COMPUTATIONAL FLUID DYNAMICS INVESTIGATION OF AIR VELOCITY AND TEMPERATURE DISTRIBUTION IN A ROOM EQUIPPED WITH ACTIVE CHILLED BEAM AIR-CONDITIONING

By

ABHIJYOTH REDDY VEMPATI

A THESIS PRESENTED TO THE GRADUATE SCHOOL
OF THE UNIVERSITY OF FLORIDA IN PARTIAL FULFILLMENT
OF THE REQUIREMENTS FOR THE DEGREE OF
MASTER OF SCIENCE

UNIVERSITY OF FLORIDA

2011

To my parents and my sister

ACKNOWLEDGEMENTS

First and foremost, I would like to thank my parents, Mr.Obula Reddy Vempati and Mrs. Lalita Vempati and my sister M/s Deepti Vempati for their emotional and financial support, without which I would not have been a part of the rich and competitive graduate program offered by the Department of Mechanical and Aerospace Engineering at the University of Florida.

It gives me great pleasure to place on record my deep sense of gratitude for my committee chair Dr. H. A. (Skip) Ingley, for his inspiring and valuable guidance and untiring interest given at every stage of this work right up to the preparation of this dissertation. I wish to thank Dr. S. A. Sherif for his constant encouragement during the course of the project and for agreeing to be a member in my committee My special thanks are due to Dr.Subrata Roy for his invaluable suggestions and his lab members: James, Navya Mastanaiah and Jignesh Soni for guiding me in the right direction whenever I faced problems regarding numerical simulations and CFD techniques.

My heartfelt thanks are due to Dr. Chin-Cheng (James) Wang and doctoral candidate Tae Sook Lee for graciously allocating me their valuable time to answer my queries. It was because of some interesting discussions with them, this research was possible. I would also like to thank Laurence Goodall(P.E) from Affiliated Engineers Inc, Gainesville Branch for teaching me the concept of chilled beam air-conditioning and providing me with the necessary data to carry out the numerical simulation.

Finally, I take this opportunity to thank my beloved friends Uday Kiran Mahakali, Anand Ankala, Bhageerath Bogi, Deepthi Thanigundala to name a few , who have always been there for me.

TABLE OF CONTENTS

page

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamics
HVAC	Heating Ventilation and Air-Conditioning
Btuh	British Thermal Unit/hour
SIMPLE	Semi-Implicit Method for Pressure Linked Equations
RNG	Re-Normalization Group
FEM	Finite Element Methods
SST	Shear Stress Transport
DNS	Direct Numerical Simulation
LES	Large Eddy Simulation
RANS	Reynolds-Averaged Navier Stokes
Re	Reynolds Number
k	Turbulent Kinetic Energy
ε	Turbulent Dissipation Rate
ω	Turbulent Specific dissipation rate
μ	Viscosity
k-ε	k-epsilon
k-ω	k-omega
V1	Local velocity at the top of the occupied zone at a distance of two inches from the wall
V2	Local velocity at the top of the occupied zone directly below the point of collision of opposing air streams
A	Centerline distance between two active beams with opposing blows
X	Distance between active beam centerline and an adjacent wall
H	Mounting height of active chilled beam

Abstract of Thesis Presented to the Graduate School
of the University of Florida in Partial Fulfillment of the
Requirements for the Degree of Master of Science

COMPUTATIONAL FLUID DYNAMICS INVESTIGATION OF AIR VELOCITY AND TEMPERATURE DISTRIBUTION IN A ROOM EQUIPPED WITH ACTIVE CHILLED BEAM AIR-CONDITIONING

By

Abhijyoth Reddy Vempati

May 2011

Chair: H.A. (Skip) Ingley
Major: Mechanical Engineering

Chilled beam air-conditioning technology evolved in Europe in the late 1970s, but its presence in the United States heating ventilation and air conditioning market has not been felt until recently. These systems are supposedly more energy efficient and are claimed to be reducing electricity usage by about 20-30% because of reduced fan power requirement. The unit makes use of a “nozzle concept” which uses the high velocity of primary air coming from nozzles inside the beam unit to create a low pressure region which induces part of the room air into the air-conditioning unit. The high speed primary air jet induces about two-thirds of the total air requirement from the room itself. This induced air is cooled and then mixed with primary which is sent back for air conditioning. Because of the reduced fan power requirement to pump the air, the chilled beam unit turns out to be energy efficient. Their added advantages are less noisy operation, reduced CO2 emissions, easier customization and high comfort levels.

The objective of the present research was to investigate the velocity and temperature distribution inside a room fitted with two active chilled beams (TROX 612B-HC active chilled beam unit) by use of numerical simulations. For the study purpose, a

test room with the dimensions 40ft×10ft×10ft (l×w×h) was considered. The room environment was simulated by creating and preprocessing the model in GAMBIT 2.4.6 and using FLUENT 6.3.26 for solving, analyzing and post processing the results. The study was focused on summer air-conditioning. Key input parameters such as the type of chilled beam used, amount of primary air supplied, primary air temperature, nozzle type, room conditions, water temperature etc were taken from data sheets obtained from Affiliated Engineers Inc, Gainesville Branch.

Numerical results have been validated by velocity measurements obtained from TROX DID 612B-HC active chilled beam selection program. The results obtained from the numerical simulations agreed well with the data available from the TROX selection program. In the end, a two dimensional simulation of a room with a multi-cone diffuser was performed for comparison of airflow distribution between an active chilled beam and a multi-cone diffuser. It was found out that with the active chilled beams, there was an almost uniform temperature distribution inside the room and quicker response to changes in heat load.

Though the numerical simulation provides considerable insight into the theoretical indoor airflow distribution, the underlying assumptions that went into the modeling are questionable. Experimental studies will be necessary to develop a more accurate model.

CHAPTER 1
INTRODUCTION

1.1 Chilled Beam Air Conditioning

An air-conditioner is a mechanical appliance which supplies and maintains desirable internal atmospheric conditions for human comfort, irrespective of external conditions. It is a system which effectively controls the temperature, humidity, purity and motion of air. It typically consists of cooling and dehumidifying processes for summer air-conditioning or heating and humidification process for winter air-conditioning. A system which performs the task of heating, ventilation and air-conditioning may be called an HVAC system.

A chilled beam forms a part of an HVAC system which uses chilled water flow through a hollow beam (heat exchanger) to remove heat from a room .It brings the chilled water closer to the occupied space than central air handlers and limits the utilization of fan power which ultimately leads to energy savings and cost cutting. They are capable of handling only sensible (dry) loads and hence must be connected to a source of primary dry air to provide dehumidification. Fresh air must be supplied with the primary air for ventilation purposes. The water flowing in the beam unit needs to be heated/cooled outside of the conditioned space and care must be taken to maintain the water temperature higher than the room dew point temperature so as to avoid condensation on the coil. A rule of thumb is to keep the water flowing through the chilled beam at a temperature approximately one degree Fahrenheit above the room dew point temperature. Hence the water is usually supplied at 55-59°F.This results in considerable energy savings and gives scope for geothermal cooling i.e. use of water reservoirs for cooling. [2]

Advantages of Chilled Beam Technology are reported to include:

a) Reduction in carbon emissions by about 15%

b) Sustainability: chilled beams can utilize chilled water supplied at 57°F and returned at 62°F .Hence for a fair amount of the year, ground water; evaporative cooled water etc. can be utilized eliminating the need for using mechanical refrigeration for further cooling of the water.

c) No need for suspended ceilings. Chilled beams can be directly fitted into a soffit and left exposed.

d) Lower noise levels

e) Mechanical system and duct reductions

f) Reduced or eliminated reheat requirement

g) Pump energy instead of fan energy

h) Can fit into tight space

Disadvantages of Chilled Beam Technology include:

a) Difficult to handle rooms with high heat loads without supplemental air
b) Condensation can be an issue if not installed properly
c) High first cost ($80/$m^2$)

1.2 Objectives of Present Study

The present investigation aims at simulating an air-conditioned room fitted with two Active Chilled Beams (TROX DID 612B-HC). The velocity and temperature distribution inside the room will then be investigated to rate the performance of chilled beam air-conditioning in terms of occupant comfort. The computational studies are performed on a 40ft×10ft×10ft room (12.192m×3.048m×3.048m) (l×w×h) by creating and pre-processing the room model in GAMBIT 2.4.6 and analyzing and post-processing the results in FLUENT 6.3.26, a commercial CFD code.

1.3 Proposed Investigation and Scope of Research

Both two-dimensional and three –dimensional simulations of the room with active chilled beam are performed. The airflow velocity and temperature distribution inside the room is of interest in the present research. The velocities and temperatures at certain locations inside the room will be evaluated to compare the results of the numerical simulation with those obtained from a TROX Active chilled beam selection program. To reduce the computational intensity and complexity of the problem, a simplified geometrical model of the chilled beam will be considered. The suitability of various turbulence models will also be investigated in terms of achieving well converged solutions for the given problem-set-up. A two dimensional simulation of a room fitted with a multi-cone diffuser will also be performed in the end for comparison with the results obtained with a chilled beam unit .This will give a qualitative idea about the performance of chilled beam air-conditioning compared to conventional diffusers.

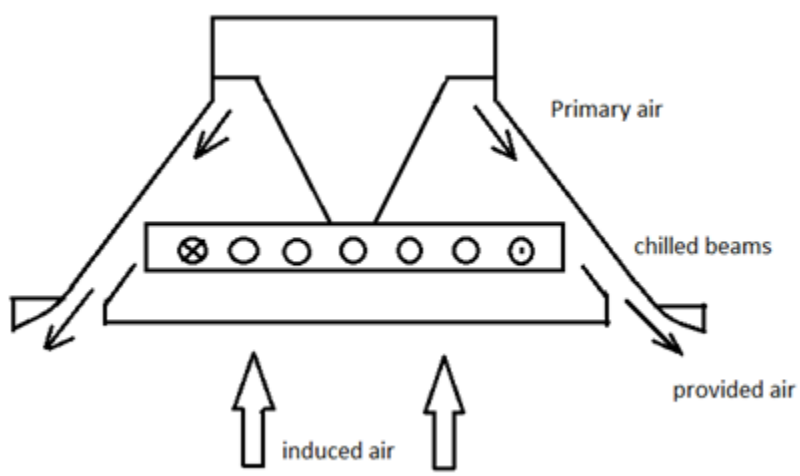


Figure 1-1. Active chilled beam (adapted from: CPD module [1])

CHAPTER 2
LITERATURE REVIEW

2.1 History of Chilled Beams

Chilled beams appeared in the European market in the late 1970s.They combine air supply with cooling or heating. The main principle of the active chilled beams is induction of room air caused by the high velocity of primary supply air. Inducted room air is drawn into the beam through a water-air heat exchanger where it is cooled or heated and subsequently mixed with the fresh air and supplied to the room through slot diffusers. The air flows from the beam over the ceiling, cascading down into the room. In case of parallel positioning of the beams the jets can meet in the middle, lose momentum and fall down to the occupied zone.

2.2 Types of Chilled Beams

Chilled beam technology uses closed circuit water based systems to facilitate heat transfer. Water flows through the cooling system absorbing and removing heat from the occupied space. Instead of radiation, chilled beams rely on convection heat transfer to deliver cooling. [1]The chilled beams can be broadly classified into four categories depending on the working principle as follows:

a) Chilled ceilings
b) Passive chilled Beams
c) Active chilled Beams
d) Multi-Service Chilled Beams

Chilled ceilings

Also referred to as the original chilled beam system; the chilled ceilings work on the principle of radiation. They exchange heat with the occupied space by radiation instead of introducing cold air. These systems resemble a standard suspended metal

ceiling system and are constructed from a copper cooling element bonded to the rear of a metal ceiling tile. Alternatively, these systems can also be used for heating purposes.

Advantages:

a) The most energy efficient of all the chilled beam systems

Disadvantages:

a) Lack of control of the humidity inside the room and hence need to be coupled with a forced air system to tackle latent heat loads

b) Very low sensible cooling capacity of 75.7 $\frac{W}{m2}$ (24 BTUH/square foot) of linear beam

c) Do not provide fresh air without an auxiliary outdoor air system.

Passive chilled Beams

Also referred to as the second generation of chilled beams, these were the first to use a pipe surrounded by fins in order to form a radiator system and are often used with under-floor air distribution systems. Rather than radiation, these rely on the phenomenon of natural convection to provide cooling. As the room air gets hot, it rises through the chilled beam where it is cooled and falls back to the occupied space. Thus its efficiency increases as the room temperature increases (refer to Figure 2-1).

Advantages:

a) Increased cooling and heating capacity
b) Extremely effective in taking care of solar heat load and sensible heat load.
c) Moderate sensible cooling capacity of 384.6 $\frac{W}{m}$ (400 BTUH/ft.) of linear beam

Disadvantages:

a) Lack of control of the humidity inside the room and hence need to be coupled with a forced air system to tackle latent heat loads

b) Do not provide ventilation air

Active chilled Beams

Also known as high induction diffusers, these entered the HVAC industry in the early 1990s and thus represent the third generation of chilled beams. These work on the principle of forced air induction and hence can provide both heating and cooling. All the HVAC functions are taken care of by a single unit .The ventilation air is ducted through the chilled beam as part of the primary air. It induces air movement over the coil, mixes it with the conditioned air and sends it back to the occupied space (refer to Figure 2-2).

Advantages:

a) Takes care of both sensible and latent heat loads by providing ventilation air

b) Very high cooling capacity of 576.911 $\frac{W}{m}$ (600 BTUH/linear foot) of beam

Multi-Service Chilled Beams

Referred to as the fourth generation of chilled beam systems, these are new to the US market. An active or passive chilled beam system can be made a multi-service beam unit. These combine building operations to a single unit containing lighting fixtures, sprinkler systems, smoke detectors, security sensors etc.

Advantages:

a) Decrease in building construction time by about 25% due to decrease in installation time

b) Reduced ceiling space requirement. In some cases for a 9’6’’ ceiling height, five floors were built in the same space occupied by a standard four story building

Disadvantages

a) Suitable for offices, labs, schools, hotels etc but not for residential areas because of lack of humidity control in case of multi-service beams fitted with passive chilled beams

b) If a window is left open, condensation can occur on the coil

2.3 Numerical and Experimental Studies of Indoor Air-Conditioning

The history of using CFD as a tool to analyze indoor airflows dates back to 1974 with Neilsen being the pioneer in initiating such studies. Jelena and Qingyan [2], ASHRAE Members, came up with a manual illustrating the various steps involved in the verification, validation and reporting of CFD simulations of indoor air flows by citing references of numerous papers published in Roomvent (2000).

The following can be deduced from their manual:

a) Use a steady state simulation and start with a simpler turbulence model such as standard k-ε model. The low Reynolds number turbulence models took longer computing time

b) Use of first-order differencing scheme and SIMPLE algorithms (for pressure-velocity compounding) seem to be more stable

c) Zero pressure and zero gradient for all other flow parameters were used as boundary conditions for the exhaust from the space

d) For a complicated analysis, grid independent results are very difficult to obtain. Hence there is a tradeoff-between the grid resolution and computational intensity so as to attain sensible physical results

e) Uncertainty and error in the measurement of turbulent intensity is the largest in both the CFD model and experimental methods. It is difficult to judge which of them is accurate

Zhu, Li, Yuan [3] researched indoor airflow by cold air distribution systems. They used Fluent to perform numerical simulations of indoor cold air distribution under different conditions and compared their results with an experimental setup. They threw light on the possible mathematical model and boundary conditions that are appropriate for simulations of indoor airflow. They made the following assumptions:

a) Incompressible, invariable, steady state flow which coincides with Boussinesq assumption and finite-volume approach

b) Velocity inlet boundary condition for the diffuser inlets

c) Negligible air leakage throughout the conditioned space

d) Negligible solar radiation and internal heat sources

e) High Reynolds number 2-equation k-ε turbulence model combined with wall function

Their experimental set-up involved a room of dimensions 23.6ft×18.3ft×10.5ft (7.2m×5.6m×3.2m).They made use of a thermo-static multi-anemometer to measure the temperature and velocity of air in the room. Their experimental results matched very well with the numerical simulations carried out in FLUENT, validating the appropriate way of setting up indoor airflow problems in a commercial CFD code like FLUENT.

Juan,Daniel,Marcelo,Benoit [4] researched airflow modeling and comfort levels in a computer room .They used GAMBIT for pre-processing the model, FLUENT 6.0.2 to carryout numerical simulations and Vu for post-processing the results. They considered species transport as well as radiation conversation equations along with mass, momentum, energy and turbulence equations .They validated their numerical results with a MINIBAT experimental set-up by considering a k-ε “realizable” turbulence model with a two layer approach near the walls. They made the following assumptions:

a) Segregated (pressure based) solver, SIMPLE pressure- velocity coupling and finite-volume approach

b) Renormalization group (RNG) k-ε turbulence model to treat turbulence near the air-supply

c) Discrete Ordinates (DO) radiation model and H20 was used in species transport equations

d) Modeled computers and human beings (through thermal mannequins)

e) Room boundaries were modeled as adiabatic walls and were considered diffuse with ε= 0.9 -1.0

f) Velocity inlet boundary conditions for diffuser inlets

Through numerical simulations, they estimated four indoor air-quality parameters - mean age of air, mean radiant temperature, predicted mean vote, predicted percentage of dissatisfied. Their investigation revealed that boundary conditions and inclusion of real-world geometries like diffuser inlets have a major impact on fluid-flow.

Awbi [[5] investigated the application of CFD in room ventilation by using a computer program based on finite-difference approach to solve 2-D AND 3-D ventilation problems.

Arsen, Boryana, Lyuben, Viktor and Risto [6] studied the impact of airflow interactions on occupants' thermal comfort in a full-scale test room 17.7ft×13.7ft×8.2ft (5.4m×4.2m×2.5m) fitted with four active chilled beams. They researched the impact of primary air flow rate and heat load strength on the thermal environment created in the occupied space. They used thermal mannequins, artificial windows, ceiling lights as heat sources inside the room. The air speed and temperature measurements were used to calculate air diffusion performance index (ADPI) and draught rating index (DR).They found out that the supplied air flow rate has a significant impact on DR. They concluded that:

a) Substantial changes of local thermal conditions at different workplaces occurred in the room when heat load and supply flow rate were changed

b) Higher heat load and higher supplied flow rates increase the risk of draught discomfort in the occupied zone

Ristom, Pekka, Hannu and Alex [7] researched the impact of heat load location and strength on air flow patterns in a mock-up office 11.8ft×10.8f×10.8ft (3.6m×3.6m×3.3m) fitted with two passive chilled beams, three light fittings and a swirl diffuser. They used human dummies and computers as heat sources inside the room. Their findings are summarized as follows:

a) Negligible buoyancy force. The plumes from the heat loads were not powerful enough to fight the downward flow from the beam. Instead the heated air was counter-acted by the downward cold flow from the beam .The human dummy only acted as a flow obstacle. Hence the shape of the occupant simulator is important

b) The point of occurrence of the maximum velocity in the occupied zone depends on the strength of the heat source and its distribution in the room

c) Layout of internal equipment had a minor impact on air distribution

d) Full-scale studies and CFD simulations are important for complex interactions of flows

Cammarata and Petrone [8] studied the thermodynamic and fluid dynamic performance of an active chilled beam for indoor air-conditioning by investigating 2D and 3D models in an FEM based software, COSMOL Multiphysics. They evaluated the maximum airflow velocity and horizontal and vertical temperature gradient. An incompressible, steady state fluid flow with a two equation k-ε turbulence model was used. They concluded that there is an almost constant velocity distribution in the occupied portion of the conditioned space.

