Application of computational fluid dynamics for high energy efficiency design with human comfort of Cad-Vav and Ufad systems

Bhanu Rekha Bandhakavi
University of Nevada, Las Vegas

Follow this and additional works at: https://digitalscholarship.unlv.edu/rtds

Repository Citation
https://digitalscholarship.unlv.edu/rtds/2091

This Thesis is brought to you for free and open access by Digital Scholarship@UNLV. It has been accepted for inclusion in UNLV Retrospective Theses & Dissertations by an authorized administrator of Digital Scholarship@UNLV. For more information, please contact digitalscholarship@unlv.edu.
NOTE TO USERS

Page(s) not included in the original manuscript and are unavailable from the author or university. The manuscript was scanned as received.

vi

This reproduction is the best copy available.
APPLICATION OF COMPUTATIONAL FLUID DYNAMICS FOR HIGH ENERGY EFFICIENCY DESIGN WITH HUMAN COMFORT OF CAD – VAV AND UFAD SYSTEMS

By

Bhanu Rekha Bandhakavi

Bachelor of Engineering
Acharya Nagarjuna University, Guntur, India
2004

A thesis submitted in partial fulfillment of the requirements for the

Master of Science Degree in Mechanical Engineering
Department of Mechanical Engineering
Howard R. Hughes College of Engineering

Graduate College
University of Nevada, Las Vegas
May 2007
INFORMATION TO USERS

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleed-through, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.
The Thesis prepared by

Bhanu Rekha Bandhakavi

Entitled

Application of Computational Fluid Dynamics for High Energy Efficiency Design with Human Comfort of CAD-VAV and UFAD Systems

is approved in partial fulfillment of the requirements for the degree of

Master in Mechanical Engineering

Examination Committee Chair

Dean of the Graduate College

Examination Committee Member

Examination Committee Member

Graduate College Faculty Representative
ABSTRACT

Application of Computational Fluid Dynamics for High Energy Efficiency Design with Human Comfort of CAD – VAV and UFAD Systems

By

Bhanu Rekha Bandhakavi

Dr. Yitung Chen, Examination Committee Chair
Associate Professor, Department of Mechanical Engineering
University of Nevada, Las Vegas

This thesis deals with the numerical simulation of the ceiling air distribution (CAD) system and the Under Floor Air Distribution (UFAD) system based on the dimensions of BTLab at UNLV. Ceiling Air Distribution (CAD) with variable air volume (VAV) and Under Floor Air Distribution (UFAD) systems have been widely used in different countries. CAD-VAV and UFAD systems designs have been influenced by increasing emphasis on indoor air quality (IAQ), energy conservation, environmental effects, safety, and economics. So, 3-D Computational Fluid Dynamics (CFD) analysis technique was applied to design high energy efficiency and human comfort CAD-VAV and UFAD systems. The goal of this research project is to analyze energy efficiency with thermal comfort for CAD – VAV and UFAD systems and reduce the design cycle through the development of mathematical and computational models. The University of Nevada Las Vegas (UNLV) has conducted the laboratory phase of this task which was conducted by a different research team by which a test protocol has been developed and
implemented in the UNLV Center, the BTLab. This experimental task is to test the performance of UFAD systems compared to CAD systems, including comfort, energy use, indoor air quality IAQ. The experiment has been conducted based on ASHRAE Standard 113-1990 - Method of Testing for Room Air Diffusion. FLUENT® 6.2 is a computational fluid dynamics (CFD) software package to simulate fluid flow problems. The general purpose CFD code FLUENT® is used as a numerical solver for the present 3D simulation. A non-staggered grid storage scheme is adopted to define the discrete control volumes. The solver used is a segregated solver which is a solution algorithm with which the governing equations are solved sequentially. The SIMPLE algorithm is used to resolve the coupling between pressure and velocity. An implicit technique is used to linearize the discrete and non-linear governing equations. The discretization method used by the FLUENT® is FVM in which the space is divided into a finite number of control volumes and solves the partial differential equations. Integration of the governing equations on the individual control volumes constructs algebraic equations for the discrete dependent variables such as velocities, pressure and temperature. In this research work, thermal comfort environment of the CAD & UFAD system is investigated and compared with the experimental values.

