynolds stress model (RSM) and the Transition k-kl-w model. The literature on these four turbulence models can be found in any turbulence methods book.

The same non-uniform structured mesh is used for all the simulations in order to avoid grid diffusion errors. The results are plotted at two poles as shown in Figure 11.

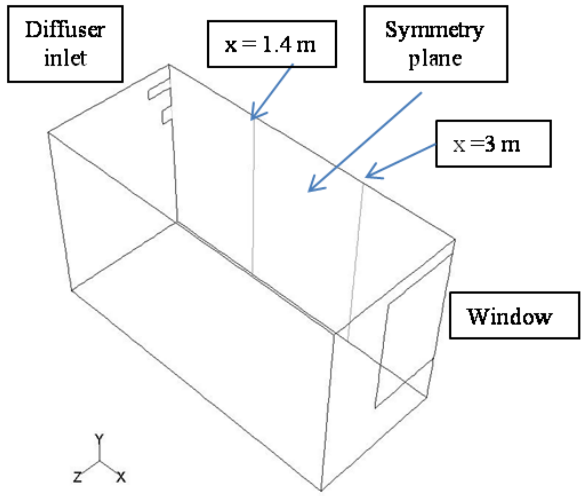



**Figure 11 Symmetry model of the room with post processing lines used for plotting the results**

The simulated results are compared with the experimental data from the paper "Simulation of a multiple – nozzle diffuser" by Chen and Moser (1991).These measurements were carried by Blomqvist (1991) and Fossdal (1990).

As shown in Figure 12, LKE predicts lower velocities than the measured values in the near- ceiling zone and higher momentum in the occupied zone (which is considered to be 6 ft i.e. from floor level till y = 1.8 m). The RNG K-epsilon model predicts good values for velocities in the occupied zone but has higher momentum values in the near ceiling region. The RSM and Transition k-kl-w models seem to predict the velocities better than the other models.

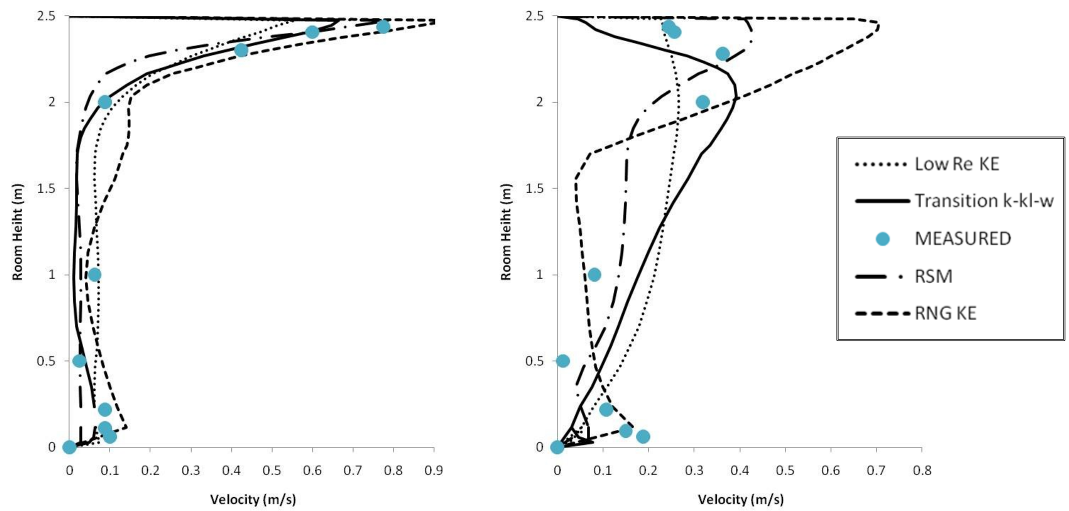



**Figure 12 Velocity profiles at x=1.4m and 3 m**

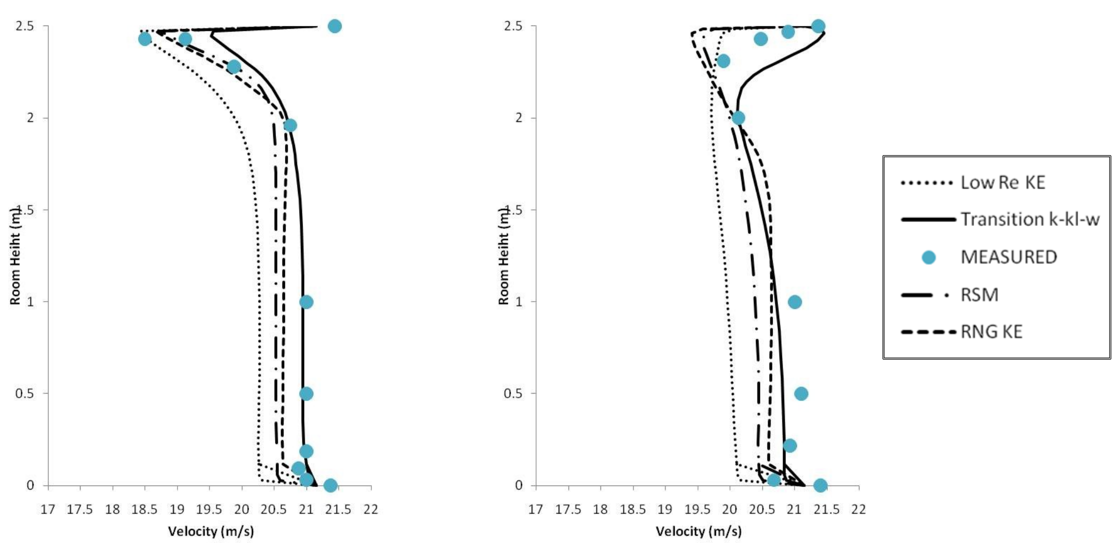



**Figure 13 Temperature profiles at x=1.4m and 3 m**

In Figure 13, temperature values with different turbulence models are compared with the measured data. The LKE model predicts temperature values approximately one degree lower than the measured values in the occupied zone. The RSM and RNG k-epsilon models predict good values of temperature in the near ceiling region, but they do not predict well in the occupied zone. Transition k-kl-w model predicts temperature values in the occupied zone more closely than the other models.

To summarize, of the four turbulence models studied, the Transition k-kl-w turbulence model is recommended for room air simulations as it predicts the most accurate values of velocity and temperatures in the occupied zone.

#### *4.4.2 Effect of radiation*

Out of three modes of heat transfer – conduction, convection and radiation, convection is the dominant mode of heat transfer in room air flows. However, radiation should not be neglected to accurately predict the room air temperatures. Thus to study the effect of radiation, two cases – "with and without enabling radiation" are considered and the temperature values are compared with the measured temperature values (as shown in Figure 14). The surface to surface radiation model is used for the case – "with radiation model". In this model, it is assumed that the medium separating the walls of the room does not participate in the radiation process, which is a reasonable approximation for air. The temperature values computed using the radiation model were in better agreement with the experimental measured data than without enabling radiation with a maximum of 0.5 $^0$C difference between simulated results by enabling radiation and measured data.

In practice, the simulation time increases by enabling the radiation model. Hence, an alternate solution is suggested, which is to increase the wall temperature by 1 $^{0}$C. The simulation results with wall temperatures were close with results obtained by enabling radiation model. However, this approximation applies to only specific problem and does not apply if one changes the inlet flow rate or inlet temperature. Moreover, the wall temperature will have a fixed single value.

Hence, it is suggested that the radiation model be incorporated in room air simulations to accurately predict the air temperatures.

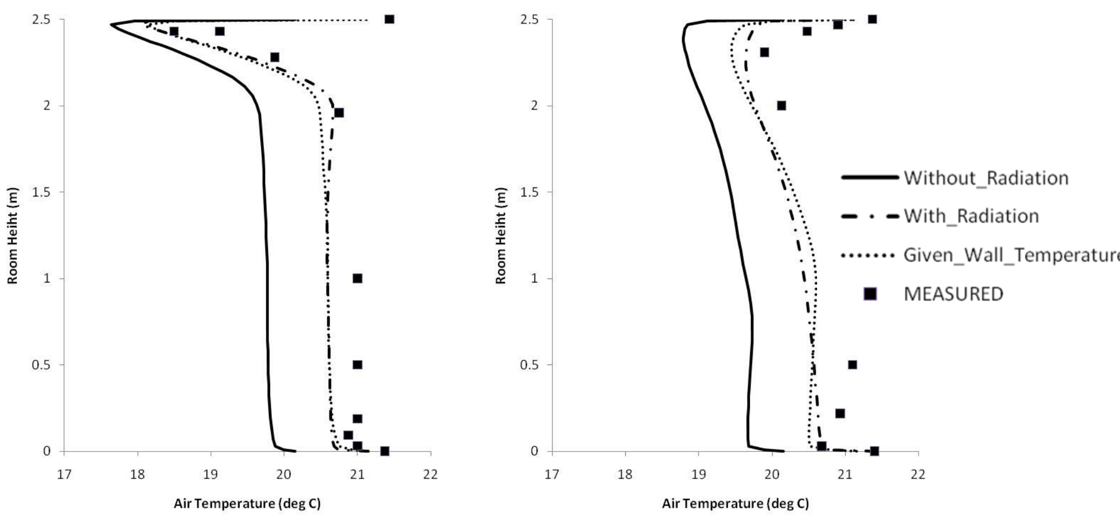



**Figure 14 Effect of radiation: Temperature profiles at x=1.4m and 3 m**

### *4.4.3 Effect of different diffuser inlet angles*

To study the effect of diffuser jet inlet angle on the jet momentum and room air velocities, the diffuser inlet angle was varied in the range of ± 10 degrees of the original diffuser inlet angle. Velocity profiles with different jet angles are compared with the measured data at two different locations in the room (as shown in Figure 15 and 16). The velocities in the near ceiling region are affected more with the change in inlet angles than the velocities near the floor. With a 30 degree inlet angle, the jet momentum is as much as 25% lower than the measured values in the near ceiling region. The momentum increases as the jet angle is increased. 50 deg seems to be the best fit in the occupied and near floor region, but it is over-predicted by 12% in the near- ceiling region. Velocities in most of the occupied zone (from y=0.5 m to y=2m) increases as the diffuser angle is reduced with a maximum 20 % deviation from the velocity computed by the original 40 degree angle.

This analysis shows that the diffuser inlet angle has an effect on jet momentum (mainly in the ceiling region) and velocity distribution inside the room. Hence, 40 degree angle is selected for the diffuser jet inlet angle.