Jan, Viktor, Risto and Arsen [9] came up with practical guidelines to minimize draught discomfort in rooms with four active chilled beams. They focused on factors affecting thermal comfort. They conducted experiments in a full-scale test room with the dimensions 17.7ft×13.7ft×8.2ft (5.4m×4.2m×2.5m). The chilled beams had a velocity control device to control the induced airflow rate. They experimented with different strengths of heat load and locations, different layout of chilled beams, heat generated by occupants, primary and induced airflow rate. They came up with the following guidelines:

a) The room height with chilled beams should be no more than11.48ft (3.5m) which could otherwise lead the thermal forces to cause draught risks in the occupied zone, when there is a high cooling load

b) Chilled beams installed in a lengthwise position can cause unnecessary draught risks

c) Higher airflow rate causes higher mean velocity leading to greater draught risk

d) Chilled beams should be installed perpendicular to window façades to avoid flow interaction between heated surfaces and flows from chilled beam

Posner, Buchanan, Rankin [10] studied the measurement and prediction of indoor airflow in a model room. They compared 3-dimensional CFD simulations carried out in FLUENT with experimental results obtained from a one-tenth sub-scale isothermal model room. They used lased Doppler anemometry (LDA) and particle image velocimetry (PIV) to measure temperature and velocity fields in the sub-scaled room made of anodized aluminum of dimensions 30ft×15ft×10ft (9.14m×4.57m×3.05m) having four plain glass windows. They focused more on the impact of obstructions in the airflow distribution. For numerical simulation, their problem set-up involved the following:

a) Fluid flow was modeled using three different configurations as: laminar, standard k-ε turbulence model and RNG k-ε turbulence model. Default values for constants were used

b) A power law scheme of descritization was used and transient simulation was carried out to achieve convergence

c) Buoyancy effects were neglected. There were no heat sources and no heating/cooling by ventilated air

They concluded that laminar and RNG models compare better with experimental measurements than the standard turbulence model. The standard model tends to smooth out steep gradients in the flow field.

Stamou and Katsiris [12] studied the suitability of SST k-ω model for numerical simulation of indoor air-conditioning and compared it with laminar and k-ε turbulence models, by use of CFX, a commercial CFD code. Already available experimental results on displacement ventilation were used to rate the suitability of all these models. They

showed that an SST model with a suitable grid shows the best agreement. The following can be deduced from their paper:

a) Of the three main types of CFD methods i.e. Direct Numerical Simulation (DNS), Large Eddy Simulation (LES) and Reynolds Averaged Navier–Stokes (RANS), the RANS method was the least computationally intensive

b) The standard k-ε model was valid only for fully-turbulent flows and often failed to capture low-velocity (especially near-wall) regions. It over-estimated the turbulent diffusion. Hence, one must resort to using low Re models such as the RNG k-ε model. Many researchers claim this model to be more stable than any other k-ε model in predicting indoor-airflow simulation

c) The low Re number version of the k-ω model was claimed to be more numerically stable and it gave faster converged solutions than the corresponding k-ε models. However, its applicability to an indoor environment was not suitable due to its strong sensitivity to free-stream conditions

d) The k- ω SST model combined the k-ε and k- ω model using a blending function and used the k- ω model for near wall regions and k-ε model for the rest of the flow

e) The Reynolds Stress model (RSM) enabled detection of the presence and the localization of separated flow and correctly predicted airflow patterns but storage and execution times were computationally intensive for 3D indoor flows

In the numerical simulation, they made the following assumptions for problem set-up:

a) A fully-implicit coupled solver (density based) was used

b) The diffusers were modeled as velocity inlets

c) At the outlet, the average static pressure was set to atmospheric and vertical gradients of all other variables were set to zero

d) They modeled thermal mannequins, PC simulator and lighting simulator to simulate heat loads inside the room

They concluded that there are vertical temperature gradients and negligible horizontal temperature gradients .A steady state solution was not obtained for the standard k-ε, RNG k-ε and laminar flow models. Their calculations showed that all the tested turbulent models predicted satisfactorily the main qualitative features of the flow and the layered type of temperature fields. Thus, all these models can be used for

practical purposes. Calculations with the SST k– ω based model showed the best agreement with measurements and the laminar model the worst.

Kotani, Yamanaka, Momoi [13] carried out 3-D CFD simulations of airflow in a room with a multi-cone ceiling diffuser and validated the results with the measurements obtained from an experimental set-up. They used a constant temperature hot film anemometer to measure the velocities, turbulent kinetic energies and length scales around the diffuser and ultra-sonic anemometer to investigate airflow velocities and turbulent kinetic energies in the room. They concluded that a CFD simulation using the measured values as the supply boundary condition in the box method can predict the airflow pattern inside the room, except for the decay of axial jet velocity in the heating condition. A summary of their problem set up in, STREAM, Version 4, a commercial CFD code, is given below:

a) Standard k-ε turbulence model and measured velocities, turbulent parameters were used as boundary conditions to perform the experiments

b) A third-order QUICK descritization scheme was used

c) The geometry of the multi-cone diffuser was approximated using the “box method” to capture both radial and axial flows

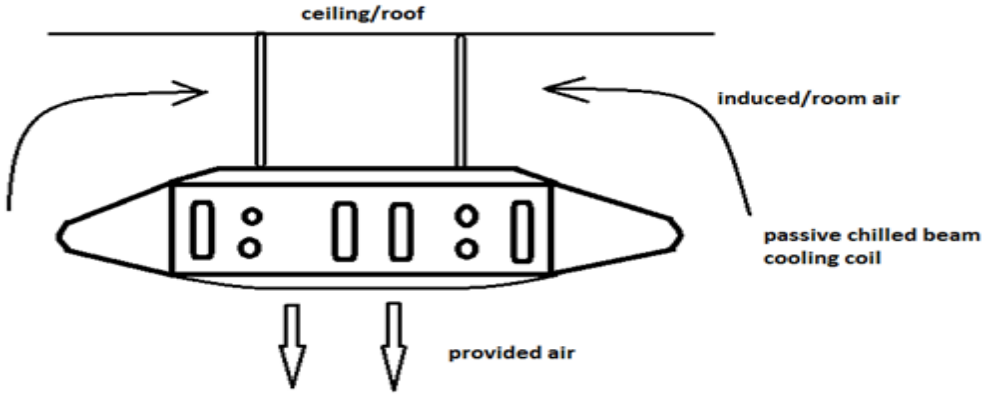


Figure 2-1. Passive chilled beam (adapted from: CPD module [1])

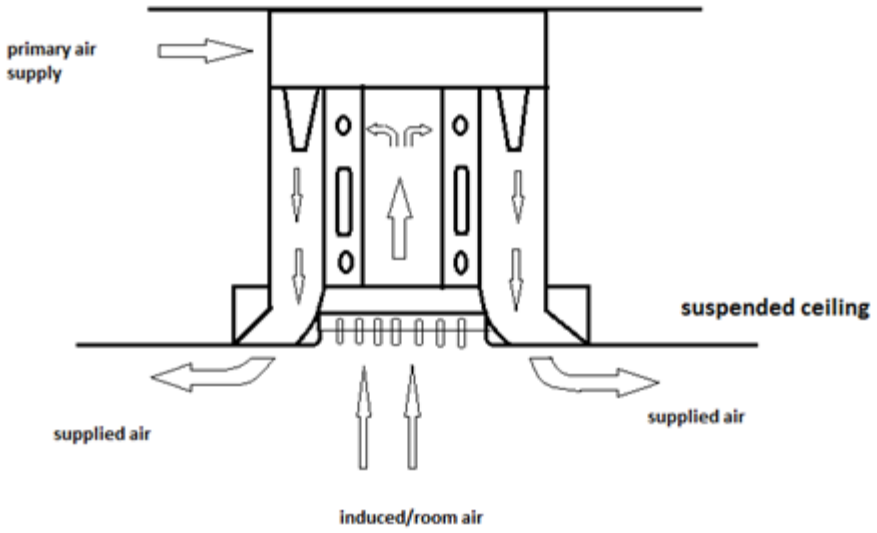


Figure 2-2. Active chilled beam (adapted from: CPD module [1])

CHAPTER 3
PROBLEM STATEMENT

3.1 Data Acquisition and Physical Calculations

Key input parameters such as the type of chilled beam used, amount of primary air supplied, primary air temperature, sensible cooling capacities, nozzle type, room conditions, water temperature, etc. were taken from data sheets obtained from Affiliated Engineers Inc, Gainesville Branch. The room geometry was modeled similar to the model room considered in the chilled beam design guide. The same figure was used in the TROX DID 612B-HC Active Chilled Beam selection program. For details, refer to the figure 3-1.The parameters X and A/2 in the figure are both 10ft (3.048m) and the room height is also 10ft (3.048m). The width of the room is considered to be 10ft (3.048m). Hence, the room dimensions 40ft× 10ft×10ft (12.192m× 3.048m×3.048m) i.e. 400 sq ft (37.16 sqm). Two Trox DID 612B-HC active chilled beams were modeled .The beam units are 6ft long with g-type nozzles. The primary airflow rate is 125 cfm. In order to find the total airflow rate, the amount of induced air is calculated using the following relations:

$$\frac{qcoil}{1.1 * Q\ induced} + \frac{qcoil}{500 * gpm} = Troom - T\ enteringchw$$

Where,

qcoil =sensible cooling capacity of the coil=4205 Btuh.

GPM=chilled water flow rate= 2 GPM.

T room=room temperature= 74°F (23.3°C).

T entering chw= chilled water supply temperature=56°F (13.3°C).

Using these values, the amount of induced air was found out to be

Q induced=277.1 cfm. Thus the total airflow becomes Q total=Q primary + Q induced=402.1 cfm (0 .19 m^3/s). This 400 cfm of air is being supplied by each beam unit corresponding to 2cfm/sq. ft. which is typical for laboratory applications.

3.2 Identifying the Physical Domain and Input Parameters Required

Both 2-D and 3-D simulations were performed. The physical domain is the area within which the fluid flow is to be studied, which is the room in the present study. For 2-D numerical simulations, an empty room with the dimensions 40ft× 10ft (12.192m×3.048m) is modeled. The present research is focused on the airflow distribution in an empty room. The room is fitted with two active chilled beams and four light simulators. For the 3-D simulation, the room dimensions are 40ft× 10ft×10ft (12.192m×3.048m×3.048m). Lights are each 4ft×1ft (1.2192m × 0.3048m) (l×w). The beam unit is 6ft long and for the rest of the dimensions are as shown in the Figure 3-2. Input parameters required: Total airflow through each beam unit: 402 cfm (0.19 m^3/s).

The heat emitted by each tube light is 33$\frac{W}{m^2}$.

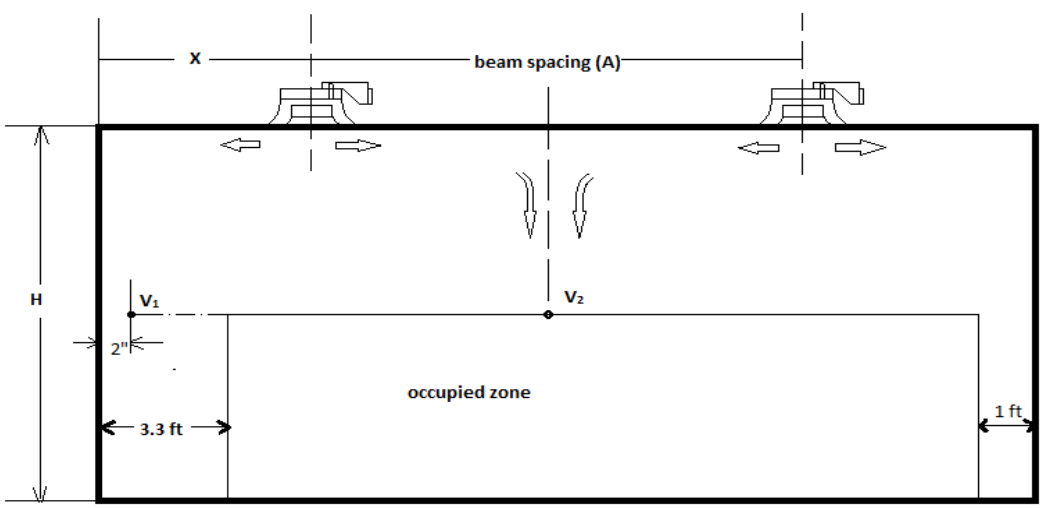


Figure 3-1. Room air velocity and temperatures parameters used in the design (adapted from: chilled beam design guide [14])

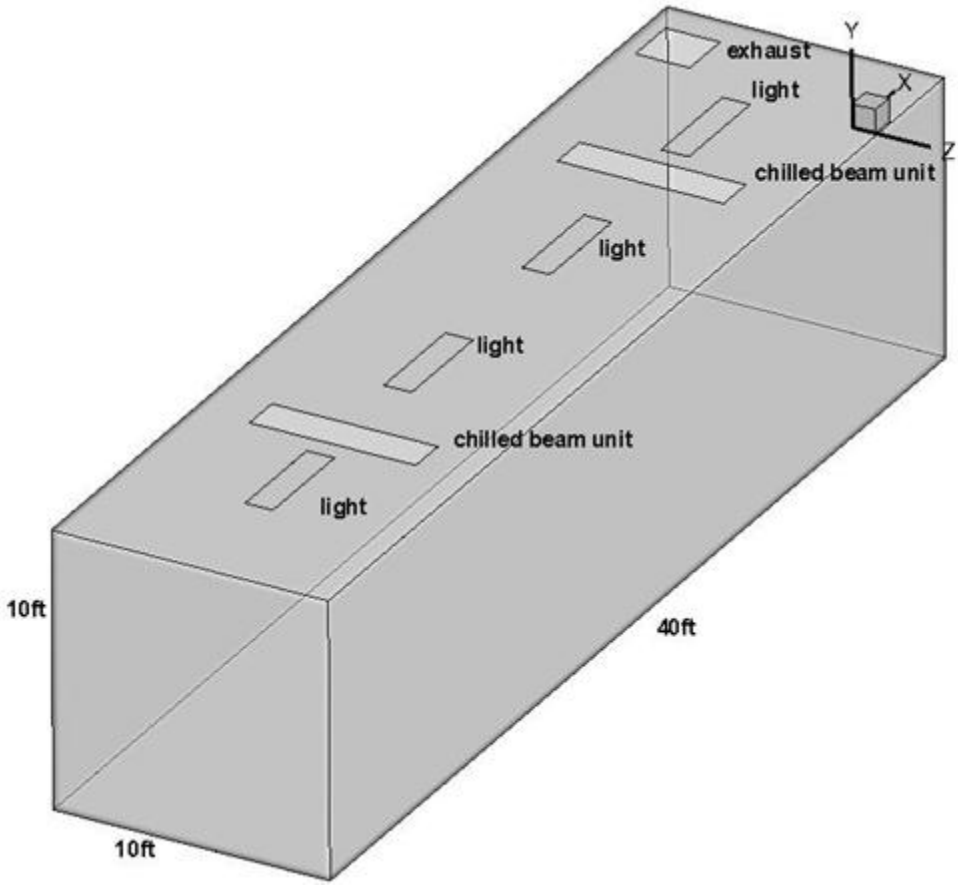


Figure 3-2. Schematic showing the location of lights and the air-conditioning unit

CHAPTER 4
CFD SIMULATION OF TEST ROOM

There are several commercial CFD software packages in the market. The University of Florida maintains a license for FLUENT 6.3.26 from Fluent Inc. Hence this software was employed for the present research. It has already been tried and tested in the field of indoor air-conditioning. The FLUENT software solves the 3-D Reynolds-Navier-Stokes equations for the mass-averaged velocity and the time-averaged pressure, energy and density. The software is an integrated and complex Navier-Stokes fluid flow prediction system, capable of diverse and complex multi-dimensional fluid flow problems. It uses a flexible, multi-block grid system, a graphical interface and several sophisticated modeling tools, making it suitable for indoor airflow simulation. The fluid flow solver, FLUENT provides solutions for incompressible / compressible, steady state / transient, laminar / turbulent single-phase fluid flow in complex geometries.

4.1 Steps Involved In the Simulation Process

The outline of the simulation process is summarized as follows:

a) Pre-processing
- Modeling the geometry and the flow domain
- Establishing the boundary and initial conditions
- Mesh generation

b) Solving
- Reading the mesh file and grid check
- Establishing the simulation strategy
- Establishing the input parameters and files
- Performing the simulation
- Monitoring the simulation for convergence

c) Post-processing
- Post-processing the simulation to get results graphs, plots, contour plots etc.(this will be dealt in the next chapter)

4.1.1 Pre-processing.

For modeling the geometry, GAMBIT 2.4.6 software is used. It is a general purpose pre-processor for CFD analysis which provides meshing capabilities wherein the model can be meshed and subsequently imported into FLUENT and solved. GAMBIT is designed for the creation of high quality computational meshes. Predefined grid topology templates are used to minimize grid setup time and optimize the mesh for the given application. GAMBIT enables the user to generate computational grids quickly through the automatic management of grid topology and grid attachment.

Modeling procedure. (approximation and simplification of geometry) In numerical simulations, approximations of the geometry and simplifications may be required in an analysis to ease the computational effort. Especially, for the case of indoor air simulations, it is very difficult to model the diffusers, nozzles, vents etc of the air-conditioning unit because these are much smaller compared to the room dimensions and also because of their complicated geometry. It increases the computational effort because of the increased number of nodes and meshed elements making the problem set-up complicated.

> Srebric and Chen said, “The user should not be afraid of making assumptions. Good assumptions can simplify the complex physical phenomena in the real world with negligible effect on the accuracy of the CFD prediction” [2]

Keeping this in mind, the inlets and the induction unit of the chilled beams were modeled as flat, plane openings through which the fluid flows at an angle set by the actual geometry of the chilled beam unit.

2-D modeling. The room dimensions are 40ft×10ft (12.192m×3.048m)**.**The chilled beam units and light simulators were modeled as edges along the ceiling of the room.

All the edges forming the room boundary were defined as a single face. This model was saved as .dbs file.