From the numerical study of the BTLab for CAD system, results show that the temperature and velocity profiles inside the test space are well mixed. Three test planes have been studied to compare these numerical results with the experimental study which show good agreement. The spray angle of the swirl diffuser is considered to be the crucial part and the airflow distribution strongly depends on the spray angle. A Parametric study has been made on the spray angle of the swirl diffuser considering the spray angle from
3° to 7° in which the spray flow for single swirl diffuser is studied. From the velocity and temperature profiles and path lines, the approximated spray angle is around 5.3° which is considered to be a good choice. The UFAD system of BT Lab is numerically studied and thermal load is not considered in this study. The results show that the flow from the diffuser is highly helical and twisted and a clean zone is formed as per the previous publication [50]. This shows that the obtained results from the numerical study are reasonable. These numerical results for UFAD system are benchmarked with the experimental results.
# TABLE OF CONTENTS

ABSTRACT ......................................................................................................................................... iii

TABLE OF CONTENTS .................................................................................................................. vii

LIST OF FIGURES ......................................................................................................................... ix

ACKNOWLEDGEMENTS ................................................................................................................ xi

CHAPTER 1  INTRODUCTION ....................................................................................................... 1
  1.1 Background ............................................................................................................................... 1
  1.2 Basic Introduction ..................................................................................................................... 2
  1.3 Objective .................................................................................................................................. 9
  1.4 Motivation ................................................................................................................................ 9
  1.5 Literature Review .................................................................................................................... 10
  1.6 Thesis Outline .......................................................................................................................... 16

CHAPTER 2 PROBLEM DESCRIPTION AND METHODOLOGY .............................................. 17
  2.1 Problem Description ............................................................................................................... 17
  2.2 Experimental Setup ............................................................................................................... 19
  2.3 Methodology ........................................................................................................................... 22
    2.3.1 Software ............................................................................................................................ 22
    2.3.2 Governing Equations .................................................................................................... 24
    2.3.3 Standard k-e Model ........................................................................................................ 25
  2.4 Numerical Procedure ............................................................................................................. 30

CHAPTER 3 NUMERICAL MODELING OF THE CAD (CEILING AIR DISTRIBUTION) SYSTEM ........................................................................................................ 38
  3.1 CFD Analysis on the Single Diffuser Characteristics .................................................... 40
  3.2 CFD Analysis on the CAD System of BTLab ............................................................... 46
  3.3 Discussions .............................................................................................................................. 53