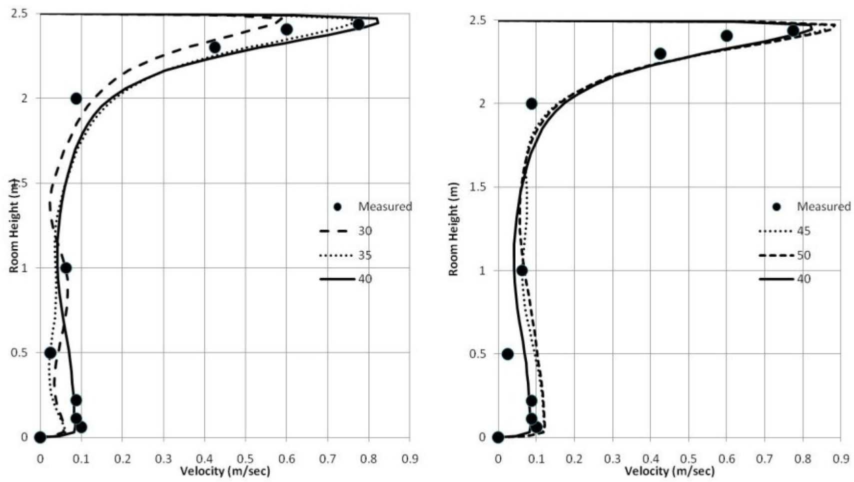



**Figure 15 Comparison of velocity profiles for different jet angles with measured data at x=1.4m**

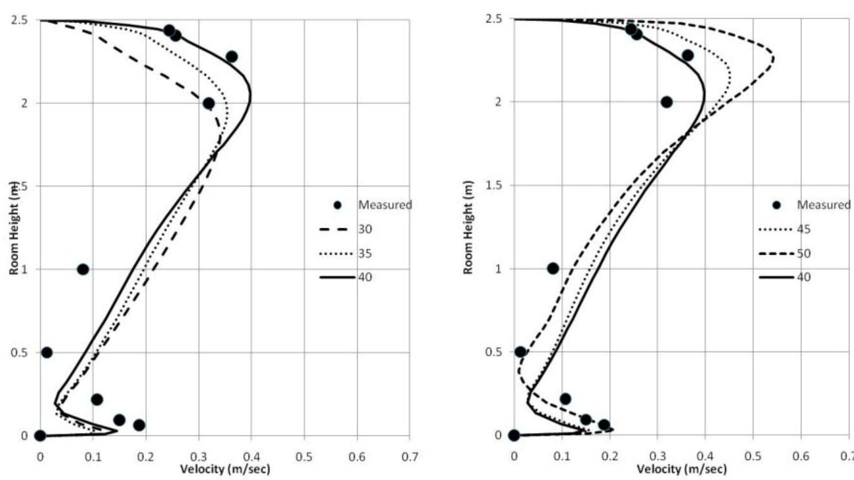
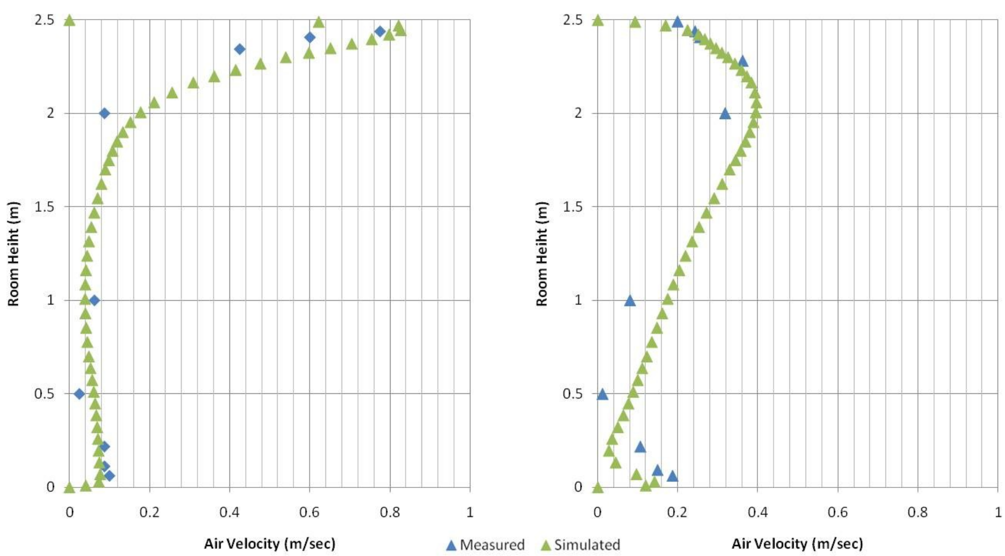



**Figure 16 Comparison of velocity profiles for different jet angles with measured data at x=3m**

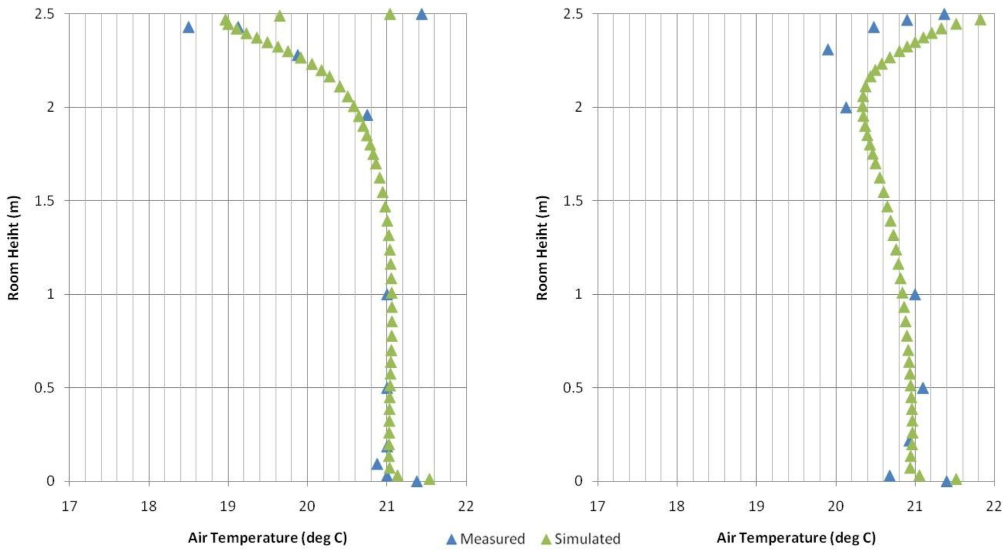
### *4.5 Validation of simulation results with measured data*

From the detailed study of parameters mentioned in section 3.1.1, the best fit parameters are determined, which are, Transition k-kl-w Turbulence model, incorporation of Radiation model & 40 degree inlet diffuser angle; and the Benchmark simulation is validated against the measured data. From the velocity and temperature profiles, shown in Figure 17 and 18, it can be seen that the simulation results are in good agreement with the measured data.




**(a)** **(b)**

**Figure 17 Comparison of velocity profiles for the measured and simulated results (a) at x=1.4m, (b) at x=3m**




**(a)** **(b)**

**Figure 18 Comparison of temperature profiles for the measured and simulated results (a) at x=1.4m, (b) at x=3m**

### *4.6 CFD simulations with different flow rates*

After the Benchmark simulation, the flow rate is varied to see the jet flow, velocity and temperature distributions inside the room. Two cases are considered.

- Case 1 – Flow rate - 1.5 ACH (half the Benchmark Case flow rate).
- Case 2 - Flow rate - 6 ACH (double the Benchmark Case flow rate).

In order to see the jet flow, path lines of particles are released from the inlet, as shown in Figures 19-21. The path lines are plotted only for the initial iterations to see how the flow is discharged from the diffuser and enters the occupied zone.

We can see from the figures above, for Case 1, the jet is attached to the ceiling for approximately one-fourth of the ceiling length and subsequently, most of the jet falls directly into the occupied zone. For the Benchmark Case, the jet is attached to the ceiling for almost half of the room length, whereas for Case 2, the jet directly hits the opposite wall due to its high momentum.