Mesh generation. The entire face is divided into innumerable small finite number of elements. This process is called meshing and the grid generated is called a mesh. Meshing gives us a scope to study the behavior of various parameters (such as pressure, velocity etc.) at each of these elements. The finer the mesh (more elements) the better is the scope for analysis since it gives us more number of points to study the behavior of parameters. In GAMBIT, the 2-D mesh elements can be of two types namely Triangular and Quadrilateral. Use of triangular element creates an unstructured mesh whereas; the use of a quadrilateral element creates a structured mesh. To create a mesh, either the element count (number of elements) or the element size can be specified. Gambit also provides us with *size functions* which allow us to control the size of mesh-element edges for the geometric edges and for faces or volumes that are meshed. Since an empty room is considered with no complex structures such as thermal mannequins or pc simulators, a uniform structured mesh of 0.02m was employed.

Establishing the boundary conditions. Once the mesh is generated, various edges of the grid are given names for easy understanding and for setting the appropriate boundary conditions while solving. The continuum type (fluid/solid) is also specified. Finally, this meshed model is *exported* as *.msh file* in a format that can be directly into FLUENT.

3-D modeling. The room dimensions are 40ft×10ft×10ft (12.192m×3.048m×3.048m). The chilled beam units and light simulators were modeled

as faces along the ceiling of the room. All the faces forming the room boundary are stitched to form a single connected volume. This model was saved as .dbs file.

Mesh generation. The entire volume is divided into innumerable small finite number of elements. This process is called meshing and the grid generated is called a mesh. In GAMBIT, the 3-D mesh elements can be of two types namely Tetrahedral (unstructured) and Hexagonal (structured). In the present model, a uniform structured mesh of 0.15m has been used for meshing the face.

Establishing the boundary conditions. Once the mesh is generated, various faces of the domain are given names for easy understanding and for setting the appropriate boundary conditions while solving. The continuum type (fluid/solid) is also specified. Finally, this meshed model is *exported* as *.msh file* in a format that can be read directly into FLUENT. A schematic of the boundary conditions used is shown in the figure 4-2.

4.1.2 Solving

Solving is an important phase in CFD analysis. The software used for solving in the present study is FLUENT6.3.26. The .msh file is read into FLUENT and a routine grid check is performed to detect the presence of any skewed cells. Skewness is the difference between the shape of the cell and the shape of an equilateral cell of an equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. These skewed cells can be weeded out either by use of smooth/swap grid option in FLUENT or by re-meshing the model in GAMBIT .Then, we may proceed to setting up the problem.

Establishing the simulation strategy. To perform the simulation, we must lay out some rules that affect the physics of the problem. The problem is then set-up based on

these under-lying assumptions. The present study is based on the following assumptions:

a) Incompressible fluid flow and finite-volume approach

b) Steady state –summer air conditioning simulation of the room environment

c) Negligible buoyancy effects

d) Negligible air leakage throughout the conditioned space

e) Negligible solar radiation and internal heat sources

f) Semi-Implicit Method for Pressure Linked Equations (SIMPLE pressure- velocity coupling)

g) Negligible radiation and species transport

h) Room boundaries were modeled as adiabatic walls

i) Two solvers are available in FLUENT namely:

j) Pressure based (Segregated) solver

k) Density based (Coupled) solver

The pressure based solver is used for low-speed incompressible flows while the density based solver is used for high speed compressible flows, where the velocity and pressure are strongly coupled (high pressures and high velocities). The indoor airflow simulation falls under the category of low-speed incompressible flow, which can be deduced from extensive literature survey .The air velocities at the inlets, also indicate the same. Thereby, in the present study a pressure based solver has been employed. In this method, governing equations are solved sequentially (i.e. segregated from one another). Because the governing equations are non-linear, several iterations of the solution loop must be performed before a converged solution is obtained. Once the grid is checked, the pressure based implicit solver is applied.

Viscous model. The following turbulence models are applicable to indoor airflow analysis:

a) k-epsilon

b) k-omega

The k-epsilon (k-ε) model is by far the most popular two-equation turbulence model for industrial applications. The model computes the Reynolds stresses by solving two transport equations: one for the turbulent kinetic energy and one for the rate of dissipation, ε, which represents the conversion of k into thermal internal energy. The turbulent viscosity is then calculated as a function of k and ε. It uses moderate computational resources and gives a solution of good accuracy.

Just like the k-ε models, the k- ω model computes the Reynolds stresses by solving two transport equations: one for the turbulent kinetic energy and one for the specific dissipation rate, ω, which can be seen as the ratio between ε and k. The turbulent viscosity is then calculated as a function of k and ω. The major difference in performance between the k-ε models and the k- ω models is found in the fact that the k-ε models are primarily valid for turbulent core flow (i.e., flow in regions relatively far from walls) whereas the k- ω models are created to be applicable through the boundary layer close to the wall. There exists one variant of the standard k-ω model in FLUENT: the SST (Shear-Stress Transport) k-ω model. This model combines the features of the standard k-ε and the standard k-ω model by using the former for the flow somewhat far away from walls and the latter when modeling the flow close to a wall. It uses slightly greater computational resources than k-epsilon but gives a solution of better accuracy as it also includes the boundary layer formation effects in the solution. Reynolds stress

model uses seven equations and require 50 % to 60 % greater computational resources than k-omega. It gives a solution of the highest accuracy among the solvers mentioned above. The k-epsilon solver was chosen for the analysis over k-omega as it captures the effects of swirl flows effectively.

FLUENT provides three k-epsilon models:

a) Standard

b) RNG(Renormalizing Group Theory)

c) Realizable

To make the most appropriate choice of model for our application, we need to understand the capabilities and limitations of the various options. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. Among these models, the k-epsilon Realizable has the latest additions which include new formulation for turbulent viscosity and a new transport equation for the dissipation rate. Hence, to treat turbulence near the air-supply, realizable k-ε turbulence model is employed with standard wall treatment option enabled. However, it will be interesting to see the performance of other turbulence models in terms of convergence time, solution prediction and higher order accuracies. Hence, the standard and RNG k-ε model simulations have also been performed.

Establishing the input parameters. (material properties, boundary conditions definition) The fluid inside the room is air with the default properties as defined in the FLUENT library of materials. The boundary conditions employed in this model can be grouped into three categories:

Mass-flow inlet: The inlets of the chilled beam unit have been defined as mass-flow inlets with a direction set by the actual geometry of the chilled beam. The air flow comes out through the slot diffusers at an angle of 32.47° with respect to normal with a discharge of 402 cfm through each beam unit. The air inlet temperature was taken to be 57°F (287K).

Pressure-outlet: Zero pressure and zero gradient for all other flow parameters were used as boundary conditions for the exhaust and the induction unit. The air leaving the room was assumed to be at room temperature i.e. 74°F (296K)

Wall: the light simulators and the room were defined as adiabatic walls. The room walls were set to a temperature of 80°F (300K). The lights were given a heat flux corresponding to a heat flow of 33 $\frac{W}{m^2}$

4.2 Setting the Solution Controls and Obtaining a Converged Solution

A summary of the options set in FLUENT is included in tables 4-1 through 4-5. The rest of the options were all set to default. With these settings and boundary conditions, the solution was initialized and iterated till convergence. Iteration consists of the following steps:

a) Fluid properties are updated, based on the current solution. And if the calculation has just begun, the fluid properties will be updated based on the initialized solution

b) Three momentum equations are solved in turn using current value of the pressure and face mass fluxes, in order to update the velocity field

c) The velocity obtained in first step may not satisfy the continuity equation locally. A Poisson-type equation for the pressure correction is derived from continuity equation and the linearized momentum equation. This pressure correction equation is then solved to obtain the necessary corrections to the pressure and velocity fields and the face mass fluxes such that continuity is satisfied

d) When interface coupling is to be included, the source terms in the appropriate continuous phase equations may be updated with a discrete phase trajectory calculation

e) A *check for convergence* of the equation set is made

Above steps re-occur until convergence criterion is achieved.

Convergence It is the point at which the solution no longer changes with successive iteration. Convergence criteria, along with reduction in residuals help in determining when the solution is complete. Convergence criteria are pre-set conditions on the residuals of continuity, momentum, energy, k and ε which indicate that a certain level of convergence has been achieved. Residuals are the small imbalances that are created during the course of the iterative solution algorithm. This imbalance in each cell is a small, non-zero value that, under normal circumstances, decreases as the solution progresses. If the residuals for all problem variables fall below the convergence criteria but are still declining, then the solution is still changing to a greater or lesser degree. A better indicator occurs when the residuals flatten in a traditional residual plot (of residual value vs. iteration). This point, sometimes referred to as convergence at the level of machine accuracy, takes time to reach and sometimes may be beyond what is needed. For this reason, the convergence is said to be achieved when all the residues fall below the order of a micro level ($10^{-6)}$ .Alternative tools such as reports of mass balances have also been employed. A mesh convergence study was performed to arrive at an optimal mesh size, such that the results do not change by an appreciable amount even after further reduction in mesh size. For details, refer to appendix-B.

Table 4-1. Solver settings for 2-D simulation

Feature	Status
Space	2D
Formulation	Implicit
Time	Steady
Energy equation	Enabled
Viscous	Realizable k-ε model, standard and RNG k-ε model.
Near-wall treatment	Standard wall functions
Viscous heating	Disabled

Table 4-2. Solver settings for 3-D simulation

Feature	Status
Space	3D
Formulation	Implicit
Time	Steady
Energy equation	Enabled
Viscous	Standard k-ε model.
Near-wall treatment	Standard wall functions
Viscous heating	Disabled

Table 4-3. Descritization scheme for 2-D simulation

Variable	Discretization scheme
Pressure	Standard
Momentum	Second Order Upwind
Turbulent kinetic energy	Second Order Upwind
Turbulent dissipation rate	Second Order Upwind
Energy	Second Order Upwind
Pressure –velocity coupling	Simple

Table 4-4. Descritization scheme for 3-D simulation

Variable	Discretization scheme
Pressure	Standard
Momentum	First Order Upwind
Turbulent kinetic energy	QUICK
Turbulent dissipation rate	QUICK
Energy	QUICK
Pressure –velocity coupling	Simple

Note: Because of limited computational resources, only first order discretization scheme was employed for solving momentum equation while third order QUICK scheme was used for the rest in the 3-D simulation.

Table 4-5. Under-relaxation factors for both 2-D and 3-D simulations

Variable	Relaxation factor
Pressure	0.3
Density	1
Body forces	1
Momentum	0.7
Turbulent kinetic energy	0.8
Turbulent dissipation rate	0.8
Turbulent viscosity	1
Energy	1

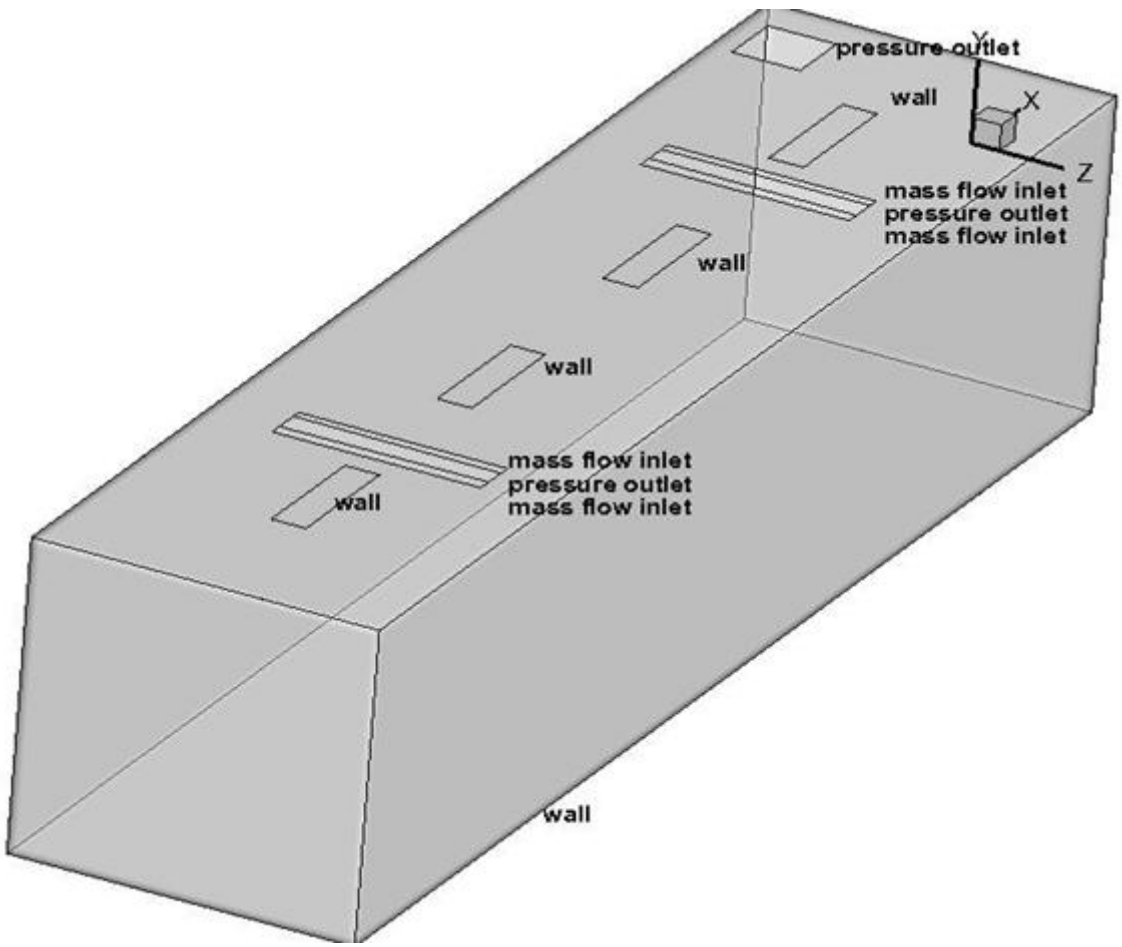


Figure 4-1. Schematic showing the boundary conditions employed; Info: co-ordinates x=0-40ft; y=0-10ft; z=0-10ft

CHAPTER 5
RESULTS AND OBSERVATIONS

Post processing. The data obtained from iterations carried out in FLUENT is analyzed in this chapter. Since the primary objective of this study was to focus on the airflow distribution inside the room, the vectors of air velocity and temperatures are plotted along with a few remarks and observations about the simulation.

5.1 Remarks and Observations

a) First order descritization scheme was used initially .It was noted that the RNG k-ε turbulence model took the longest time to give a converged solution(4338 iterations),while the standard and realizable models converged after 1100 and 1700 iterations respectively. Similar phenomenon was observed by Jelena et al [2]

b) The simulation was performed on an Intel core 2 Duo CPU T6500 6.1 GHz processor -4GB RAM desktop. It took almost about 30 hrs of computational time to reach a 3-D first-order converged solution.

c) The first order descritization schemes are often inaccurate in predicting the actual physics of the problem and can be used only as an initial solution. Hence second order schemes were employed for 2-D simulation whereas only first order scheme was used for 3-D simulation because of limited computational resources. Both standard and realizable turbulence model simulations using second order descritization scheme were made. The RNG model failed to provide a converged solution with the present mesh. The realizable model converged to a greater extent than compared to a standard model.

d) There is a considerable difference in the velocity and temperature distribution predicted by the standard and realizable models.

e) The realizable model depicts two uniform vortices in the left half of the room portion while the standard model depicts a single large vortex in that portion.

f) The plot of static temperature across the room shows that there is a horizontal gradient in temperature across the room with a realizable k-ε turbulence simulation while the standard k-ε turbulence simulation predicts an almost uniform temperature distribution inside the room.

g) The path lines predicted by a 2-D realizable model show a small vortex region forming between the inducer and the return which is absent in a standard model simulation.

5.2 Simulation Results

2-D Standard k-ε turbulence model simulation. The Figures 5-1 through 5-5 show the air velocity and temperature distribution inside a 2-D room with a standard k-ε model. The standard model gave quicker numerical results when compared to the rest of the models. A single large vortex in the left half of the room can be noticed while a stagnant region can be observed in the top right portion of the room near the inducer and the exhaust. A part of the stream goes through the inducer while the rest goes out through the exhaust/return. There is an almost uniform temperature distribution inside the room even though the colored visuals of temperature gradient may indicate otherwise. The temperature scale in the image shows that most of the room is at 288 K. As one moves to the right, the temperature is in the higher range of 288 K. This minute variation in temperature can only be detected through numerical simulations and may not be detected at all in experimental results. Hence, one might say that there is an almost uniform temperature distribution inside the room. The Figure 5-5 shows the plot of velocity magnitude of airflow inside the room with a standard k-ε model at a section 1.7 m from the bottom of the room. The velocity magnitude at locations 1 and 2 can be obtained from this plot which will later be used for validation of the results. The maximum velocity in the occupied zone was found out to be 96 fpm.

2-D Realizable k-ε turbulence model simulation. The Figures 5-6 through 5-7 show the air velocity and temperature distribution inside a 2-D room with a realizable k-ε model. Two uniform vortices in the left half of the room can be noticed (as against a single large vortex with a standard model) while a stagnant region can be observed in the top right portion of the room near the exhaust. The path lines indicate a small vortex region formation between the inducer and the exhaust. A horizontal gradient in

temperature (288-289 K) can be noticed. This gradient in temperature is again very small even though the colored visuals of temperature gradient may indicate otherwise. The temperature scale in the image shows that most of the room is at 288 K. As one moves to the right, the temperature is in the higher range of 288 K and 289K. This minute variation in temperature can only be detected through numerical simulations and may not be detected at all in experimental results. The Figure 5-10 shows the plot of velocity magnitude of airflow inside the room with a realizable k-ε model at a section 1.7 m from the bottom of the room. The velocity magnitude at locations 1 and 2 can be obtained from this plot which will later be used for validation of the results.

3-D Standard k-ε turbulence model simulation. The Figures 5-11 through 5-17 show the air velocity and temperature distribution of airflow inside a room with a standard k-ε model. The path lines of air velocity indicate that the flow comes out through the diffusers and leaves through the induction unit and the exhaust/return. The flow comes out the through the diffuser, hits the wall and cascades down along the wall. The flows coming from two opposing chilled beam nozzles meet at the mid-section of the room; they reinforce and cascade down to form a vortex of cold air which effectively cools the room. The air then moves to the rest of the room and part of it goes through the inducer where it is re-treated and sent back into the occupied space while the rest of it exits through the return/exhaust. The air velocity in the occupied zone is less than 60 fpm**.** An almost uniform temperature distribution can be noticed except for the ceiling where heat sources and diffusers are located.