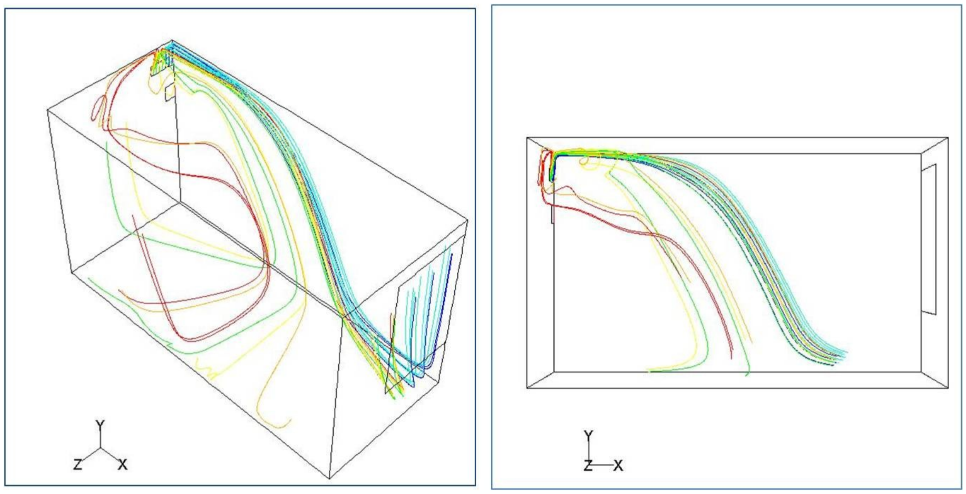



**Figure 19 Path lines for Case 1, jet released from inlet – Isometric and front view**

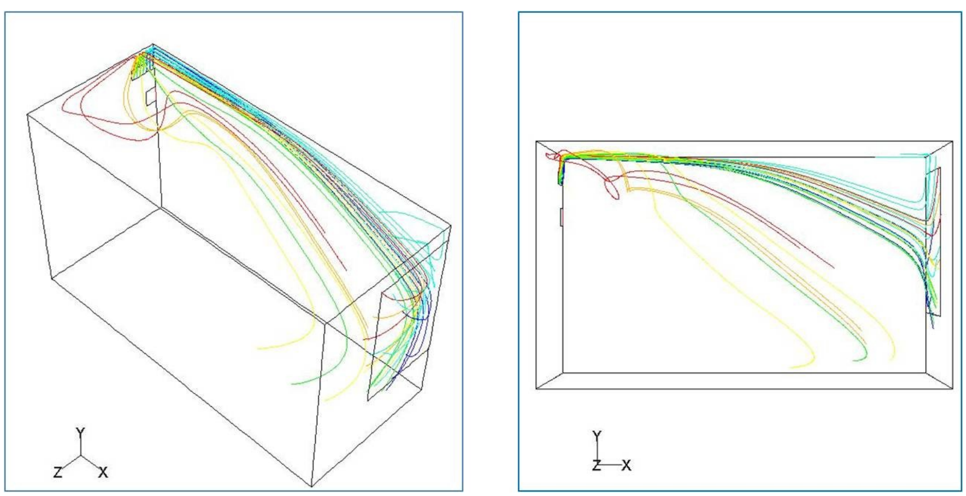



**Figure 20 Path lines for the Benchmark Case, jet released from inlet – Isometric and front view**

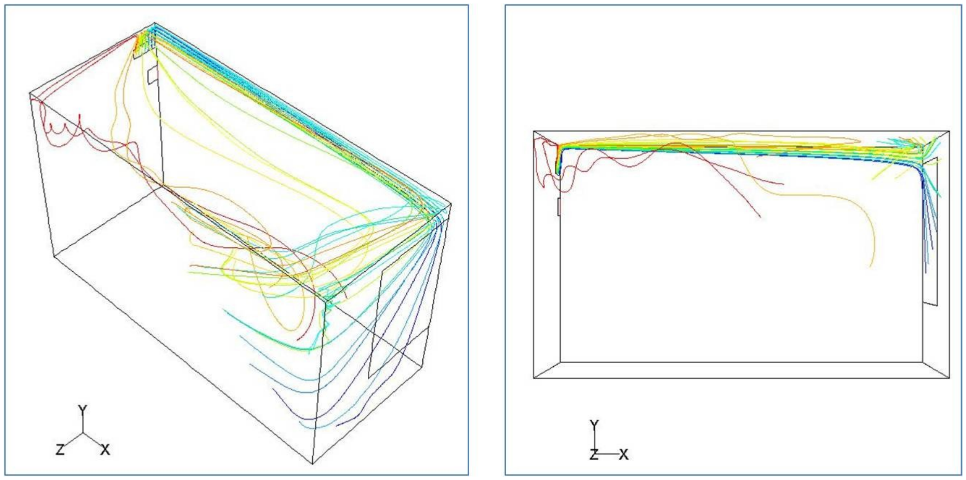



**Figure 19 Path lines for Case 2, jet released from inlet – Isometric and front view**

The velocity and temperature profiles are plotted at four "poles" placed in the symmetry plane along the lengthwise direction of the room (as shown in Figure 20).

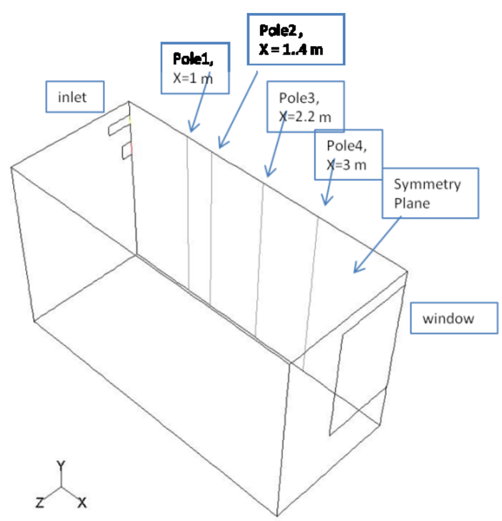



**Figure 20 Poles used for plotting velocity values**

From the velocity graphs of Figure 21, it is clearly seen that the jet momentum is directly proportional to the flow rate, i.e. the peak velocity at each pole approximately doubles as the flow rate doubles. The jet for Case 1 decayed after the second pole, whereas for the Benchmark Case it continued after the second pole and decayed between pole 2 and pole 3. For Case 3, it continued even beyond pole 3. The jet attachment length agrees with what we saw in the path lines plot. At pole 3, the velocity for 1.5 ACH is higher in the occupied zone than for the higher air flow rates; because the jet has already started falling into the occupied zone. Similarly, for 3 ACH, the jet has already begun to decay at pole 3 and started entering the occupied zone, as, between maximum jet velocity at pole 3 is 0.6m/sec at near y=2.5m whereas it is at 0.4m/sec at y=2m for pole 4.There is a considerable amount of variation in temperatures near the ceiling for the three flow rates (Figure 22). The temperatures are related to the velocities; hence for the 6 ACH flow rates, the temperature near the ceiling is lower till the jet reaches pole 3. In the occupied zone, the temperatures are higher for the lowest flow rate, due to the insufficient cold air supply and mixing of the cold jet with the room air.

The momentum of the jet, along with its behavior and circulation inside the room, can be well predicted with the help of path line plots and profiles of velocity and temperatures in different sections of the room. When the flow rate is much lower than the design flow rate, the jet directly falls into the occupied zone, which leads to improper mixing in the room so that some areas of the room are colder than the other areas.

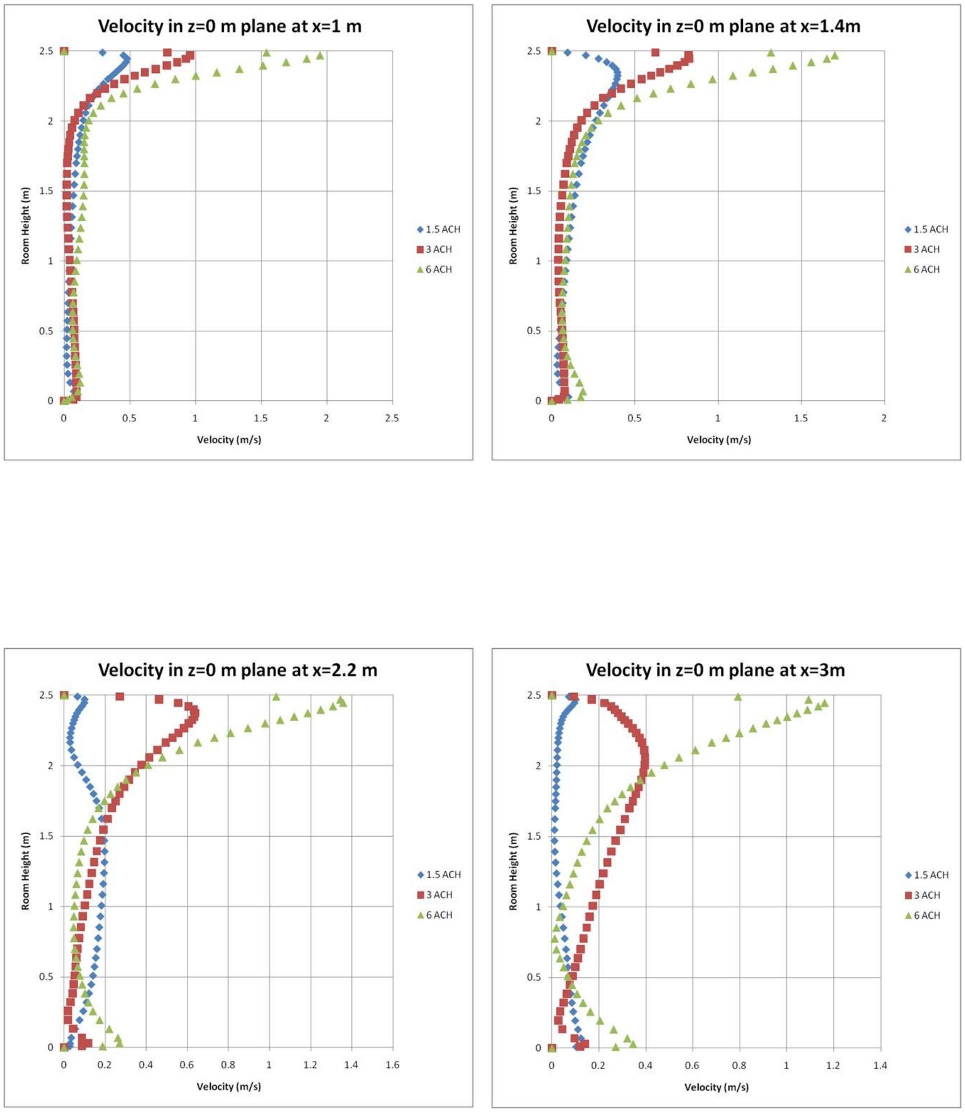



**Figure 21 Velocity profiles for Case 1, Case 2 and Benchmark Case at four poles**

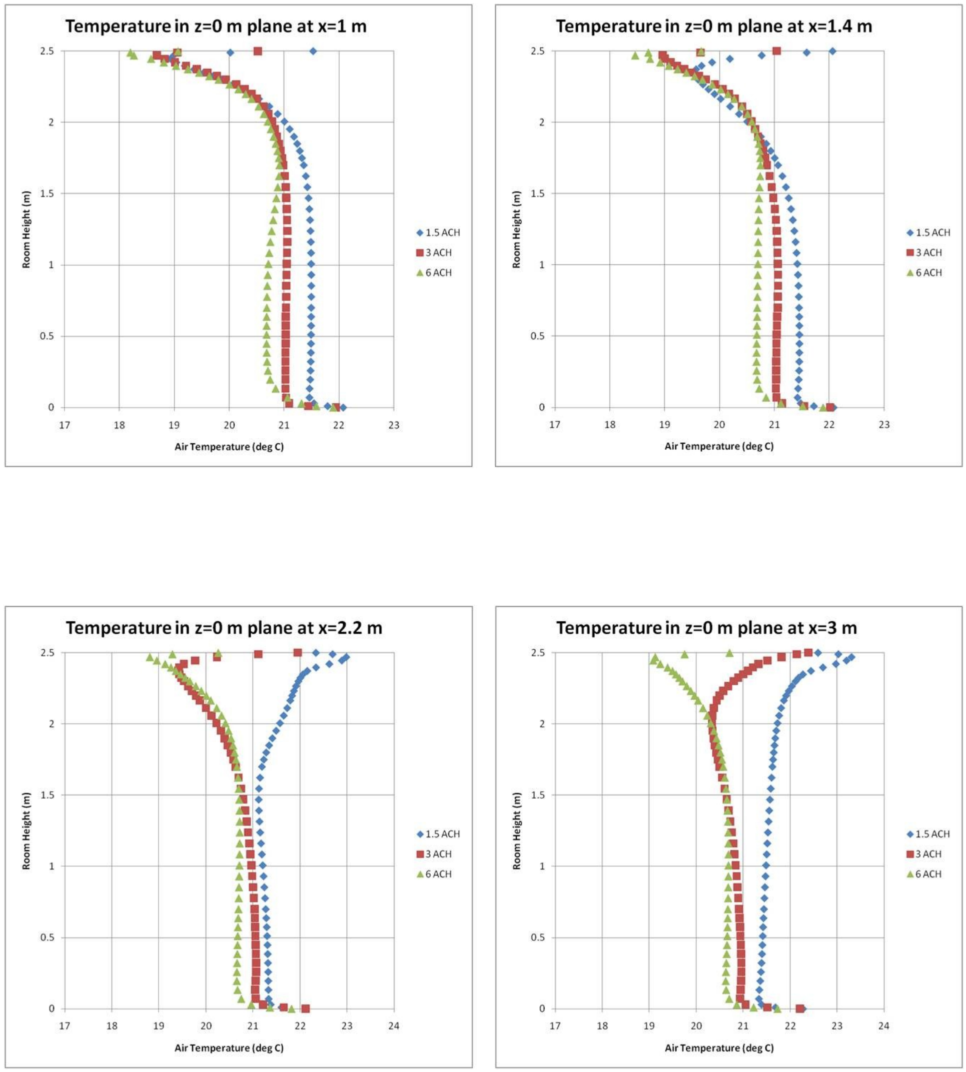



**Figure 22 Temperature profiles for Case 1, Case 2 and Benchmark Case at four poles**

Since many variable air volume units run at 50% of the design flow rate or less much of the time, the flow rates will be reduced from 50 to 15 % of the design flow rate in the remaining study. The heat loading in terms of external air temperature is reduced according to the reduction in flow rates in order to maintain a constant set point temperature of 20 $^{0}$C.

# 5. REDUCED DIFFUSER FLOW RATES

The diffuser flow rates are reduced down to 15 % of the design flow rate in order to study the jet momentum, draft and temperature distribution inside the room. The heat gain from the external window and outside walls of the office room is reduced accordingly such that the return air temperature is approximately constant as it will be with a constant thermostat set- point. The inlet temperature is maintained constant at $15^0$C.

### *5.1 Case – 15% of design diffuser flow rate*

This is the extreme case, where the flow rate is reduced down to 15% of the design flow rate of the diffuser. Figure 23 shows the path lines of air particles released from the diffuser outlet. We can see that due to the lower momentum of the jet, the cold air does not attach itself to the ceiling and directly falls down to only one side of the room, thus not mixing with the rest of the air in the room. This phenomenon is also called "dumping" of the cold jet.

The jet drop is further shown in Figure 24, where the contours of temperature are plotted on the symmetry plane in the room. The temperature in the occupied zone below the diffuser is at 18 deg C, whereas it is maintained at almost 20.5 deg C in the other regions of the room.

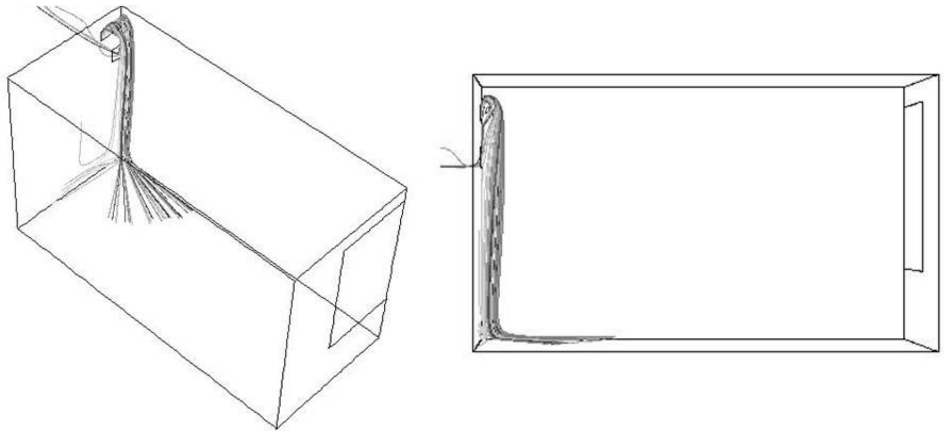


**Figure 23 Isometric and front view of path lines of particles released from diffuser for 15 % flow rate**

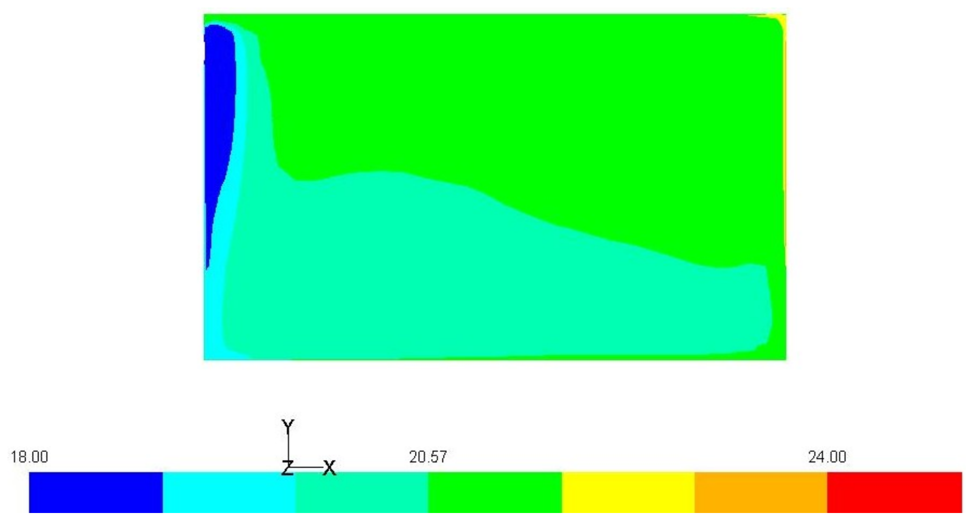



**Figure 24 Contours of temperature at the symmetry plane for 15% flow rate**

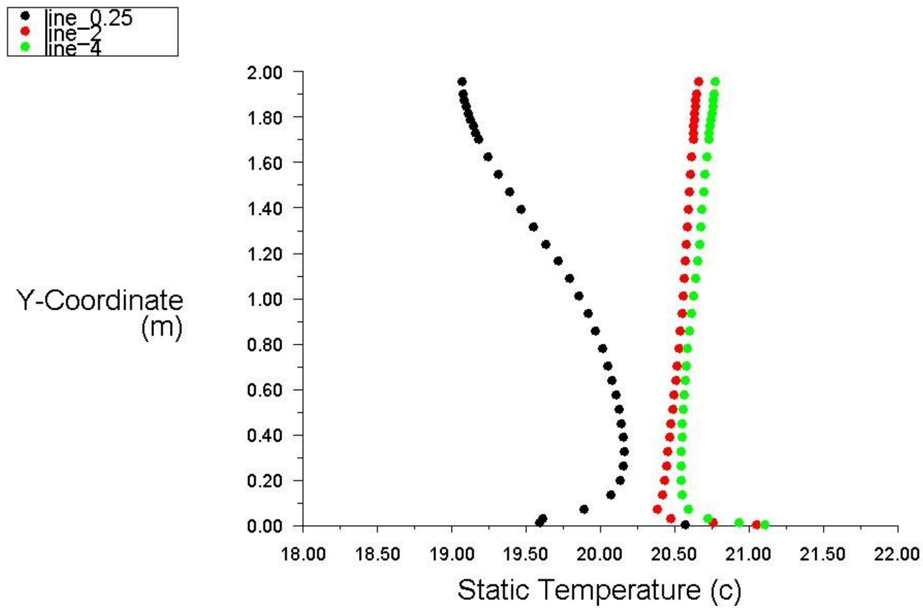



**Figure 25 Temperature profiles at the symmetry plane for x=0.25, 2 and 4m for 15% flow rate**

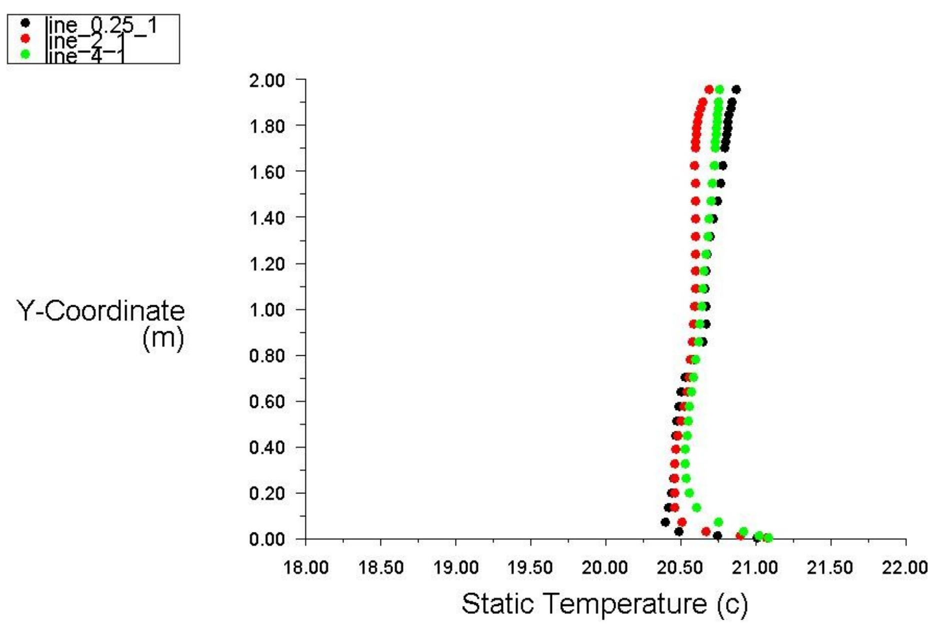



**Figure 26 Temperature profiles at the near- wall plane for x=0.25, 2 and 4m for 15% flow rate**

The temperature profiles at lines along the lengthwise direction of the room are plotted for two planes – at the center of the room (symmetry plane) and near the wall plane, which is 1 ft away from the wall (as shown in Figures 27 and 28).

We can see that at the symmetry plane for x=0.25m (where the cold jet falls into the occupied zone), the temperature varies from 19 to 20.25 deg C. The temperature values at the near-wall region are almost constant along the lengthwise direction of the room, which is due to the fact that the cold jet is not entering this region. High temperatures of air at the near wall plane are due to the fact that there is direct heat gain from the walls of the room.

Hence, a person standing immediately below the diffuser can feel the cold jet draft hitting directly on their head whereas a person standing near the window or wall will not feel the low temperatures.

### *5.2 Case – 20% of design diffuser flow rate*

The flow rate is set at 20% of the design flow rate of the diffuser. Figure 27 and 30; show the path lines of fluid particles and contours of temperature at the symmetry plane. The cold jet attaches itself to the ceiling till x=0.5 m (longitudinal direction) and then falls down to the occupied zone.