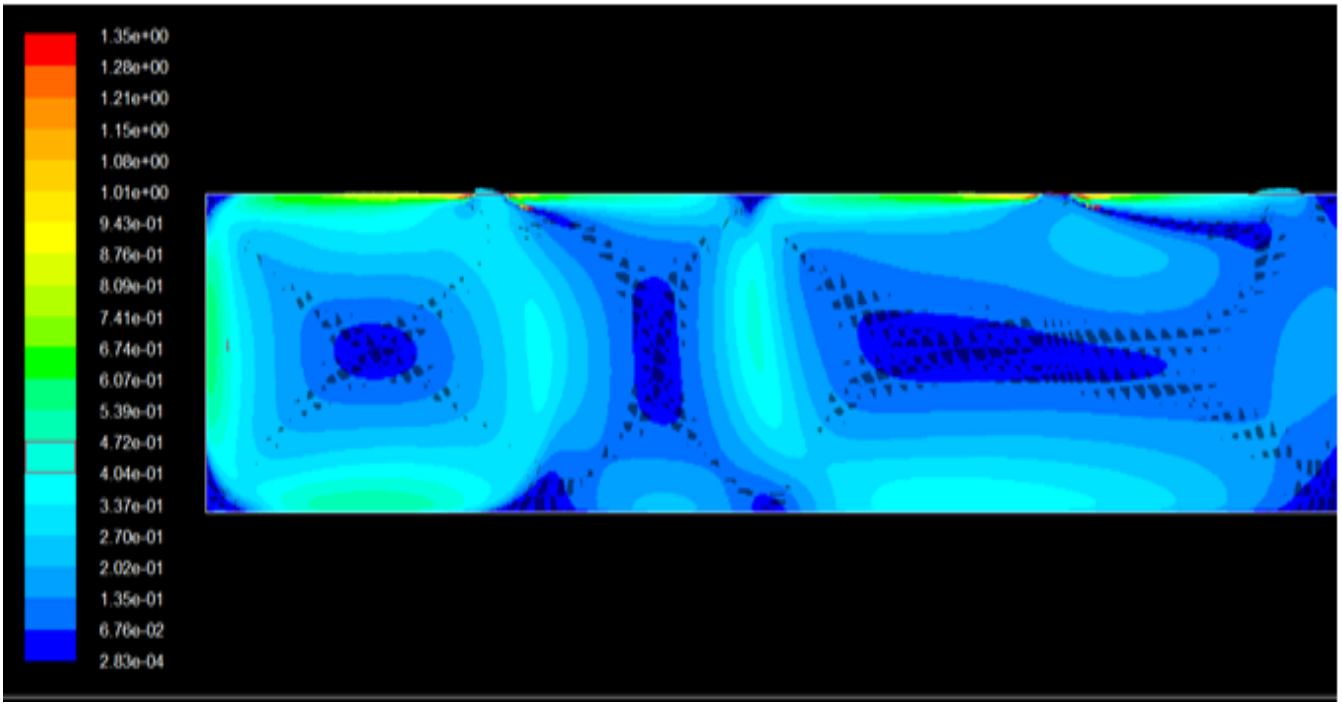


Figure 5-1. Velocity vectors of airflow inside the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); 2-D Standard k-ε Turbulence Model Simulation

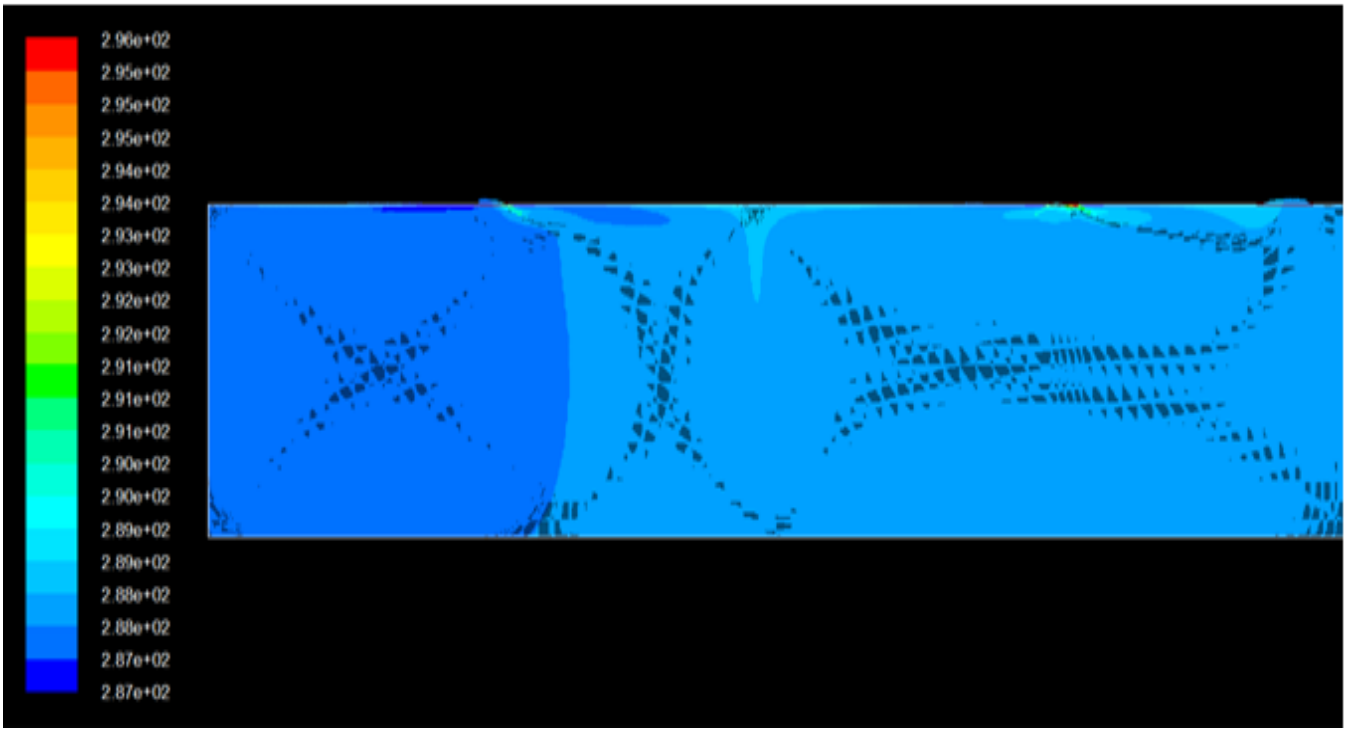


Figure 5-2. Temperature of airflow inside the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); 2-D Standard k-ε Turbulence Model Simulation

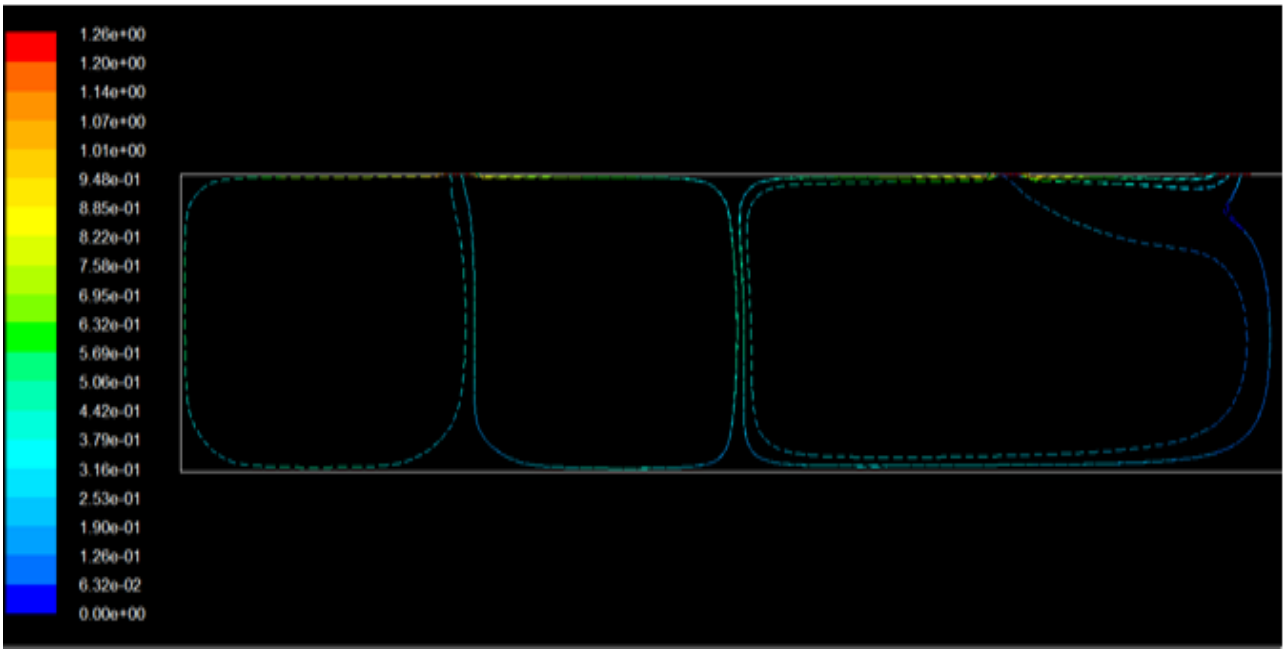


Figure 5-3. Path lines showing the velocity magnitude of airflow inside the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); 2-D Standard k-ε Turbulence Model Simulation

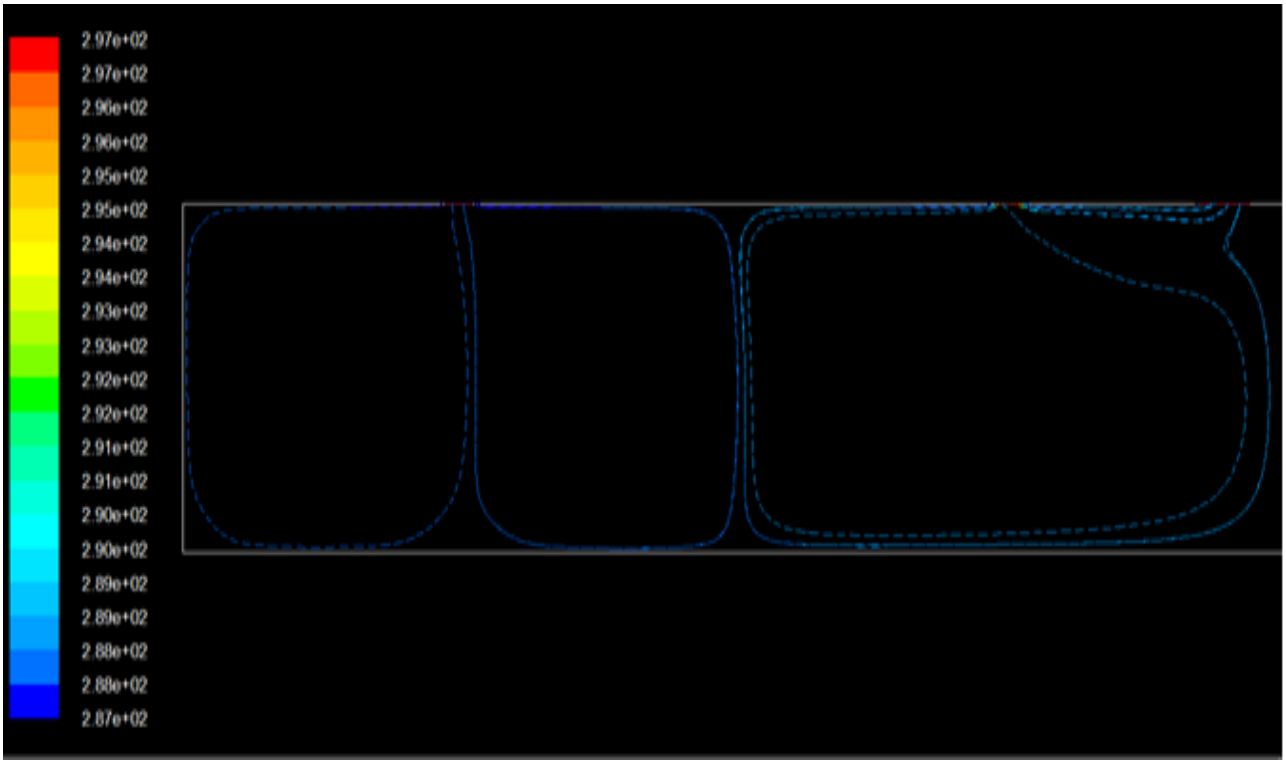


Figure 5-4. Path lines showing the temperature of airflow inside the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); 2-D Standard k-ε Turbulence Model Simulation

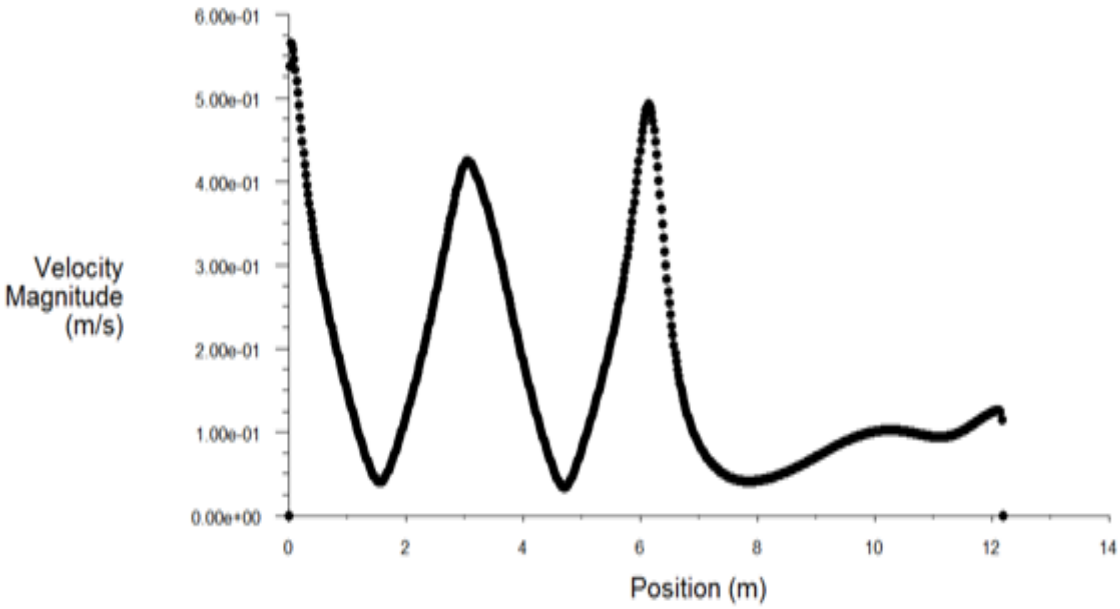


Figure 5-5. Plot of velocity magnitude (at a height of 1.7 m) across the room; Info: X-axis: length of the room (0-12.192m); Y-axis: velocity magnitude of airflow inside the room (0-0.6m/s); 2-D Standard k-ε Turbulence Model Simulation

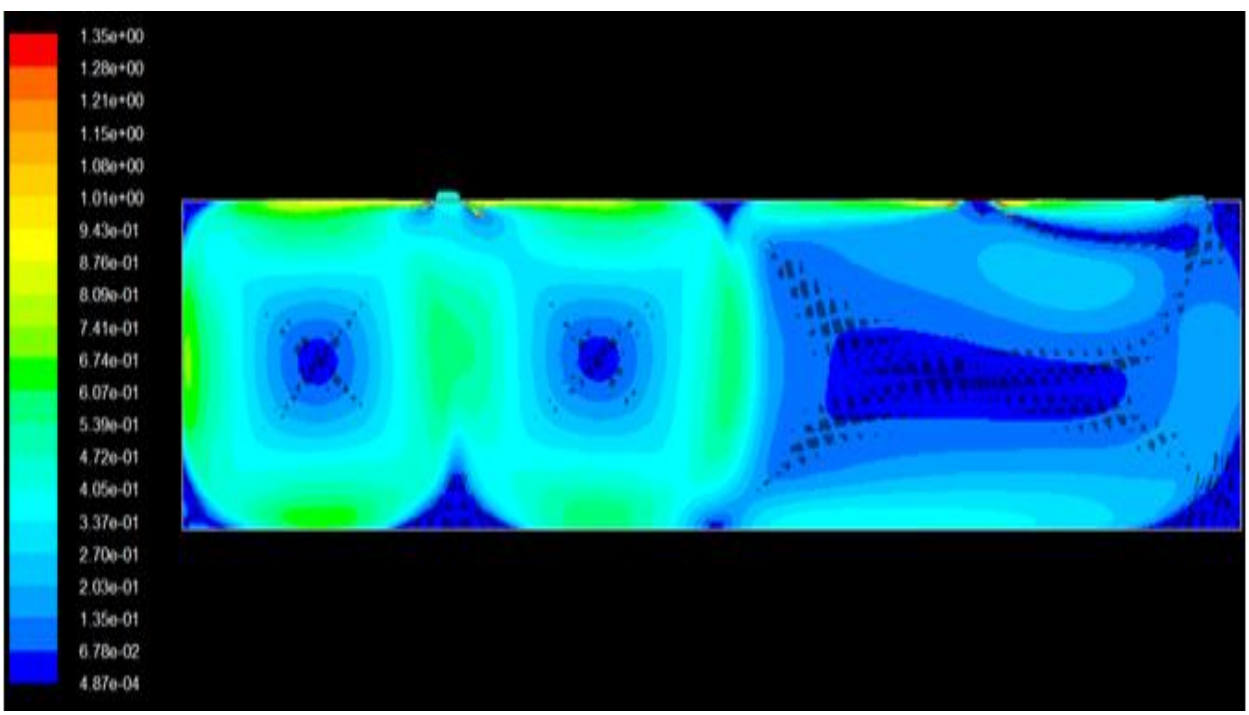


Figure 5-6. Velocity vectors of airflow inside the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); 2-D Realizable k-ε Turbulence Model Simulation

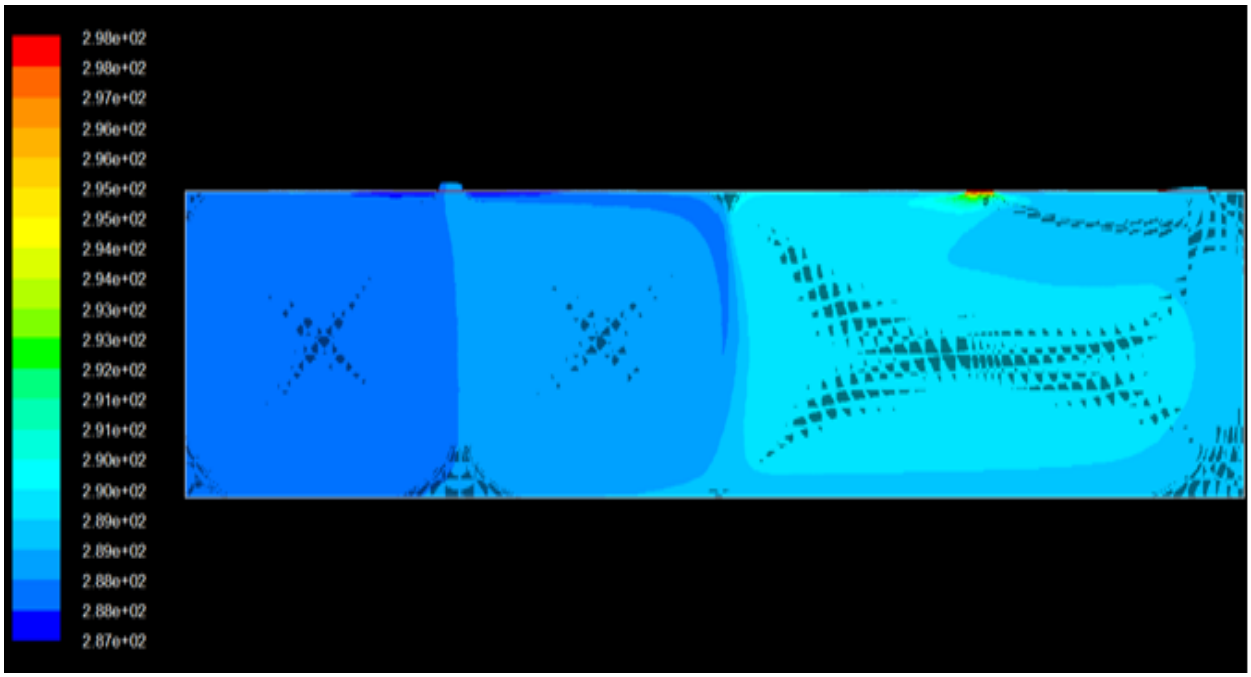


Figure 5-7. Temperature of airflow inside the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); 2-D Realizable k-ε Turbulence Model Simulation

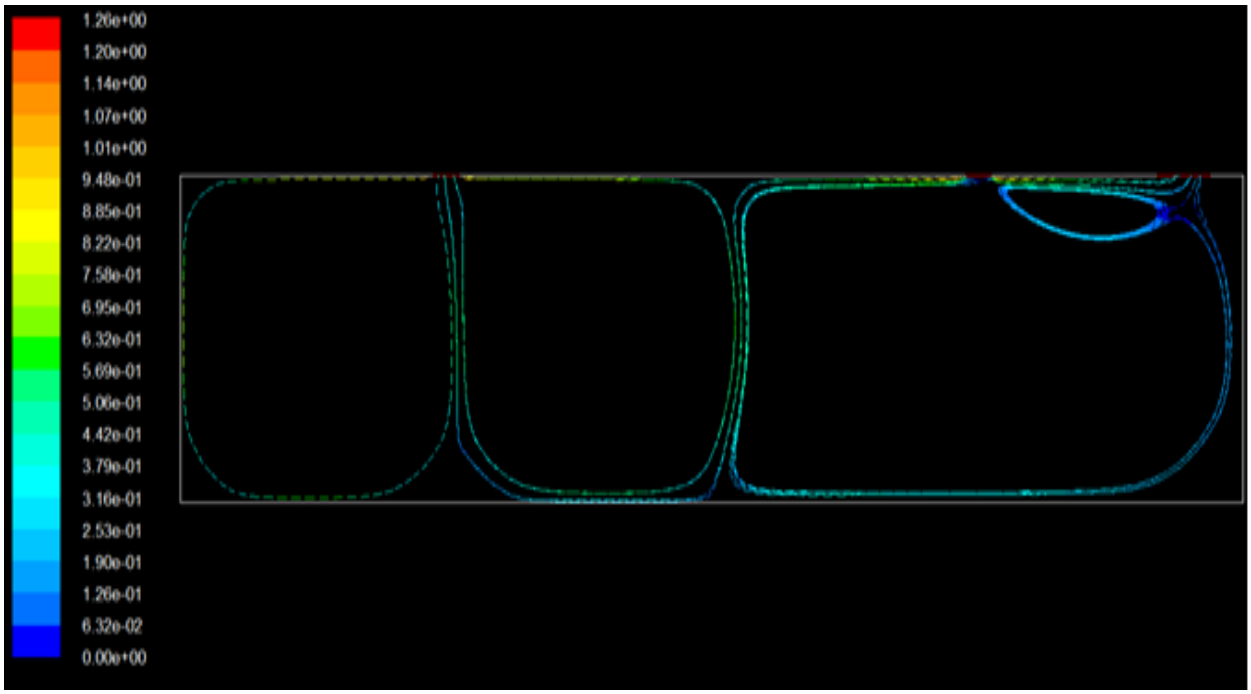


Figure 5-8. Path lines showing the velocity magnitude of airflow inside the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); 2-D Realizable k-ε Turbulence Model Simulation

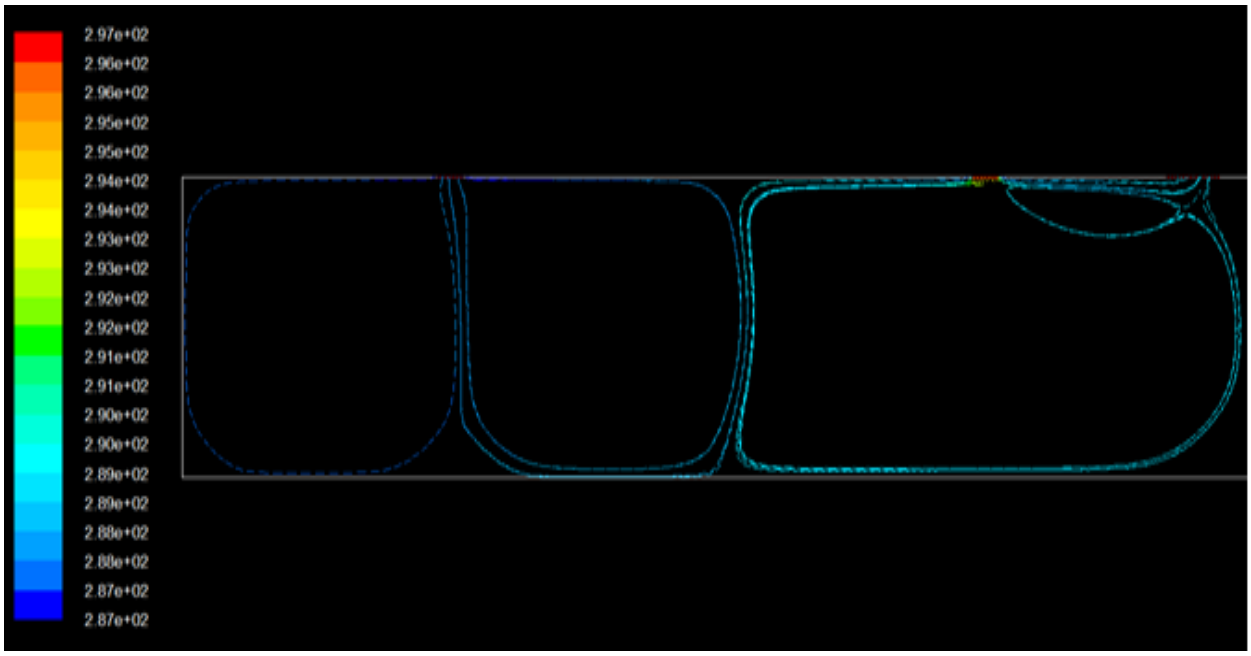


Figure 5-9. Path lines showing the temperature of airflow inside the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); 2-D Realizable k-ε Turbulence Model Simulation

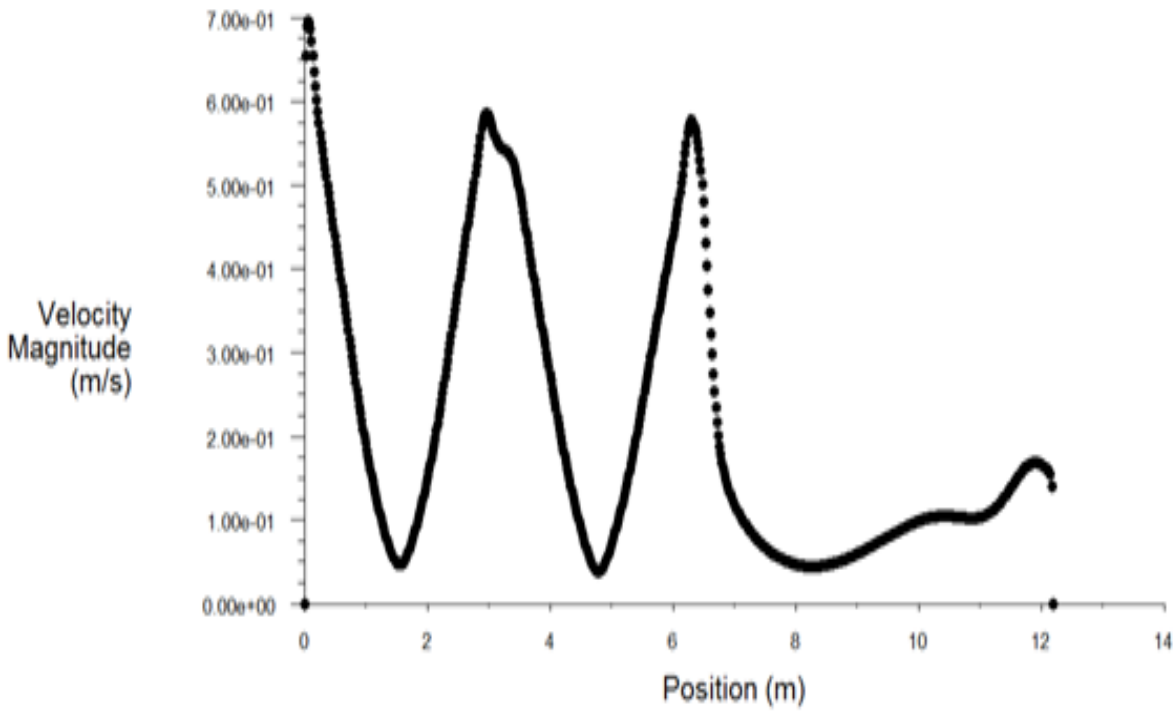


Figure 5-10. Plot of velocity magnitude (at a height of 1.7 m) across the room; Info: X-axis: length of the room (0-12.192m); Y-axis: velocity magnitude of airflow inside the room (0-0.7m/s); 2-D Realizable k-ε Turbulence Model Simulation

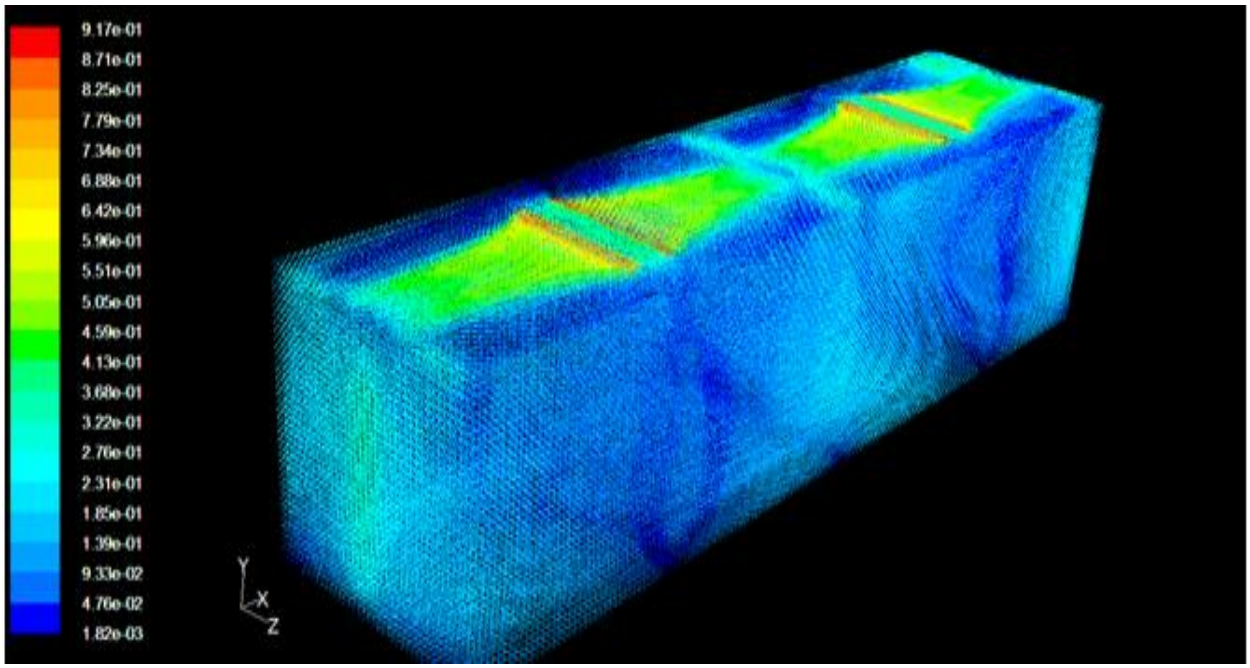


Figure 5-11. Velocity vectors of airflow inside the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); Z-axis: width of the room (0-3.048 m); 3-D Standard k-ε Turbulence Model Simulation

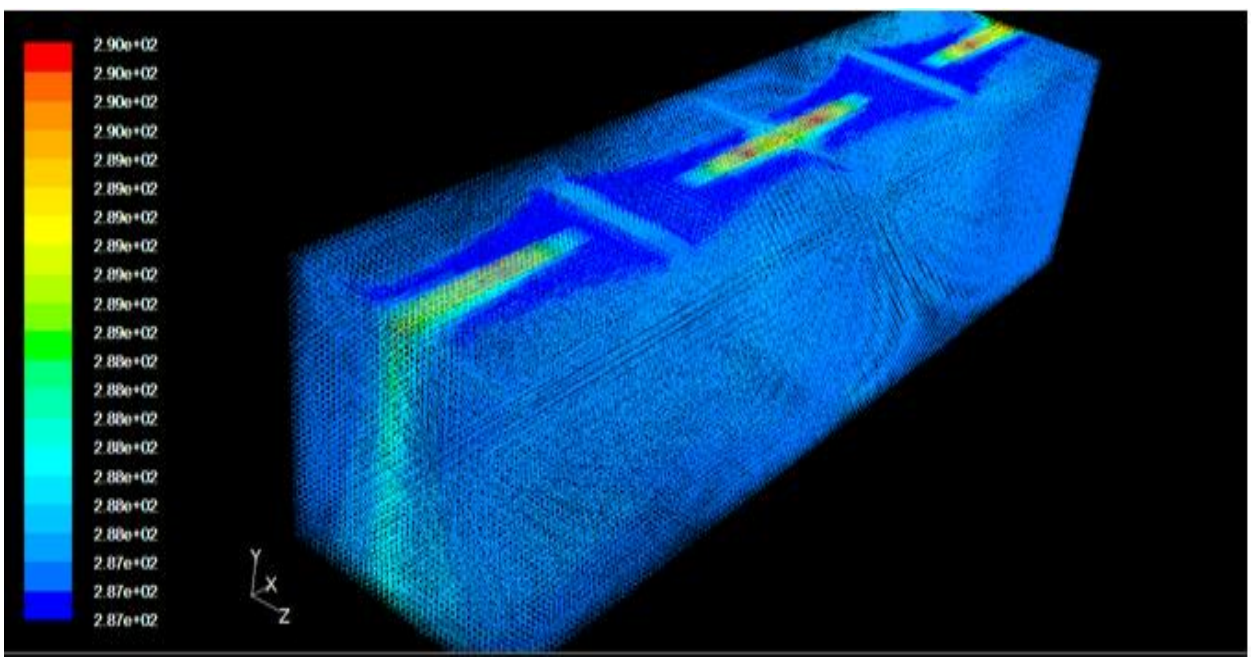


Figure 5-12. Temperature of airflow inside the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); Z-axis: width of the room (0-3.048 m); 3-D Standard k-ε Turbulence Model Simulation

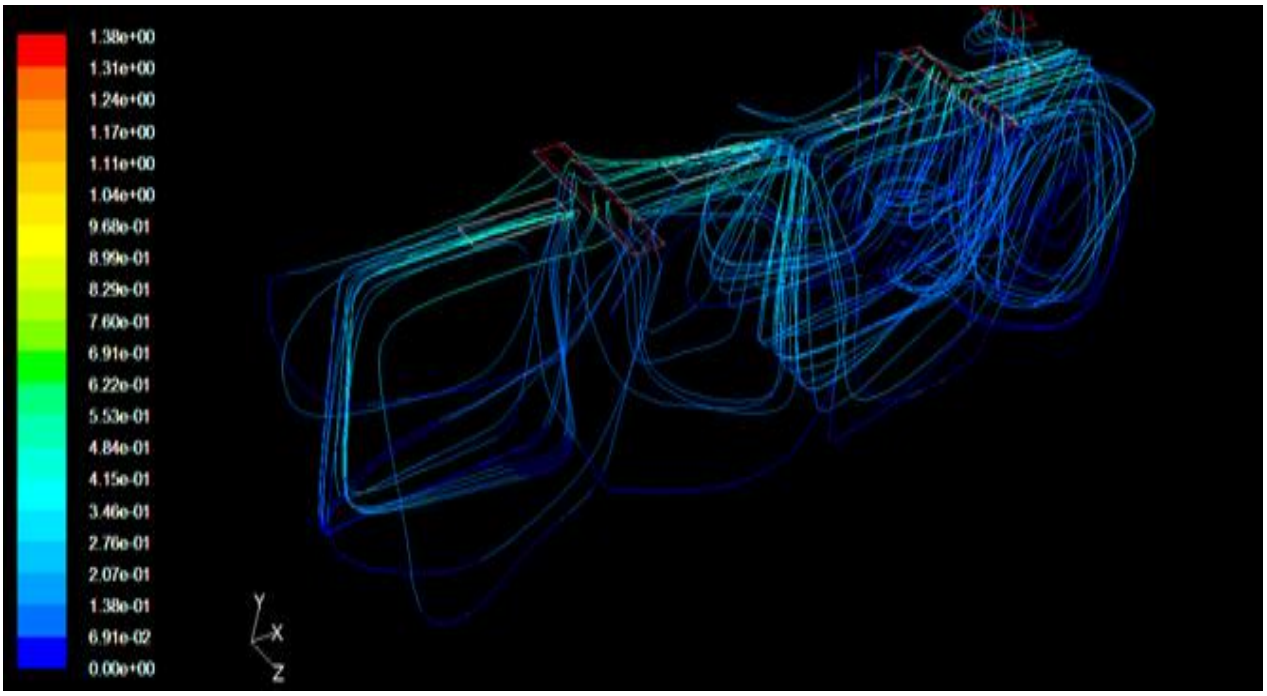


Figure 5-13. Path lines showing the velocity magnitude of airflow inside the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); Z-axis: width of the room (0-3.048 m); 3-D Standard k-ε Turbulence Model Simulation