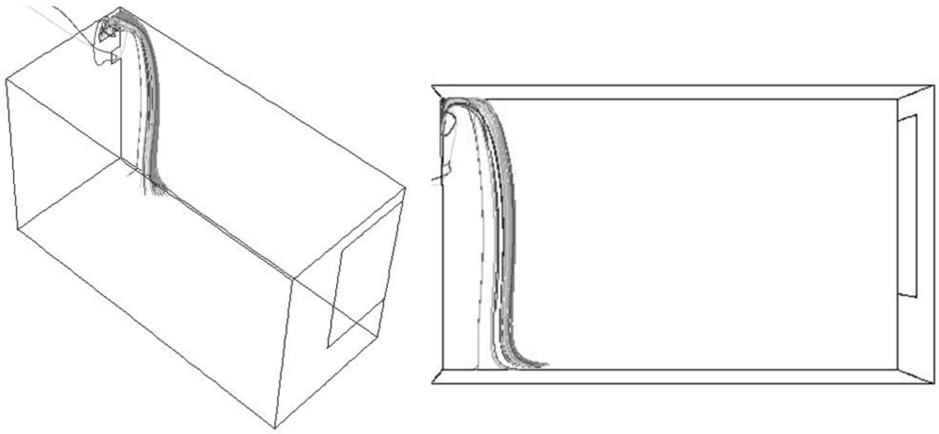


**Figure 27 Isometric and front view of path lines of particles released from the inlet for 20 % flow rate**

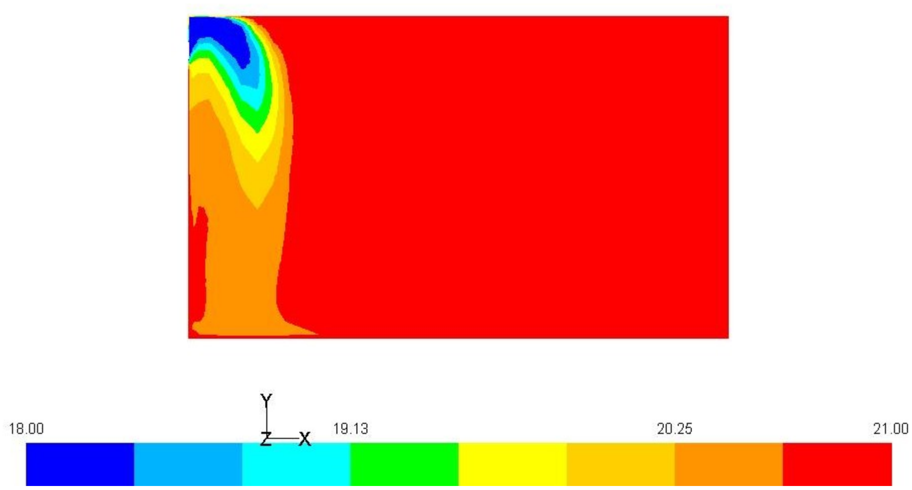



**Figure 28 Contours of temperatures at the symmetry plane for 20% flow rate**

As seen from Figure 29, for x=0.5m (which is closer to the diffuser side), the temperature varies from 18.5 to 20.5 $^{0}$C between 2m height and the floor, whereas for x = 2, 4 m (which are further away from the diffuser side) temperatures vary by less than 0.5 $^{0}$C from floor to 2m height in the room. The variation of 2 $^{0}$C vertical temperature difference at x=0.5m is due to the reason that the cold jet drops in the occupied zone immediately below the diffuser.

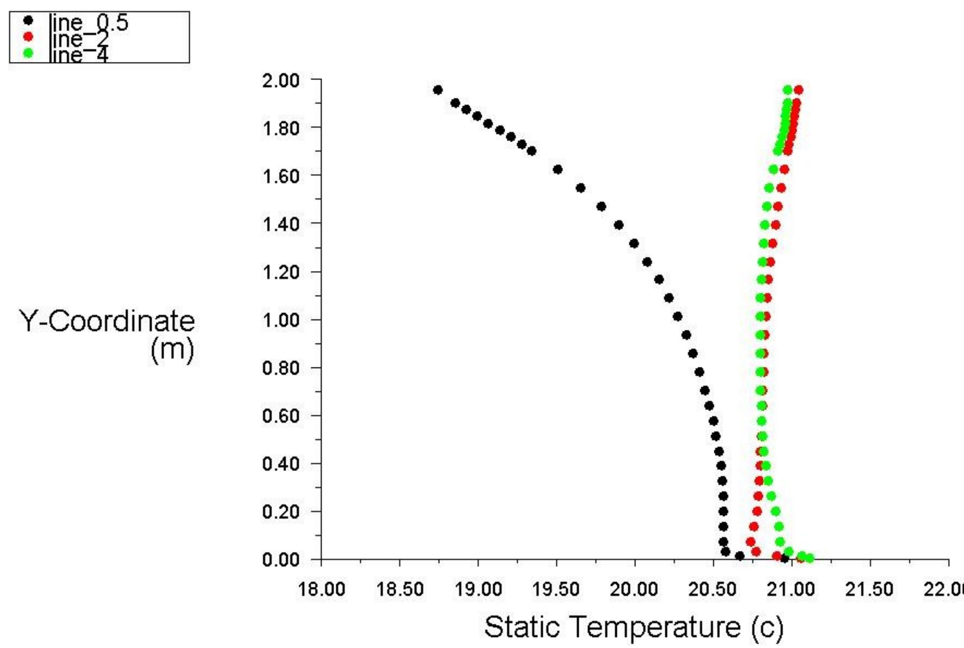



**Figure 29 Temperature profiles at the symmetry plane for x=0.5, 2 and 4m for 20% flow rate**

Temperature profiles for different x values at the near - wall plane (1 ft away from the wall) are shown in Figure 30. It is observed that the temperatures in the near- wall plane are almost constant from floor to ceiling, which is due to the fact that the low velocity jet does not reach this area, and hence the air is well mixed.

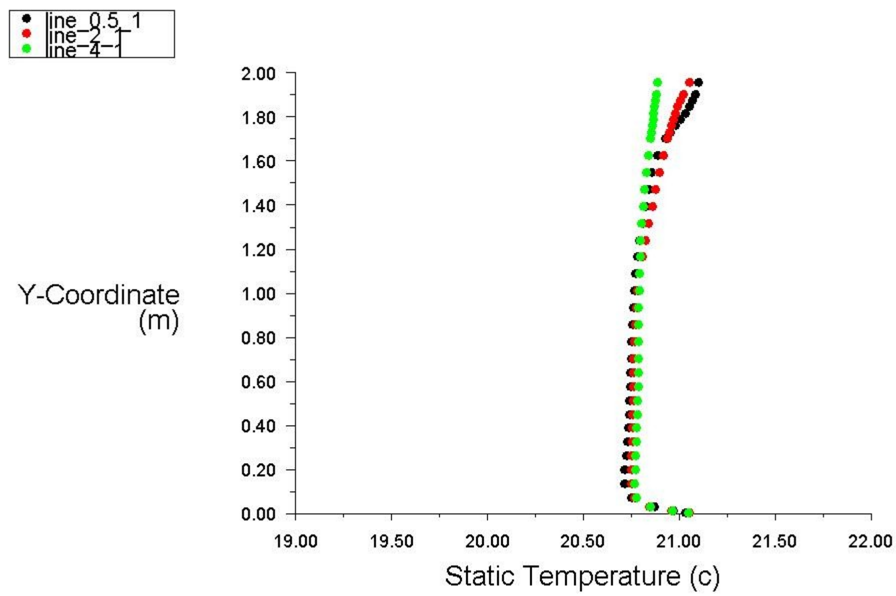



**Figure 30 Temperature profiles at the near- wall plane for x=0.5, 2 and 4m for 20% flow rate**

### *5.3 Case – 30% of design diffuser flow rate*

The flow rate is 30% of the design flow rate and the thermal loading is given accordingly. The path lines and temperature variation in the symmetry plane are shown in Figure 31 and 34.The jet attaches itself to the ceiling till x=1.5 m (due to the Coanda effect) and then drops into the occupied zone. The jet spread is broader as compared to the previous cases.

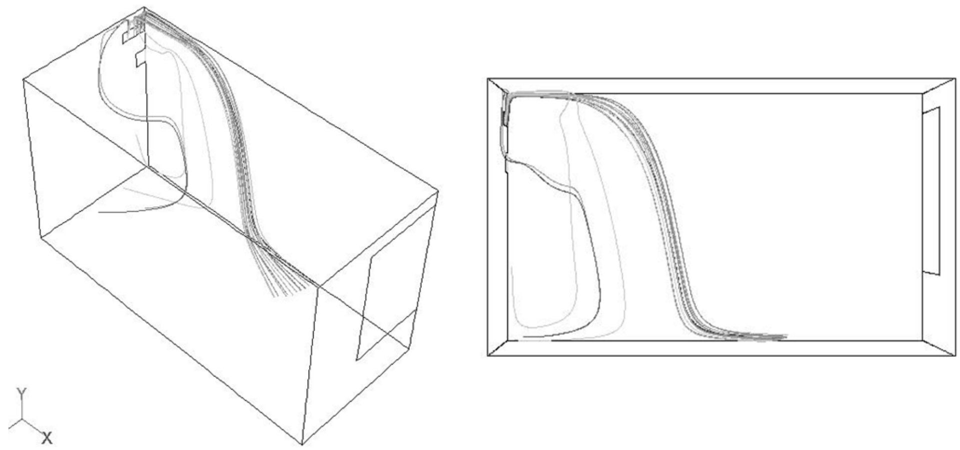



**Figure 31 Isometric and front view of path lines of particles released from the inlet for 30 % flow rate**

From Figure 33 and 36, we can see that the temperature near the ceiling varies from 18 to 22 deg C along the length of the room, but there is a maximum of 1 $^{0}$C variation near the head level (at y = 1.8 m). Hence, the influence of dumping is relatively small in the occupied zone. The temperature is approximately 0.25 – 0.5 $^{0}$C lower at the line x=1.5 m (as compared with the other locations of the room in the lengthwise direction) because the jet enters the occupied zone near this pole. The temperatures are constant at the near- wall plane where there is no direct impact of the cold jet.