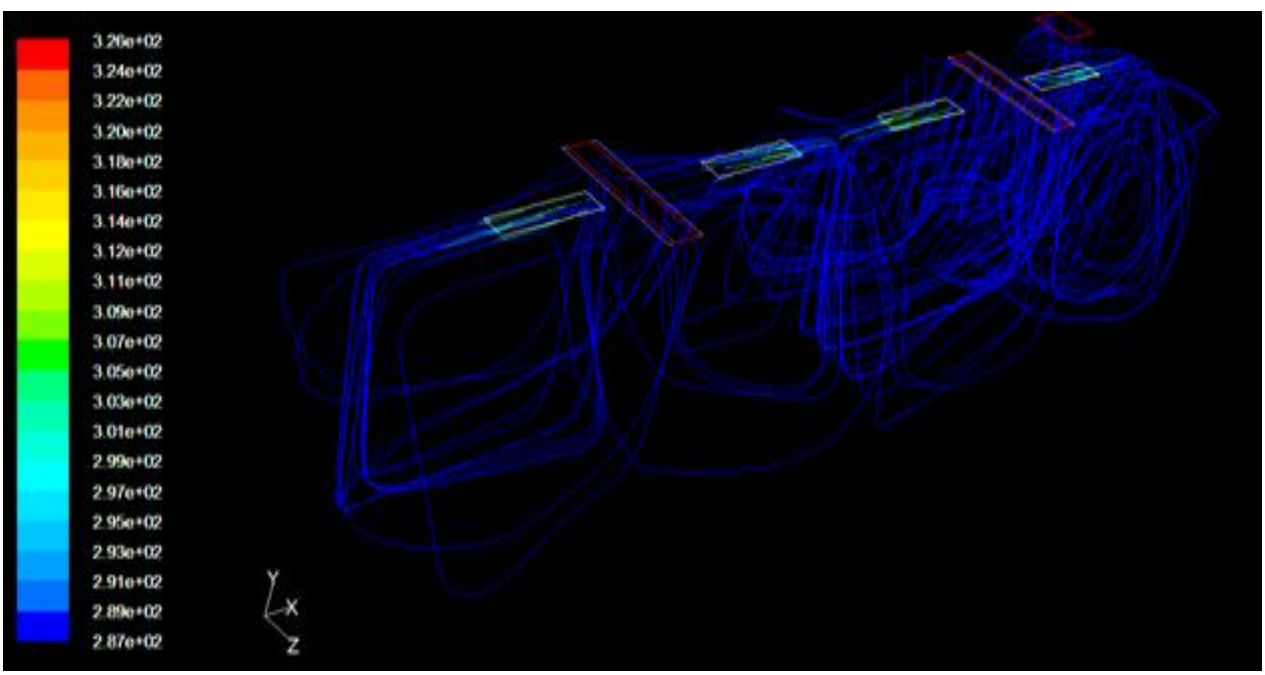


Figure 5-14. Path lines showing the temperature of airflow inside the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); Z-axis: width of the room (0-3.048 m); 3-D Standard k-ε Turbulence Model Simulation

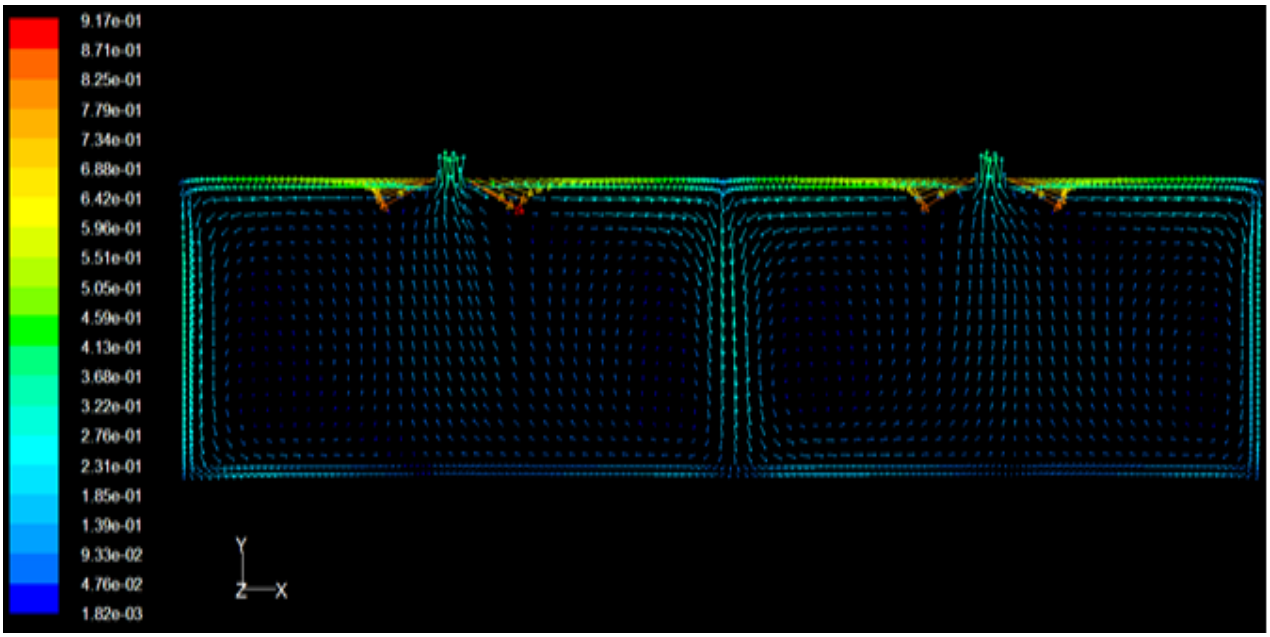


Figure 5-15. Velocity vectors of airflow (at the lateral mid-section) across the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); Z-axis: width of the room (0-3.048 m); 3-D Standard k-ε Turbulence Model Simulation\

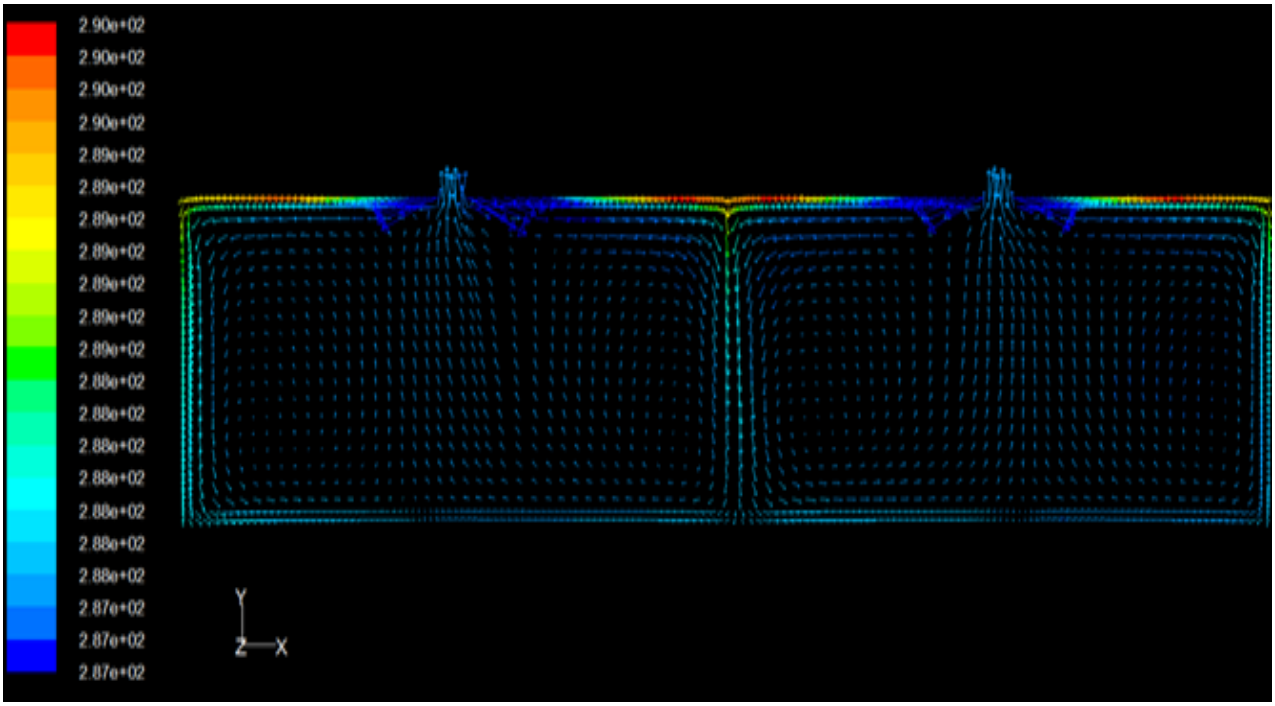


Figure 5-16. Temperature of airflow (at the lateral mid-section) across the room; Info: X-axis: length of the room (0-12.192m); Y-axis: height of the room (0-3.048m); Z-axis: width of the room (0-3.048 m); 3-D Standard k-ε Turbulence Model Simulation

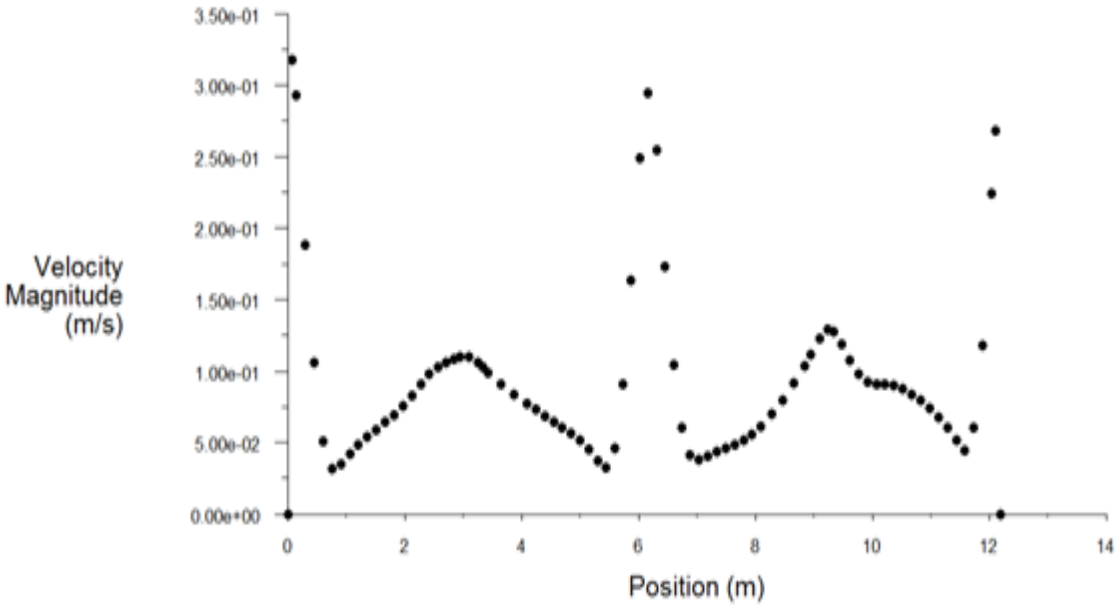


Figure 5-17. Plot of velocity magnitude of airflow (at the lateral mid-section) across the room; Info: X-axis: length of the room (0-12.192m); Y-axis: velocity magnitude of airflow inside the room (0-0.7m/s); 3-D Standard k-ε Turbulence Model Simulation

CHAPTER 6
VALIDATIONS

This chapter contains information about the data obtained from numerical simulations and TROX selection program which are then compared with each other to validate the numerical model. The accuracy of the physics depicted by a numerical simulation is predicted by comparing the results obtained from the simulation with already available standard results present in the form of a) experimental data or b) numerical results from a reliable source or c) empirical data. In the present case, because of lack of experimental data, the results are validated with the data obtained from TROX DID 612B-HC active chilled beam calculation (version 1.1) which is a spreadsheet downloadable from TROX website. The spreadsheet provides information regarding the velocities at certain locations inside the room which is believed to be empirical data. The data used for validation of the present results is shown in the Figure 6-2.

From the spread sheet, the velocity values at locations 1 and 2 (denoted as L and H_1 by TROX) are supposedly

V_1=104.2 fpm

V_2 =98.3 fpm

From the velocity plots shown in figure 5.5 and 5.11, the velocity magnitude at locations 1 and 2 predicted by the standard and realizable k-ε model are:

Standard k-ε model:

V_1 = 0.488 m/s =96.2 FPM

V_2 = 0.483m/s =95.2 FPM

Realizable k-ε model:

V_1 =0.605 m/s=118 FPM

V_2 = 0.48 m/s=94.5 FPM

These results are summarized in the table 6-1.

Table 6-1. Parameters V_1 and V_2 predicted by various models

Parameter	Empirical model	Standard model	Realizable model
V_1 (fpm)	104.2	96.2	118
V_2 (fpm)	98.3	95.2	95.2

Table 6-2. Error in prediction of V_1 and V_2 by the turbulence models

Parameter	Standard model	Realizable model
V_1 (% error)	-7.67	13.24
V_2 (% error)	-3.15	-4.02

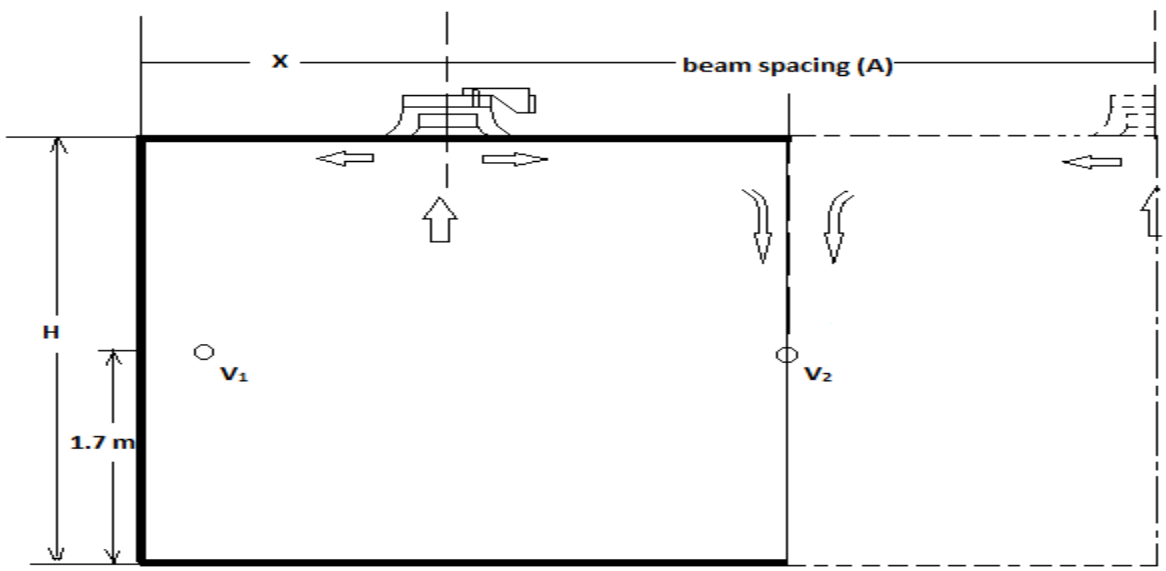


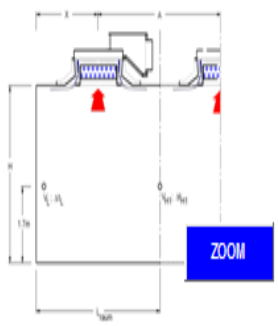
Figure 6-1. Locations showing the points of interest where the data is to be compared (adapted from: chilled beam design guide [14])

TROX® TECHNIK DID 612B-HC Active Chilled Beam Calculation (Version 1.1)

Input DID	2 water circuits (4 Pipe Coil)		1 water circuit (2 Pipe Coil)		Project	room-No.	comment
	cooling	heating	cooling	heating			
Vwater DID	2.0000 GPM	2.0000 GPM	2.0000 GPM	2.0000 GPM			
Unit length	6.0 ft						
Nozzle-type	g						
Vair-primary DID	125.0 CFM						
	25.0 l/s						
spigot diameter	6.0 inches						

Input temperatures	cooling		heating		Input room measures	
Tair-primary	55.0 °F		70.0 °F		room height	10.0 ft
Troom / rel. Humidity	74.0 °F	50.0 %	70.0 °F	50.0 %	A	10.0 ft
Twater-flow	56.0 °F		110.0 °F		X	10.0 ft

Restore nominal spigot diameter | Unit primary airflow:CFM | Vwater DID = const



results	2 water circuits		1 water circuit	
	cooling	heating	cooling	heating
Δtwater	-4.0 °F	6.9 °F	-4.4 °F	9.5 °F
Twater-return	60.0 °F	103.1 °F	60.4 °F	100.5 °F
ΔT room - water flow	-18.0 °F	40.0 °F	-18.0 °F	40.0 °F
ΔT Room water average	-16.0 °F	36.6 °F	-15.8 °F	35.3 °F
Qwater DID	-4016 BTUH	6884 BTUH	-4438 BTUH	9465 BTUH
Qair DID	-2591 BTUH	0 BTUH	-2591 BTUH	0 BTUH
Q DID	-6606 BTUH	6884 BTUH	-7029 BTUH	9465 BTUH
ΔP water	2.349 ft WG	7.506 ft WG	3.133 ft WG	2.480 ft WG
ΔP air	0.83 inches WG			
NC (including 10 dB room absorbtion)	37.3			
NC / Side entry	37.3			

Language: english

Stand:28.02.2007

					support values	
vL	104.2 fpm		104.2 fpm			
vH1	98.3 fpm		98.3 fpm			
ΔTL	-1.6 °F		-1.5 °F		N-nozzles total	134
ΔTH1	-0.5 °F		-0.5 °F		Aeff	0.039434 ft²
X-krit	14.8 ft		14.8 ft		veff	3169.9 fpm
Archimedes-number	0.000071		0.000070		H1	4.4 ft
ΔTsupply	16.6 °F		16.4 °F		L	14.4 ft
Connection-diameter / primary air	6.0 inches				room air dew point-cooling	54.2 °F
Re wasser	171	402	171	402		
F wasser	1.04	1.04	1.04	1.04		

Figure 6-2. TROX active chilled beam calculation program

CHAPTER 7
CONCLUSIONS AND RECOMMENDATIONS

7.1 Conclusions

a) The velocity parameters (V_1 and V_2) predicted by the standard k-ε turbulence model match closely with those of the 2-D empirical model with an error of about 7% for V_1 and 3% for V_2

b) The velocity parameters (V_1 and V_2) predicted by the realizable k-ε turbulence model match with those of the 2-D empirical model with an error of about 13% for V_1 and 4% for V_2

c) Based on the above figures, it seems that the standard model which is actually a high Reynolds's number turbulence model predicts the flow field better than the realizable model ,which is a more refined k-ε turbulence model intended to suit a variety of practical turbulent flow cases

d) The Laminar model failed to give a converged solution for both the two dimensional as well as the three dimensional simulation. This makes sense because most of indoor airflow problems fall under the turbulent regime. Similar phenomenon was observed by Stamou et al [12]

e) There is an almost uniform temperature distribution inside the room with a horizontal temperature gradient of about 1.8 °F (1°K) across the room with active chilled beam air-conditioning

f) The 3-D simulation shows that the velocity across the occupied zone is less than 60 FPM and that the room temperature is almost uniform

g) Based on 2-D simulations, an attempt has been made to compare the performance in terms of air velocities and temperature distribution inside the room for a conventional multi-cone diffuser and an active chilled beam. It may be concluded that the chilled beam provides a more uniform temperature distribution. (For details refer to Appendix- B)

h) The air velocities with an active chilled beam are higher than those observed with a multi-cone diffuser which should results in a quicker response to changes in heat load(For details refer to Appendix- C)

7.2 Recommendations for Further Studies

a) Unsteady state simulations can be performed to capture the dynamic variation of velocity and temperature inside the room

b) The present study is focused on an empty room and neglects buoyancy effects .However, it will be interesting to simulate a real life laboratory by placing obstructions (desks and chairs) and heat sources (like computers, people, burners etc.) and see how the airflow gets affected due to the presence of these factors

c) There is a lack of lack of experimental data for comparisons to validate the results obtained from 3-D simulation. Also, it would be unfair to say that the standard model predicts the actual physics of the problem better than the realizable model just based on two data points because the actual flow field depends on a lot of parameters such as presence of obstructions, location of the return etc and the overall airflow may be a lot different Hence, experimental studies may be conducted to check the validity of the present numerical model and also to determine the correct turbulence model which best suits the present case

d) Experimental studies may also be performed to check if the active chilled beams provide faster cooling than the multi-cone diffusers

e) Significant efforts were made to simplify the modeling of the chilled beam unit. Each of these trials are listed below which may be helpful for future research on this topic

   Trial 1: The first case involved modeling the actual geometry of the chilled beam, which resulted in a complex grid (unstructured mesh) and this also could not provide a converged solution. It requires a lot of computational intensity and time.

   Trial 2: The geometrical modeling of the chilled beam unit was simplified using a flat diffuser model i.e. modeling the geometry of the beam unit by identifying the mass flow inlet and exit portions as flat openings on the ceiling. However, this lead to airflow inside the room which contradicts with the “Coanda Effect” (jet must follow the wall) i.e. Instead of the jet diffusing out from the inlets and cascading down from the walls, it" hit the floor and came towards the ceiling in the opposite direction

   Trial 3: The third and final model assumed the same flat opening as the one in the second trial. But this time, the boundary conditions were specified so as to make the airflow eject at an angle set by the actual geometry of the beam unit. This reduced the computational intensity because of the luxury of being able to use a uniform structured mesh and also lead to good agreement with the coanda effect i.e. the air diffused out from the vents and cascaded down the walls

APPENDIX A
SOLID ROOM AND MESHED GEOMETRIC MODELS

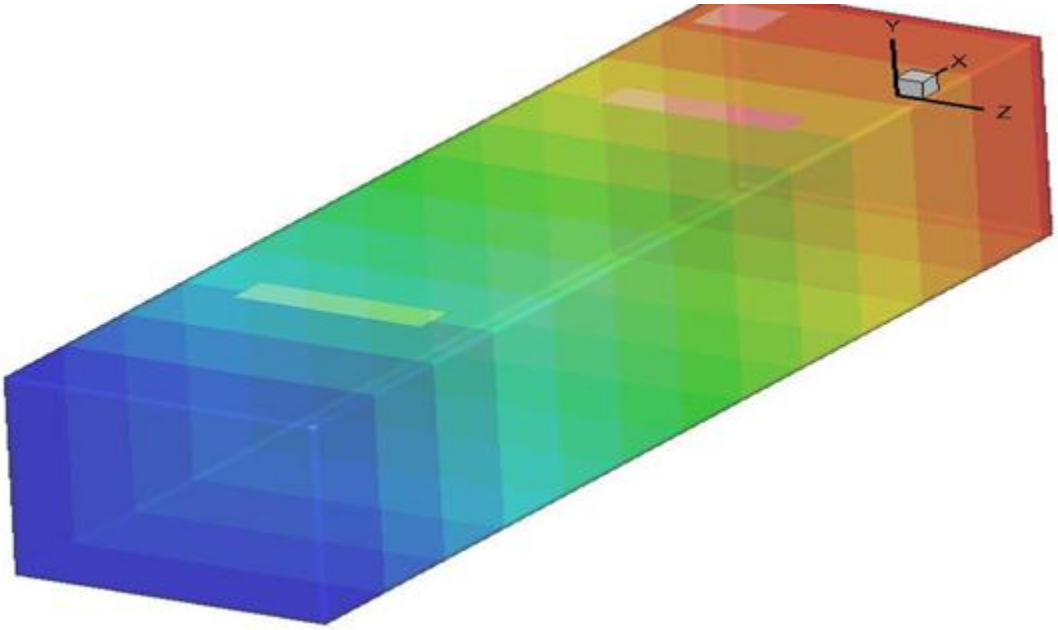


Figure A-1 Isometric view of the room model showing the gradient in x-direction

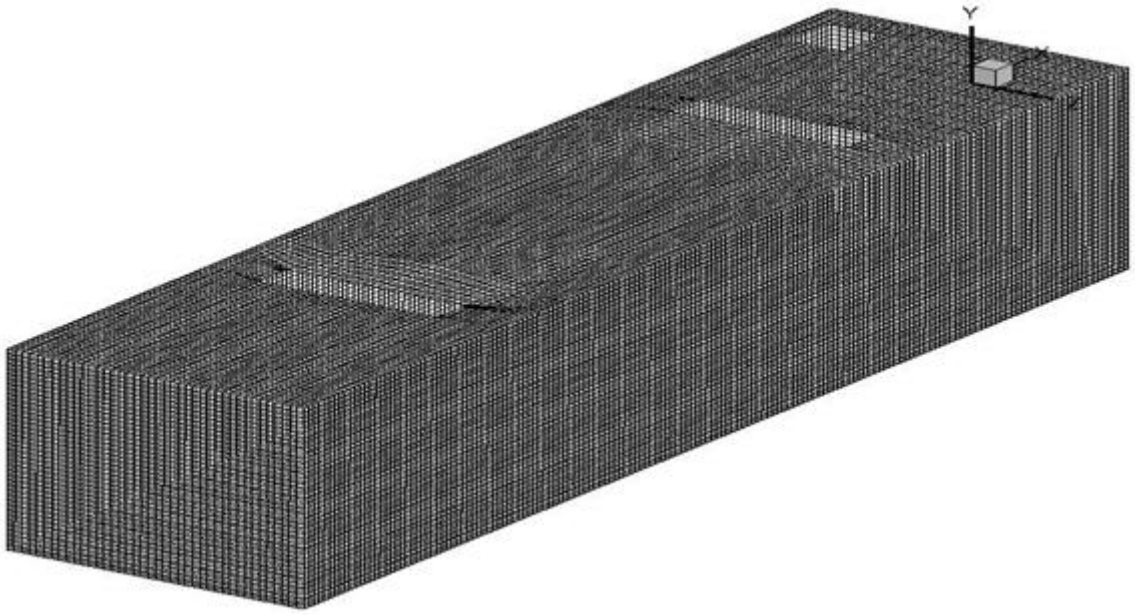


Figure A-2 Isometric view of the meshed room model

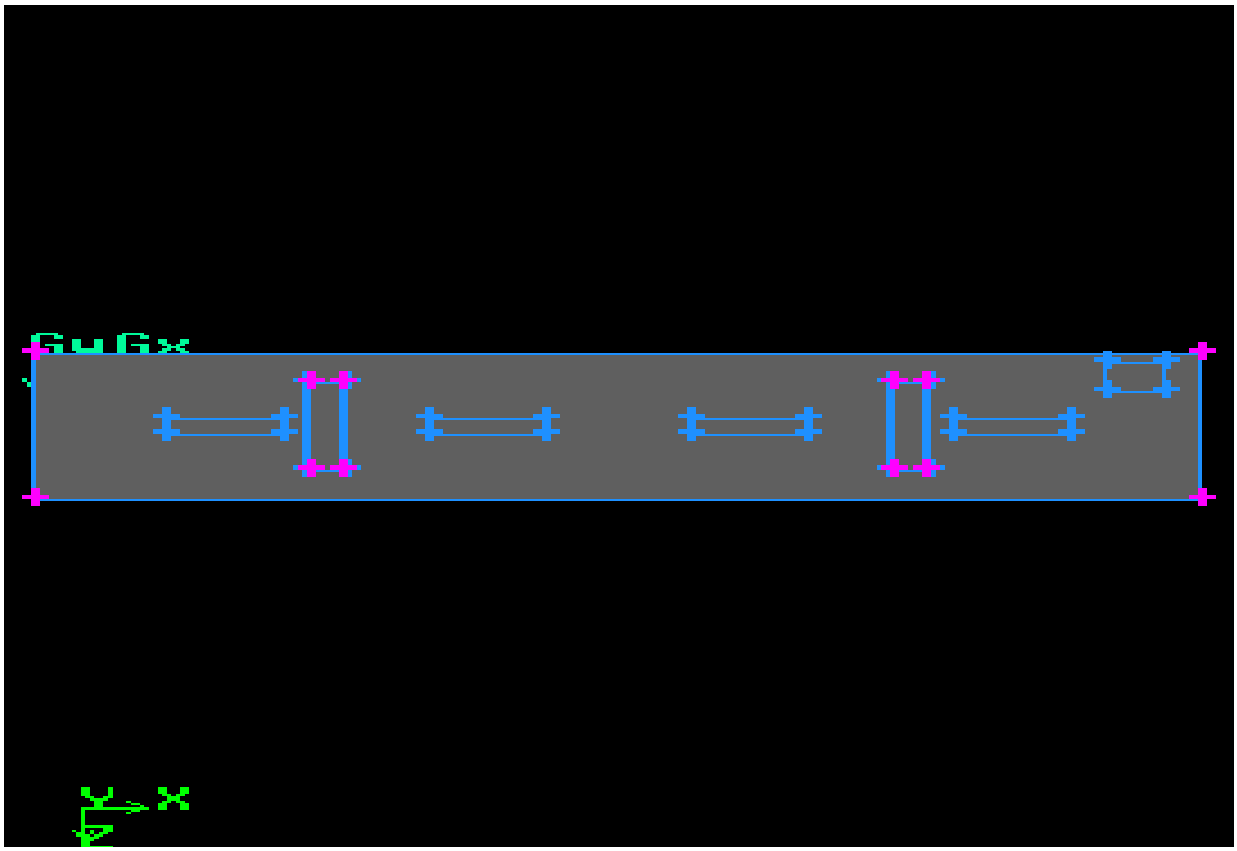


Figure A-3 Top View of the solid room model obtained from gambit

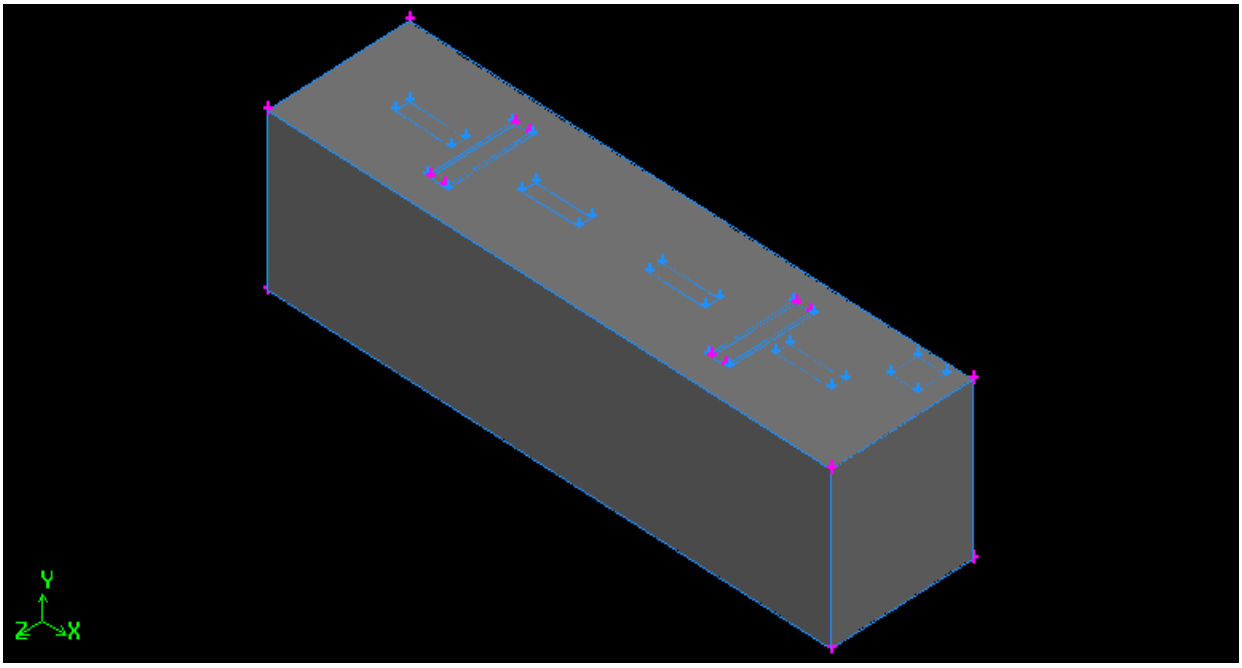


Figure A-4 Isometric view of the solid room model obtained from gambit

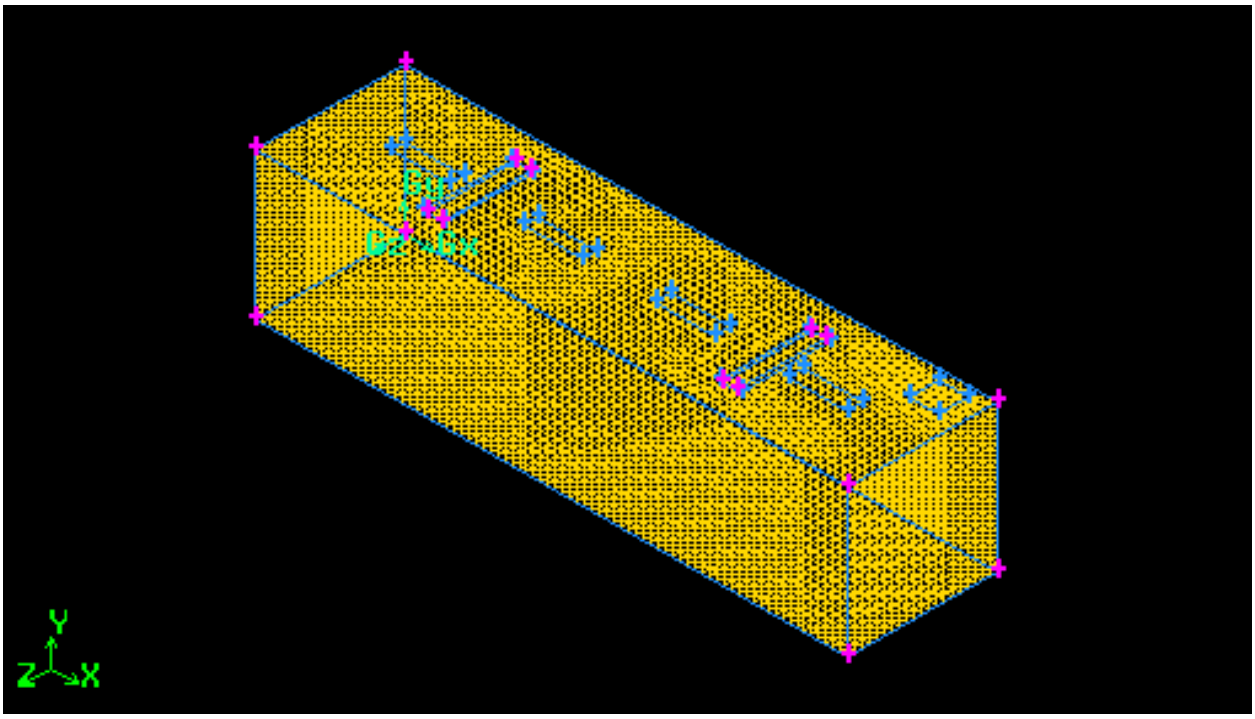


Figure A-5 Isometric view of the meshed room model obtained from gambit

APPENDIX B
MESH CONVERGENCE STUDY

In any numerical study, it is important that we use a sufficiently refined mesh to ensure that the results obtained are accurate. Coarse meshes can yield inaccurate results in analyses. The numerical solution provided by the model tends to a unique solution as we increase the mesh density. But, a finer mesh also leads to an increase in computational resources required. The mesh is said to be refined when further mesh refinement produces a negligible change in the solution. Hence, a mesh convergence study has been performed on the present 2-D model to determine the accuracy of the results. Since the primary focus is on velocity, the velocity magnitudes U_1, U_2, U_∞ obtained from various mesh sizes will be compared to arrive at an optimum mesh size (Refer to Table B-1).

Where,

U1= sum of all the velocities at a certain section across the room $(\sum \quad Ui)$

U2=square root of summation of square of all the velocities at a certain section across the room ($\sqrt{\sum \quad Ui^2}$)

U_∞ =Maximum velocity of all the velocities at a certain section across the room (U_{max})

Four grids starting from .025m mesh edge length have been generated for the 2-D geometric model of the room and FLUENT runs have been conducted for the same case on these meshes. The velocity vectors at a certain section across the room were taken and the parameters U1, U2 and U_∞ were determined for each mesh. These velocities were then plotted to check it they flatten out (Refer to Figure B-1).

It can be deduced from the plot that after the 0.02m mesh element length, the curve tends to flatten out for U_2 and $U\infty$ but the U1 is increasing. Hence, the 0.02m mesh element edge size was chosen for this analysis as it is evident from the above table that there is not much change in the results for mesh element length less than the 0.02m. Above decision was arrived at, taking into consideration the time taken for the FLUENT runs and the accuracy of the results. Hence the 0.02m mesh can be considered as a *fine mesh* for the present analysis.