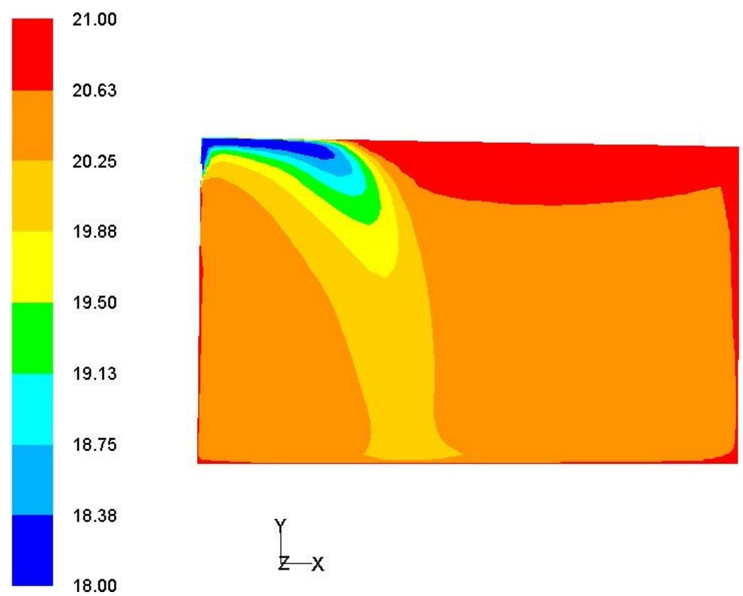



**Figure 32 Contours of temperature at the symmetry plane for 30% flow rate**

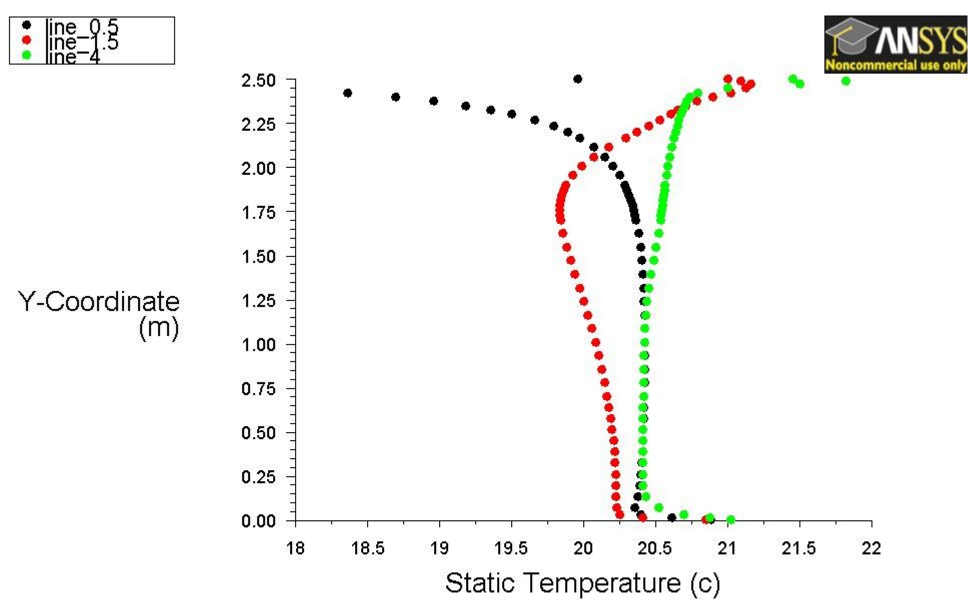



**Figure 33 –Temperature profiles at the symmetry plane for x=0.5, 1.5 and 4 m for 30% flow rate**

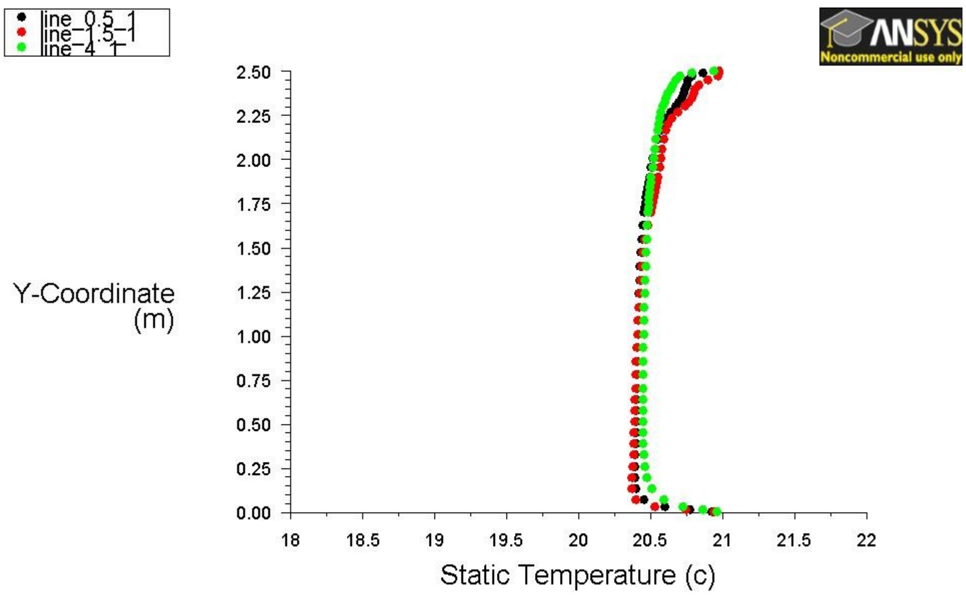



**Figure 34 - Temperature profiles at the near-wall plane for x=0.5, 1.5 and 4 m for 30% flow rate**

### *5.4 Case – 40% of design diffuser flow rate*

From the path lines plot and temperature plot (Figure 35 and 38), we can see that the diffuser jet attaches itself to the ceiling for approximately one-third of the room length and then drops into the occupied zone. From the temperature contour plot at the symmetry plane, it is seen that the jet temperatures as low as 20C are only present near the ceiling region and the temperature distribution in the occupied zone is almost all between 20.5 – 21.0 $^{0}$C, with only a small region between 20.0 – 20.5 $^{0}$C.

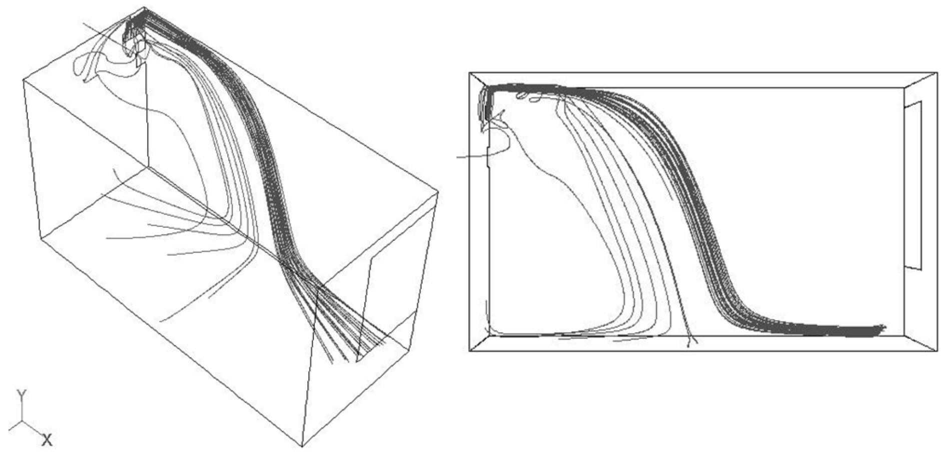



**Figure 35 Isometric and front view of path lines of particles released from the inlet for 40 % flow rate**

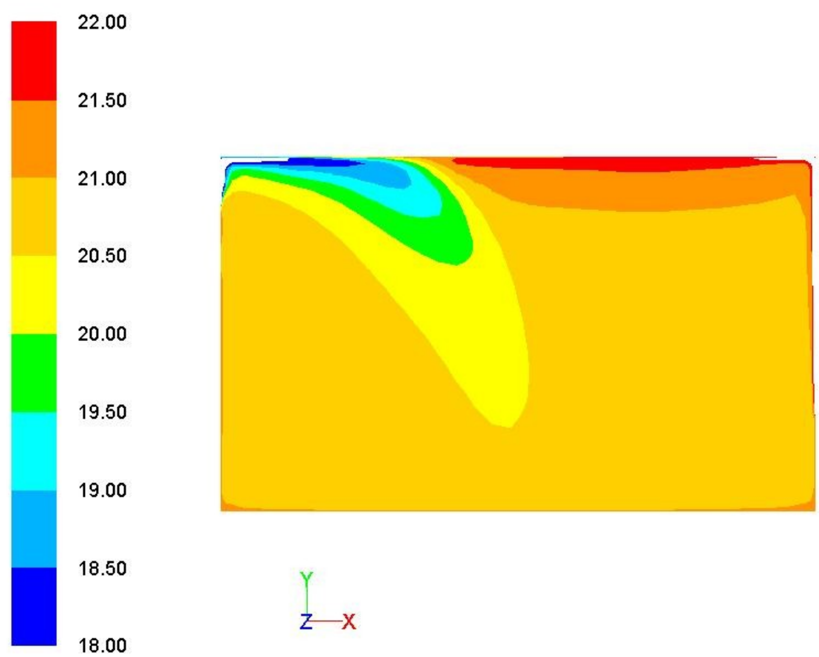



**Figure 36 Contours of temperature at the symmetry plane for 40% flow rate**

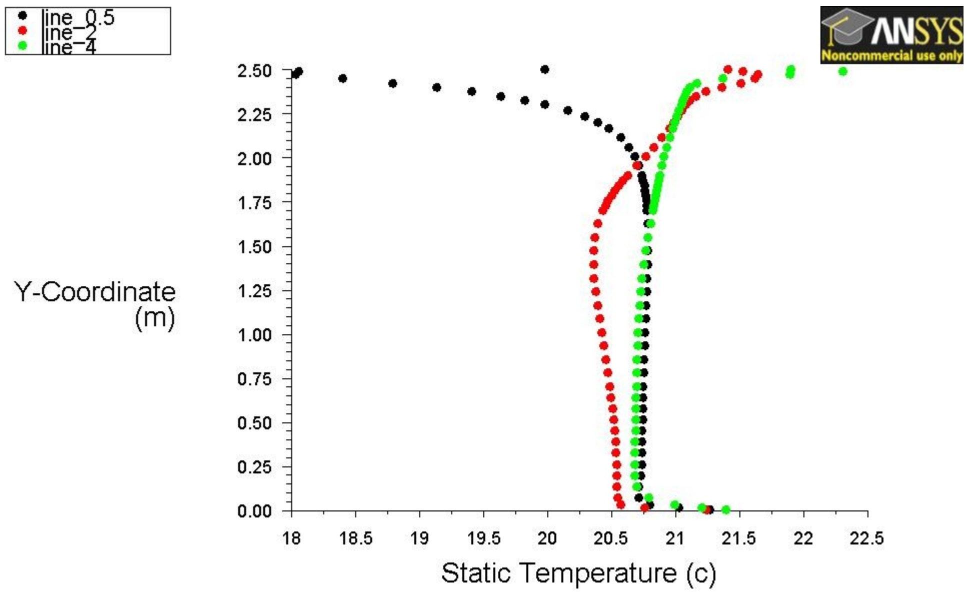



**Figure 37 Temperature profiles at the symmetry plane for x=0.5, 2 and 4 m for 40% flow rate**

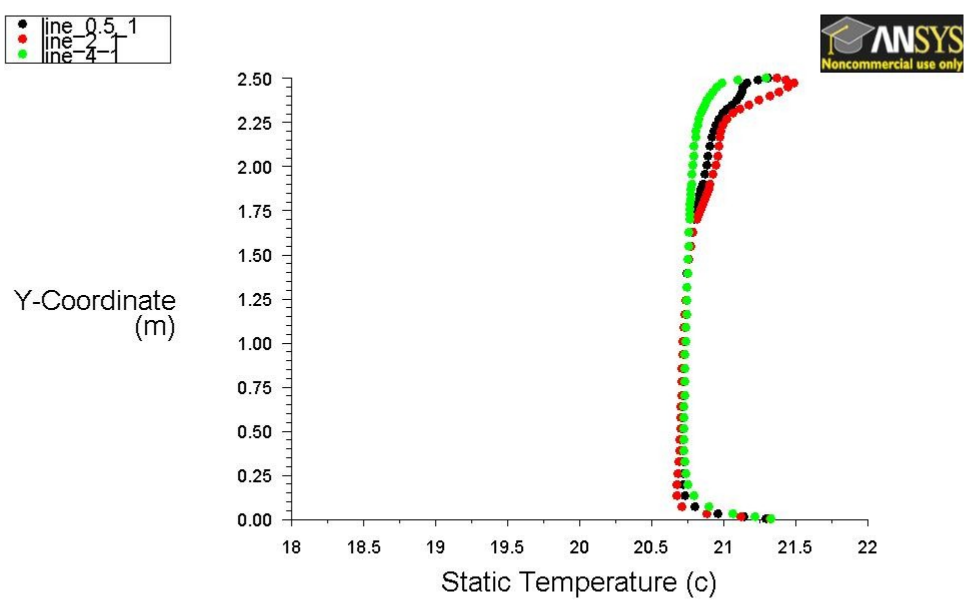



**Figure 38 Temperature profiles at the near-wall plane for x=0.5, 2 and 4 m for 40% flow rate**

From the temperature plots (Figure 37 and 40), it is seen that temperature varies from 18 to 22.5 $^0$C in the near ceiling region. The temperature at x=0.5m at the symmetry plane is a bit lower (maximum of 0.5 $^0$C difference) than the other locations because of the jet entrainment into the occupied zone. However, the jet diffuses into the room air by the time it reaches the occupied zone, and hence one can experience almost uniform temperature in the occupied zone, which is in the range of 20.25-20.8 $^0$C from floor to y=2.