Table B-1 Comparison of the effect of grid on solution

Parameter	0.025m mesh	0.02m mesh	0.018m mesh	0.015m mesh
U_1	3.370573	3.606919	3.704887	3.803583
U_2	0.919054	1.007596	1.029458	1.052654
$U\infty$	0.478278	0.501668	0.51048	0.531936

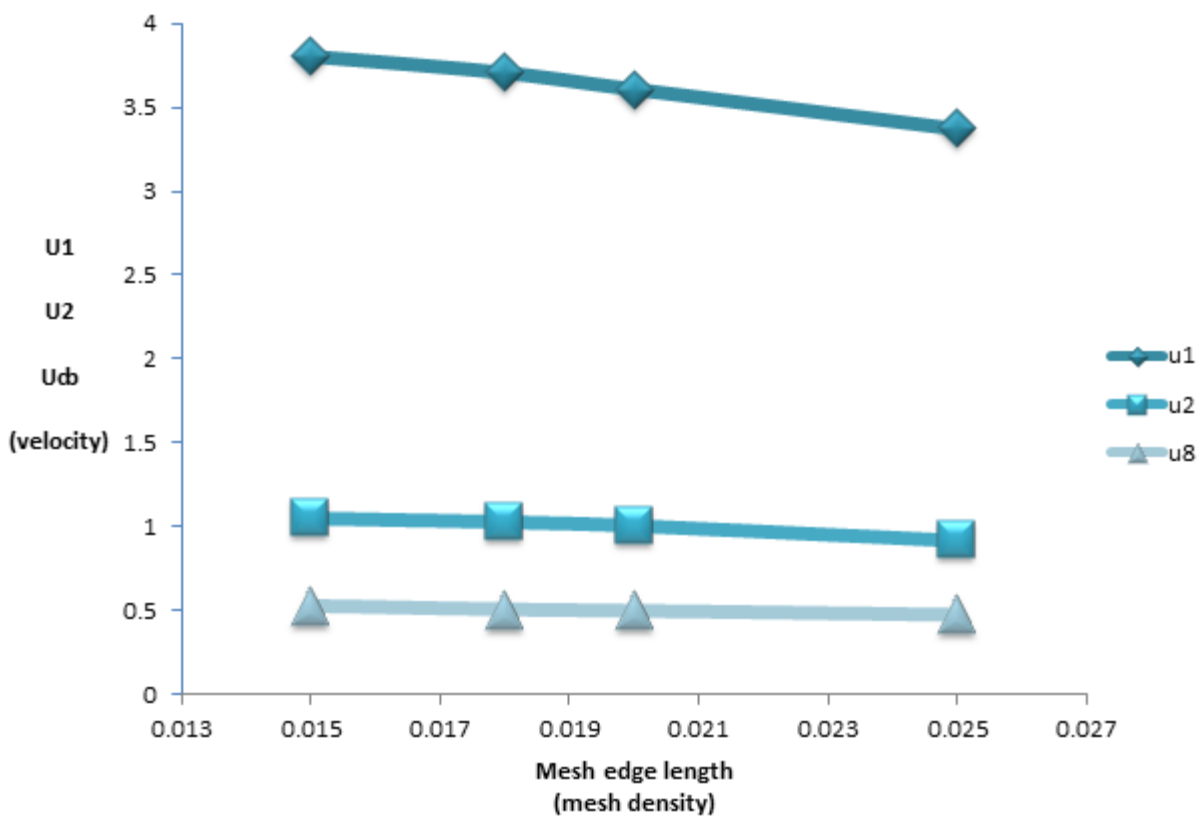


Figure B-1 Plot of velocity parameters as the mesh density varies

APPENDIX C
COMPARISION WITH A MULTI-CONE DIFFUSER

For performing the 2-D numerical simulation of a room fitted with a multi-cone diffuser, the box method suggested by Kotani et al [13] was used with Standard k-ε turbulence modeling and third-order QUICK discretization scheme. The same mass flow as that used for the active chilled beams was used for the simulation for proper comparison.

Remarks and Observations

a) From the numerical results, it may be deduced that the maximum velocity of airflow inside the 2-D room model with a multi-cone diffuser is about 60 fpm whereas that with an active chilled beam is about 90-100 fpm

b) There is an almost uniform temperature distribution inside the room with an active chilled beam

c) The portion of the room directly below the multi-cone diffuser is cooler than the other portions of the room

d) Experimental studies on a multi-cone diffuser and an active chilled beam unit may be performed for having a better understanding of the airflow properties for a fair comparison between these two models

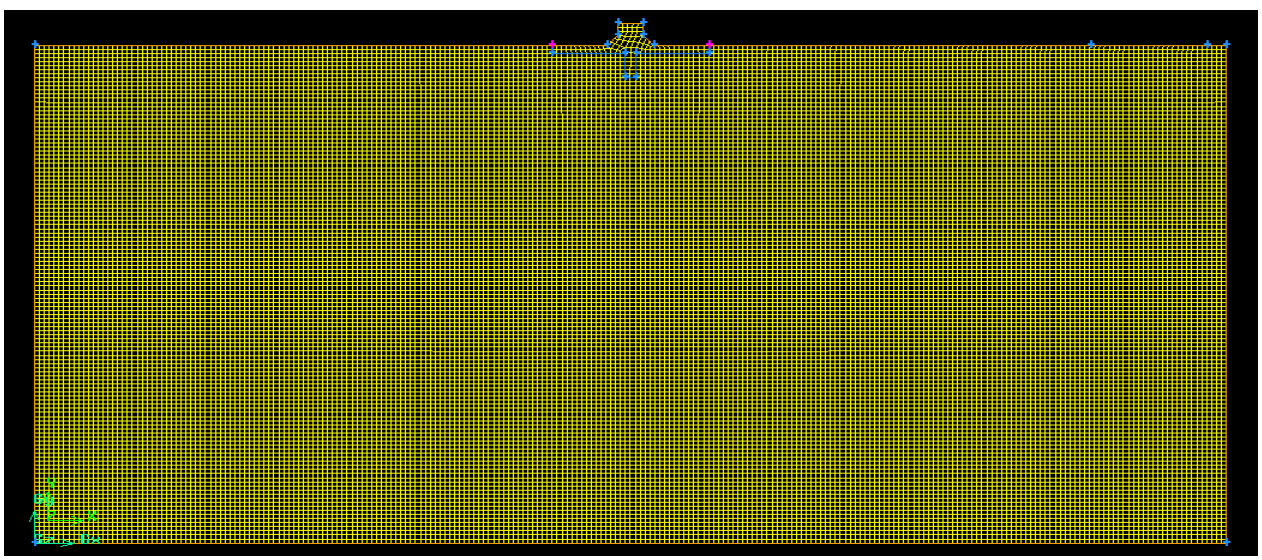


Figure C-1 Front view of the meshed 2-d room model fitted with a multi-cone diffuser

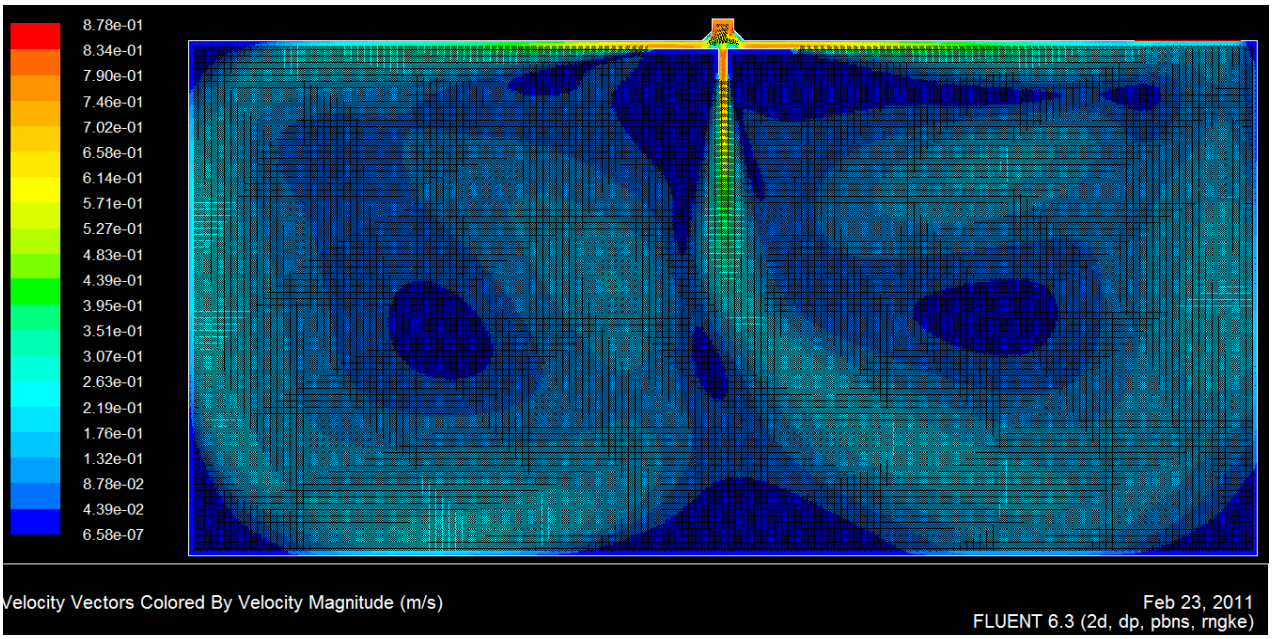


Figure C-2 Velocity vectors of airflow inside the room

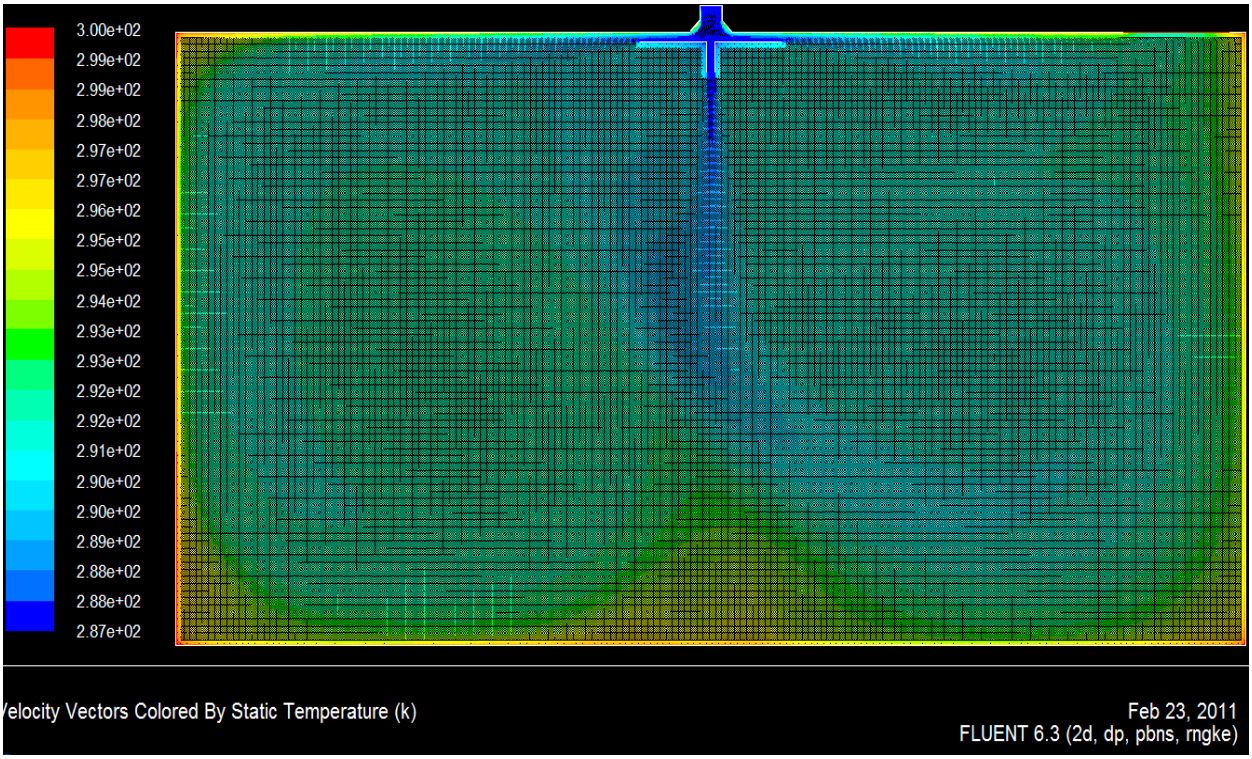


Figure C-3 Temperature of airflow inside the room

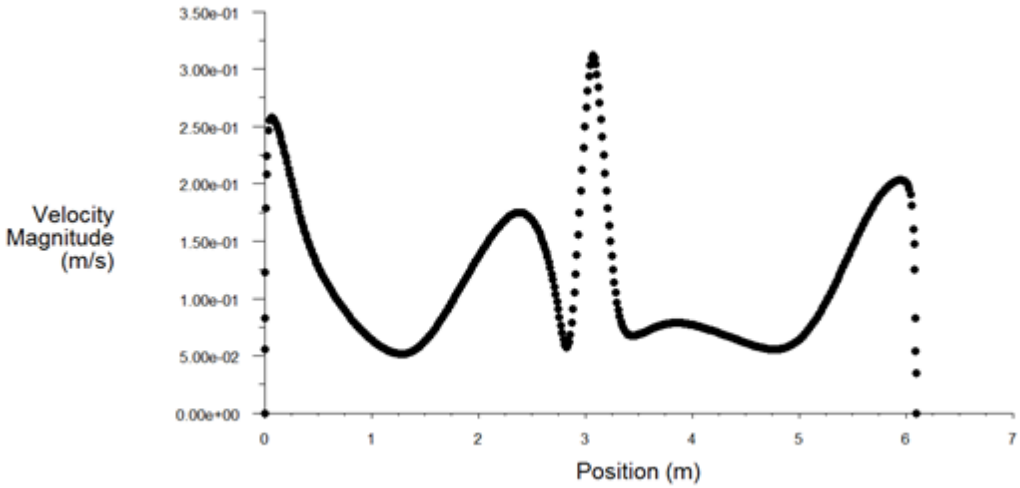


Figure C-4 Plot of velocity magnitude of airflow (at a height of 1.7m) across the room

LIST OF REFERENCES

1. Dwyer, T., and Staunton, J., 2007. “Chilled ceilings, chilled beams and integrated service modules-alternative approaches to cooling,” *CPD Collection*, SAS International, 11, 89-92.

2. Srebric, J. and Chen, Q., 2002. “An example of verification, validation, and reporting of indoor environment CFD analyses,” *ASHRAE Transactions*, 108(2), 185-194.

3. Zhu, L., Li, R., and Yuan, D., 2006. “Numerical analysis of a cold air distribution system,” *HVAC Technologies for Energy Efficiency,* Vol.IV-2-3.

4. Abanto, J., Barrero, D., Reggio, M., and Ozell, B., 2004. “Airflow modeling in a computer room,” *Building and Environment*, 39, 1393-1402.

5. Awbi, H., 1989. “Applications of computational fluid dynamics in room ventilation,” *Building and Environment*, 24(1), 73-84.

6. Melikov, A., Yordanova, B., Bozkhov, L., Zboril, V., and Kosonen, R., 2007. “Impact of the airflow interaction on occupants’ thermal comfort in rooms with active chilled beams,” *The 6th International Conference on Indoor Air Quality, Ventilation & Energy Conservation in Buildings IAQVEC 2007*, Sendai, Japan.

7. Kosonen, R., Saarinen, P., Koskela, H., and Hole, A., 2010. “Impact of heat load location and strength on air flow pattern with a passive chilled beam system,” *International Conference on Building Energy and Environment COBEE2008*, Vol.42 (1), 34-42.

8. Cammarata, G., and Petrone, G., 2008. “A numerical investigation on active chilled beams for indoor air conditioning,” *COSMOL Conference*, Hannover.

9. True, J., Zboril, V., Kosonen, R., and Melikov, A., 2007. “Consideration for minimizing draught discomfort in rooms with active chilled beams,” *Proceedings of Clima 2007-Wellbeing Indoors*.

10. Posner, J., Buchanan, C., and Rankin, D., 2003. “Measurement and prediction of indoor air flow in a model room,” *Energy and Buildings*, Vol.35 (1), 515-526.

11. Zboril, V., Bozhkov, L., Yordanova, B., Melikov, A., and Kosonen, R., 2006. “Airflow distribution in rooms with chilled beams,” *17th Air-conditioning and Ventilation Conference*, Prague, Czech Republic.

12. Stamou, A., and Katsiris, L., 2006. “Verification of a CFD model for indoor airflow and heat transfer,” *Building and Environment*, 41, 1171-1181.

13. Kotani, H., Yamanaka, T., and Momoi, Y., 2002. “CFD simulation of airflow in room with multi-cone ceiling diffuser using measured velocity and turbulent parameters in large space,” *Proceedings of the 8th International Conference on Air Distributions in Rooms (RoomVent2002)*, Copenhagen, Denmark, 117-120.

14. Trox Inc., 2009. “Chilled beam design guide,” USA.

15. Lobscheid, C., and Gadgil, A., 2002. “Mixing of a point-source indoor pollutant: numerical predictions and comparison with experiments.”*Proceedings of Indoor Air2002*, Vol.IV, 223-228.

16. Cheong, K., Djunaedy, E., Chua, E., Tham, K., Sekhar, S., Wong, N., and Ullah, M., 2003. “Thermal comfort study of an air-conditioned lecture theatre in the tropics,” *Building and Environment*, Vol.38 (1), 63-73.

17. Loomans, M., 1998. “The measurement and simulation of indoor air flow,” Doctoral Thesis, Department of Mechanical Engineering, Eindhoven University of Technology.

18. Fredriksson, J., Sandberg, M., Moshfegh, B., 2001. “Experimental investigation of the velocity field and airflow pattern generated by cooling ceiling beams,” *Building and Environment*, Vol.36 (7), 891-899.

19. Chow, W., 2001. “Numerical studies of airflows induced by mechanical ventilation and air-conditioning (MVAC) systems,” *Applied Energy*, Vol.68 (2), 135-159.

20. Melikov, A., Yordanova, B., Bozkhov, L., Zboril, V., and Kosonen, R., 2007. “Human response to thermal environment in rooms with chilled beams,” *Proceedings of Clima 2007-Wellbeing Indoors*, Helsinki.

BIOGRAPHICAL SKETCH

Abhijyoth Reddy Vempati was born in 1988 in Vijayawada, India. He was brought up in Hyderabad, the capital city of Andhra Pradesh, which is also known as the “City of Pearls”. He is passionate about machines in general and automobiles in particular which made him pursue mechanical engineering as his major in under-graduate studies. He received his bachelor’s degree in mechanical engineering from the Jawaharlal Nehru Technological University located in Hyderabad in 2009.

In fall 2009, he joined the Mechanical and Aerospace Engineering Department at the University of Florida and started working on his dissertation from spring 2010 under the guidance of Dr.Ingley. He received his master’s thesis degree from the University of Florida in the fall of 2011.

He plans to find a full time position as an engineer in the fields of Engine Manufacturing, Turbo Machinery or Thermal Analyses requiring Computational Fluid Dynamics Application and is looking forward to the challenges that await him.