***5.5 Case – 50% of design diffuser flow rate***

The diffuser flow rate is given 50 % of the design flow rate, which is a common case as variable air volume units run at 50 % of the design flow rate much of the time. From the path lines plot (Figure 39) and temperature contour plot (Figure 40), it is seen that the jet is attached to the ceiling for about half the room length and then diffuses towards the wall at the opposite end. Thus, the primary jet air does not directly enter into the occupied zone at all.

From the temperature plots (Figure 41 and 44), we can see that the temperatures vary from 18 to 22 $^0$C near the ceiling region. At x=0.5m, in the occupied zone i.e. floor to head level (1.8 m), the temperature is almost constant, whereas for x=2m, there is a variation of 1$^0$C. At x=4m (which is nearer to the other end of the room), the temperature is lower as the primary jet slides through the wall and enters the room.

Overall, the temperatures in the occupied zone are maintained almost constant, with a maximum variation of 0.5 $^0$C from floor to y=1.8 m.

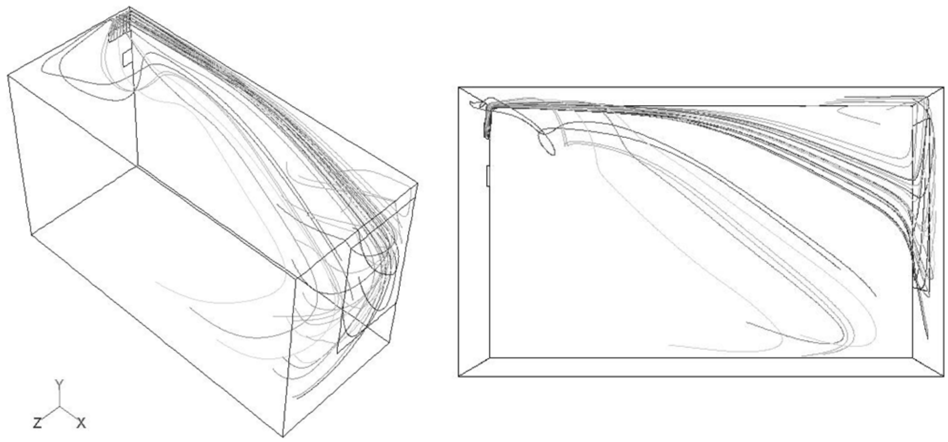



**Figure 39 Isometric and front view of path lines of particles released from the inlet for 50 % flow rate**

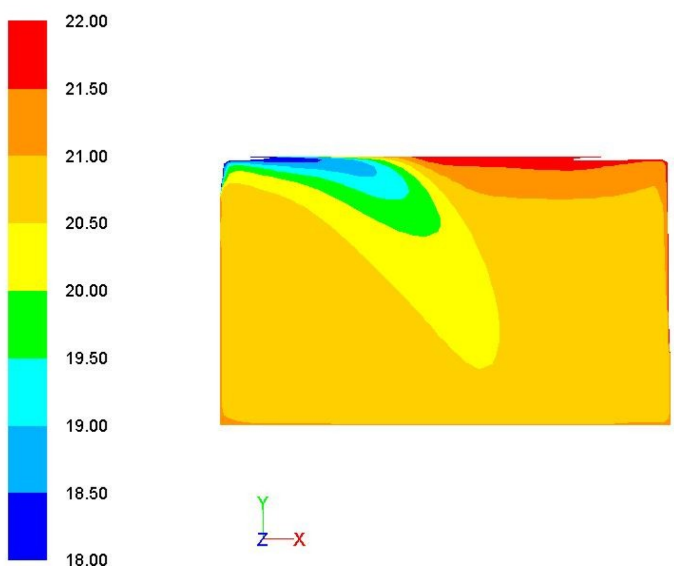



**Figure 40 Contours of temperature on the symmetry plane for 50% flow rate**

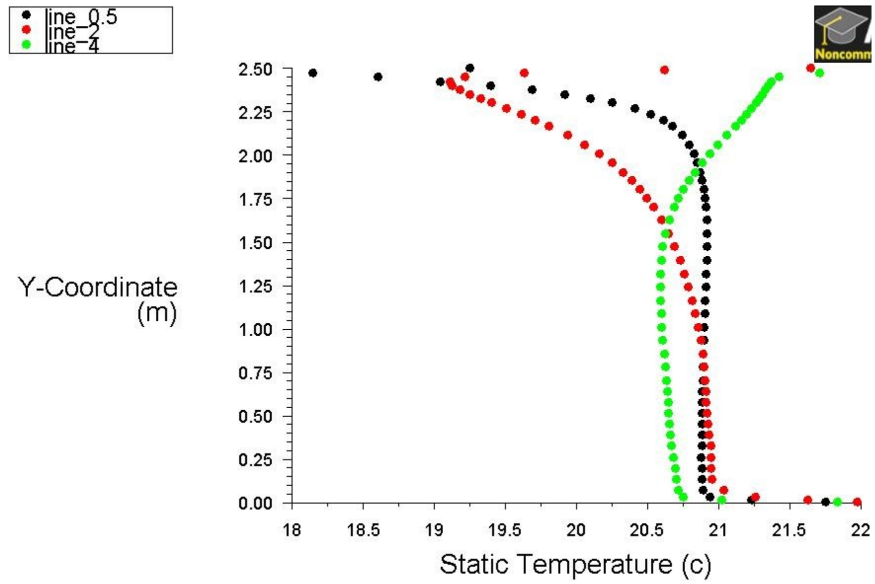



**Figure 41 Temperature profiles at the symmetry plane for x=0.5, 2 and 4 m for 50% flow rate**

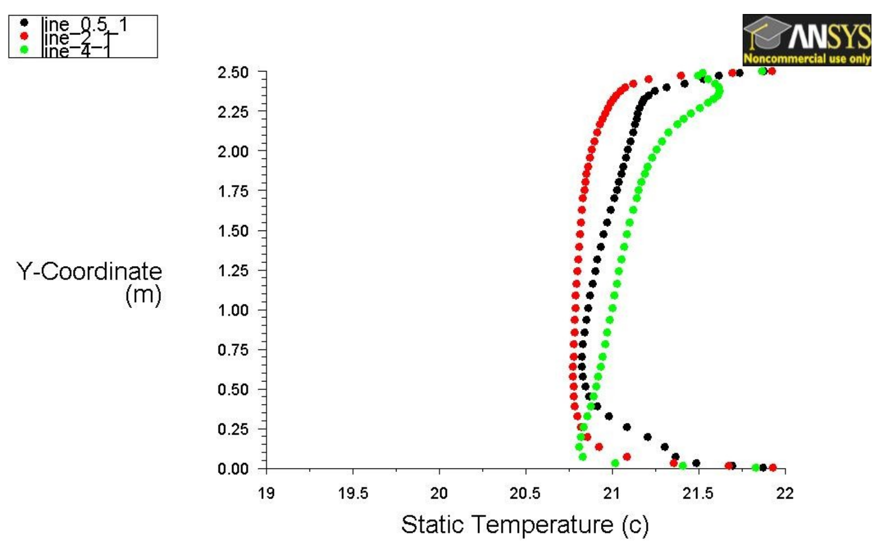



**Figure 42 Temperature profiles at the near - wall plane for x=0.5, 2 and 4 m for 50% flow rate**

# 6. SUMMARY OF RESULTS

The contours of temperature at the symmetry plane for reduced flow rates and loading are shown in the Figure 43.The jet draft and attachment length can be observed from the below figure, which is tabulated in Table 1.

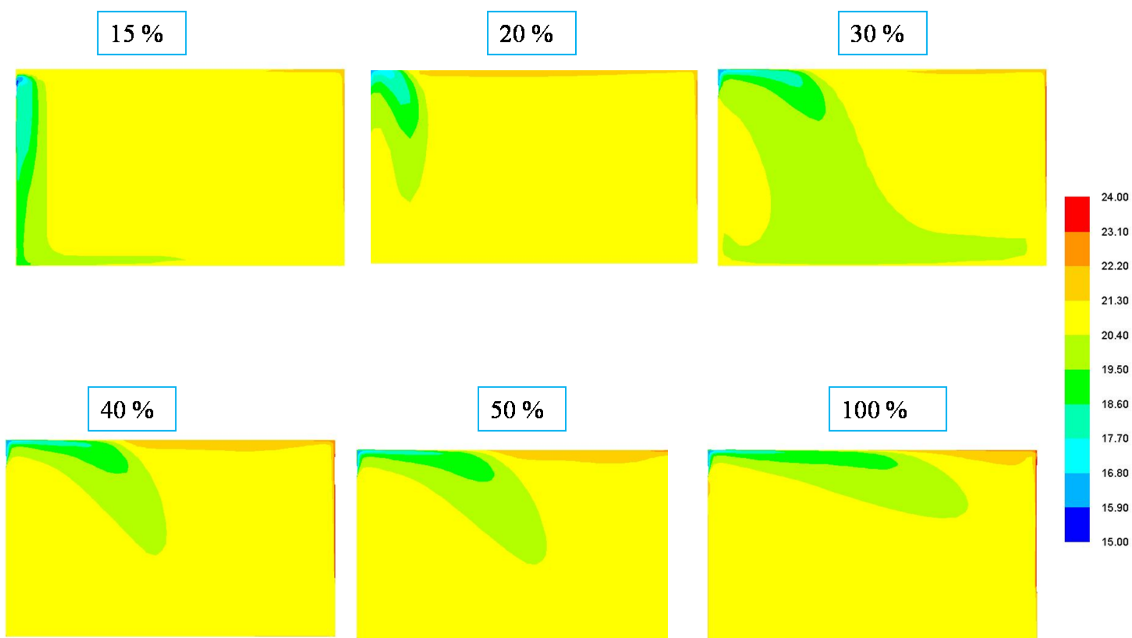



**Figure 43 Contours of temperature on symmetry plane for reduced flow rates and loading**

From Table 1, it is observed that for 15% of flow rate, the jet does not attach itself to the ceiling and directly dumps into the occupied zone. Also, the jet attachment length increases with the increase in inlet flow rate.

**Table 1 Jet attachment length for reduced flow rates**

| **Flow rates** | 15% | 20% | 30% | 40% | 50% | 100% |
|---|---|---|---|---|---|---|
| **Jet Attachment Length** | No Attachment | 0.25 m | 0.75 m | 1.2 m | 1.5 m | 2 m |

The minimum, maximum values of temperatures and typical vertical temperature distribution in the occupied zone for reduced flow rates and loading are tabulated in Table 2. It is observed that for flow rates below 30 %, the typical vertical temperature distribution is 2 $^0$C. The maximum value for vertical temperature distribution is 0.5 $^0$C for the designed flow rate.

**Table 2 Minimum, maximum and vertical temperature distribution in the occupied zone for reduced flow rates**

| **Flow rates** | 15% | 20% | 30% | 40% | 50% | 100% |
|---|---|---|---|---|---|---|
| **Minimum Temperature ($^0$C)** | 18 | 18.5 | 19.5 | 19.6 | 19.8 | 20.1 |
| **Minimum Temperature ($^0$C)** | 21 | 21.2 | 21 | 21.2 | 21.1 | 21.1 |
| **VTD Jet drop region ($^0$C)** | 2 | 2 | 0.75 | 0.8 | 0.7 | 0.5 |

### *6.1 Thermal Comfort*

The thermal comfort is quantified in terms of percentage dissatisfied index based on Fanger's thermal comfort model (Air turbulence and sensation of draft, 1988).The maximum value of percentage dissatisfied index (PD) in the occupied zone is shown in the Table 3. It is observed that, 25 % of the people will be dissatisfied with 15 % of design flow rate. The value of PD increases as the flow rate is increased.

**Table 3 Maximum value of percentage dissatisfied index for reduced flow rates and loading**

| **Flow rates** | 15% | 20% | 30% | 40% | 50% |
|---|---|---|---|---|---|
| **Percentage Dissatisfied** | 25.73 | 15.08 | 11.77 | 12.01 | 12.21 |

It is observed that the value of PD is almost constant, about 12 %, for flow rates above 30% of design flow rates, which is due to the reason that the temperature distribution in the occupied zone is almost constant.

# 7 . CONCLUSIONS AND FUTURE WORK

The jet momentum, draft and temperature distributions inside the room are studied for the impact of reduced flow rates and loading. The thermal comfort for low flow settings is quantified in terms of vertical temperature distribution and percentage dissatisfied.

The following observations are made:

i. From a parametric study of parameters, it is found that -

   a) The transition k-kl-w turbulence model seems to predict better values of velocity and temperatures for room air simulations.

   b) Radiation model should be incorporated in the CFD simulations to accurately predict the room air temperatures, as there is difference of $1^0$C between simulated results with and without radiation throughout the room height.

   c) Jet momentum is sensitive to different diffuser inlet angles (in the range of ± 10 degrees). Thus proper care should be taken for specifying the inlet angle. 40 degree inlet angle is selected as the diffuser inlet angle.

ii. As we reduce the flow rate, mixing of cold air with the room air is reduced and the jet directly falls into the occupied zone. The temperatures where the jet falls are low temperature regions whereas other regions are relatively hotter resulting in non-uniform temperature distributions with a temperature difference of as much as 3 $^0$C in the occupied zone for 15 and 20 % of flow rate.

iii. For very low diffuser flow rates (below 30 % of design flow rate), it is recommended to put the desk away from the diffuser as one may feel the cold jet draft hitting directly on the head and thus to avoid the dumping of cold jet.

iv. At flow rate equal to 30 % of the design flow rate, the influence of the jet is not much considerable with a temperature variation of maximum of 1 $^0$C in the occupied zone which is from floor to head level i.e y=2m.

v. For flow rates above 30 % of the design flow rate, the jet influence in the occupied zone is reduced, and there is an almost uniform temperature distribution with a maximum deviation of 0.75$^0$C in the occupied zone.

The impact of reduced flow rates and loading can be further studied on different types of diffusers and room configurations to achieve general guidelines for occupant spacing in a room for low flow settings and loading.

# REFERENCES

ASHRAE. 2004. Thermal environmental conditions for human occupancy, ANSI/ASHRAE Standard 55-2004.

ASHRAE. 2009. ASHRAE Handbook – Fundamentals. 2009. American Society of Heating, Refrigerating and Air Conditioning Engineers (ASHRAE), Inc, Atlanta. Chapter 20.

Blomqvist, C.1991. Measurement of test case E (mixed convection, isothermal), IEA Annex 20, Research Item 1.16, Technical Report, National Swedish Institute for Building Research (NSIBR), Gavle, Sweden.

Chen, Q. 1990. Simulation of test case E (mixed convection summer cooling ), IEA Annex 20, Research Item 1.20, Technical Report, Eidgenössische Technische Hochschule Zürich (ETH Zurich), Switzerland.

Chen, Q.1995. Comparison of different k-ε models for indoor airflow computation, Numerical heat transfer, PartB, Fundamentals, 28:353-369.

Chen, Q., and Jiang, Z.1996. Simulation of complex air diffuser with CFD technique, Proc. of ROOMVENT, Vol.1: 227-234.

Chen, Q., and Moser, A.1991. Simulation of a multiple- nozzle diffuser, Proc. of 12$^{th}$ AIVC Conference, Vol.2:1-14.

Emvin, P., and Davidson, L.1996. A numerical comparison of three inlet approximations of the diffuser in case E1Annex 20, Proc. of ROOMVENT, Vol.1: 219-226.

Ewert, M., Renz, U., Vogl, N., and Zeller, M.1991. Definition of flow parameters at the room inlet devices - measurements and calculations, Proc. of 12$^{th}$ AIVC

Conference, Vol.3: 231-237.

Fanger, P.O., Christensen, N.K., Perception of draught in ventilated spaces.1986. Ergonomics 29: 215-235.

Fanger, P.O., Melikov, A.K., Hanzawa, H., and Ring. J.1988. Air turbulence and sensation of draft, Energy and Buildings 12:21-39.

Fluent. 2009. Ansys Fluent 12.0 user's guide. ANSYS, Inc., Canonsburg, Pennsylvania.

Fossdal, S., 1990. Measurement of test case E (mixed convection, summer cooling), IEA Annex 20, Research Item 1.17, Preliminary Report, Norwegian Building Research Institute (NBRI), Oslo, Norway.

Heikkinen, J. 1991a. Measurements of test cases B2, B3, E2, E3 (isothermal and summer cooling cases), IEA Annex 20, Research Item 1.16 and 1.17, Technical Report, Technical Research Centre, Espoo, Finland.

Heikkinen, J. 1991b. Modeling of supply air terminal for room air flow simulation, Proc. of 12th AIVC Conference, Vol.3: 213-230.

Huo, Y., Zhang, J., Shaw, C., and Haghighat, F.1996. A new method to describe boundary conditions in CFD simulation, Proc. of ROOMVENT,Vol.2 : 233-240.

IEA.1993. Annex 20: Room air and contaminant flow, evaluation of computational methods, Subtask-1, Summary Report, ed. A.D Lemaire, TNO Building and Construction Research, Delft, Netherlands.

MCNair, H.P.1973.A preliminary study of the subjective effects of vertical air temperature gradients. British Gas Corporation Report WH/T/R&D/73/94, London.

McNair, H.P., and Fishman, D.S.1974.A further study of subjective effects of vertical air temperature gradients. British Gas Corporation Report WH/T/R&D/73/94, London.

Nielsen, P.V.1989. Representation of boundary conditions at supply openings, IEA, Annex 20, Internal report, Aalborg University, Copenhagen, Denmark.

Nielsen, P.V.1991. Models for the prediction of room air distribution, Proc. of 12th AIVC Conference, Vol.1:55-71.

Nielsen, P.V, 1992. Description of supply openings in numerical models for room air distribution, American Society of Heating, Refrigerating and Air Conditioning Engineers (ASHRAE) Transactions., 98(1): 963-971.

Nielsen, P.V.1998. The selection of turbulence models for prediction of room air flow, American Society of Heating, Refrigerating and Air Conditioning Engineers (ASHRAE) Transactions., 104(1): SF-98-10-1.

Olesen, B.W., M. Scholer, and Fanger, P.O.1979.Vertical air temperature differences and comfort.Danish Building Research Institue, Copenhagen, Denmark.

Rock, B.A, and Zhu, D. 2002. Designer's guide to ceiling-based air diffusion, American Society of Heating, Refrigerating and Air Conditioning Engineers (ASHRAE), Atlanta.

Skovgaard, M., and Nielsen, P.V.1991.Modeling complex inlet geometries in CFD applied to air flow in ventilated rooms, Proc. of 12th AIVC Conference, Vol.3: 183-200.

Srebric, J., and Chen, Q.2001a. A method of test to obtain diffuser data for CFD modeling of room airflow, American Society of Heating, Refrigerating and Air Conditioning Engineers (ASHRAE) Transactions, 107(2): 108-116.

Srebric, J., and Chen, Q. 2001b. Boundary conditions for diffusers in room air distribution calculations, Proc. of CLIMA 2000 World Congress, Napoli, Italy.

Srebric, J. and Chen, Q. 2002.Simplified numerical models for complex air supply diffusers, International Journal of HVAC&R Research, 8(3): 277-294.

Svidt, K.1994. Investigations of inlet boundary conditions for numerical prediction of air flow in livestock buildings, Aalborg University, Copenhagen, Denmark.

Tennekes, H., and Lumley, J.L.1972. A first course in turbulence, MIT Press, Cambridge, Massachusetts.

Zhivov, A. 1993. Theory and practice of air distribution with inclined jets, American Society of Heating, Refrigerating and Air Conditioning Engineers (ASHRAE) Transactions., 99(1):1152-1159.

# VITA


Name: Kavita Gangisetti

Address: c/o David Claridge,

Texas A&M University

Department of Mechanical Engineering

3123 TAMU

College Station, TX 77843-3123

Email Address: gkavita.14@gmail.com


Education: B.Tech., Mechanical Engineering, Jawaharlal Nehru Technological University, Hyderabad, India, 2004

M.S., Mechanical Engineering, Texas A&M University, 2010