CHAPTER 4 NUMERICAL MODELING OF UFAD (UNDER FLOOR AIR DISTRIBUTION) SYSTEM ........................................................................................................ 55
  4.1 Pre-processing ......................................................................................................................... 57
    4.1.1 Creating the Geometry ................................................................................................... 57
    4.1.2 Mesh Generation ............................................................................................................. 59
  4.2 CFD Analysis on Single Swirl Diffuser ........................................................................ 60
    4.2.1 Numerical Procedure and Boundary Conditions .................................................. 61
    4.2.2 Single Swirl Diffuser Performance Analysis ...................................................... 63
  4.3 CFD Analysis on the UFAD System of BTLab without Thermal Load ....................... 67
    4.3.1 Numerical Procedure and Boundary Settings ................................................... 71
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Schematic of the square cone diffuser</td>
<td>3</td>
</tr>
<tr>
<td>2</td>
<td>General pattern of the air from the diffuser</td>
<td>4</td>
</tr>
<tr>
<td>3</td>
<td>Conventional ceiling air distribution system</td>
<td>4</td>
</tr>
<tr>
<td>4</td>
<td>Geometry and the general pattern of the air from the diffuser</td>
<td>5</td>
</tr>
<tr>
<td>5</td>
<td>Under Floor Air Distribution (UFAD) System</td>
<td>6</td>
</tr>
<tr>
<td>6</td>
<td>Displacement ventilation airflow</td>
<td>7</td>
</tr>
<tr>
<td>7</td>
<td>Hybrid under floor air flow system</td>
<td>8</td>
</tr>
<tr>
<td>8</td>
<td>Computational domain of the BTLab</td>
<td>18</td>
</tr>
<tr>
<td>9</td>
<td>Air Ducts of the BTLab</td>
<td>19</td>
</tr>
<tr>
<td>10</td>
<td>Condenser and Piping of the Daikin VRV System in the BTLab</td>
<td>20</td>
</tr>
<tr>
<td>11</td>
<td>Diffusers used in the BTLab</td>
<td>20</td>
</tr>
<tr>
<td>12</td>
<td>Traversing Mechanism and Data Acquisition Board in the BTLab</td>
<td>21</td>
</tr>
<tr>
<td>13</td>
<td>A typical control volume and the notation used for a cartesian 2D grid</td>
<td>33</td>
</tr>
<tr>
<td>14</td>
<td>Schematics of the four-way square cone diffusers</td>
<td>39</td>
</tr>
<tr>
<td>15</td>
<td>Diffuser model</td>
<td>39</td>
</tr>
<tr>
<td>16</td>
<td>Computational domain for the single diffuser study</td>
<td>40</td>
</tr>
<tr>
<td>17</td>
<td>Velocity vector graph at the center of the domain</td>
<td>42</td>
</tr>
<tr>
<td>18</td>
<td>Variation of velocity magnitude as a function of X at a selected slice</td>
<td>43</td>
</tr>
<tr>
<td>19</td>
<td>Velocity magnitude at center of the domain</td>
<td>43</td>
</tr>
<tr>
<td>20</td>
<td>Temperature profile at the center of the domain</td>
<td>44</td>
</tr>
<tr>
<td>21</td>
<td>Variation of air temperature as a function of X at a selected slice</td>
<td>45</td>
</tr>
<tr>
<td>22</td>
<td>Path lines released from the top inlet of the diffuser</td>
<td>45</td>
</tr>
<tr>
<td>23</td>
<td>Domain and computational grid system of the test space of BTLab</td>
<td>46</td>
</tr>
<tr>
<td>24</td>
<td>Path lines released from the top inlet of the diffuser</td>
<td>48</td>
</tr>
<tr>
<td>25</td>
<td>Velocity vector graph at a selected slice</td>
<td>49</td>
</tr>
<tr>
<td>26</td>
<td>Temperature profile of the domain at a selected slice</td>
<td>50</td>
</tr>
<tr>
<td>27</td>
<td>Variation of average surface air temperature and velocity magnitude as a</td>
<td>50</td>
</tr>
<tr>
<td></td>
<td>function of height</td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>Test positions with temperature and velocity sensors at BTLab</td>
<td>51</td>
</tr>
<tr>
<td>29</td>
<td>The air velocity magnitude at the different test positions</td>
<td>52</td>
</tr>
<tr>
<td>30</td>
<td>The air temperature at the different test positions</td>
<td>53</td>
</tr>
<tr>
<td>31</td>
<td>Geometry of Nailor floor diffuser</td>
<td>58</td>
</tr>
<tr>
<td>32</td>
<td>3-D Nailor floor diffuser</td>
<td>58</td>
</tr>
<tr>
<td>33</td>
<td>Part of the diffuser</td>
<td>59</td>
</tr>
<tr>
<td>34</td>
<td>Meshed single swirl diffuser</td>
<td>59</td>
</tr>
<tr>
<td>35</td>
<td>Computational domain and mesh system for single swirl diffuser</td>
<td>60</td>
</tr>
<tr>
<td>36</td>
<td>Schematic of modeling of single swirl diffuser</td>
<td>61</td>
</tr>
<tr>
<td>37</td>
<td>Air flow pattern in the hybrid under floor system</td>
<td>62</td>
</tr>
<tr>
<td>38</td>
<td>Velocity magnitude contour distribution at a selected slice</td>
<td>63</td>
</tr>
<tr>
<td>39</td>
<td>Clear zone</td>
<td>64</td>
</tr>
<tr>
<td>40</td>
<td>Temperature profile at a selected slice</td>
<td>65</td>
</tr>
</tbody>
</table>
Figure 41: Path lines released from the inlet of the swirl diffuser
Figure 42: Iso-surfaces at different temperature values
Figure 43: Temperature distributions at different heights
Figure 44: Schematic of the UFAD system of the BTLab
Figure 45: Parallel FLUENT® architecture
Figure 46: Computational domain and mesh system of the BTLab
Figure 47: Schematic view BTLab with thermal load
Figure 48: Temperature contour distribution
Figure 49: Path lines released from the inlet of the diffuser
Figure 50: Velocity magnitude at a selected slice
Figure 51: Velocity vector graph at a selected slice
Figure 52: Velocity magnitude as a function of x for different heights at the slice
Figure 53: Temperature as a function of x for different heights at the slice
Figure 54: Average temperature and velocity magnitude as a function of height
Figure 55: Geometry modeling with different spray angle of swirl diffuser
Figure 56: Velocity magnitude (m/s) contour at a selected slice
Figure 57: Path lines released from the bottom inlet of the swirl diffuser
Figure 58: Temperature iso-surfaces with different spray angle
Figure 59: Temperature (K) contour distributions at different heights (Z-plane)
Figure 60: Schematic of test grid in BTLab
Figure 61: Temperature and velocity magnitude comparison in test grid B3
Figure 62: Temperature and velocity magnitude comparison in test grid B4
Figure 63: Temperature and velocity magnitude comparison in test grid B5
Figure 64: Temperature and velocity magnitude comparison in test grid C3
Figure 65: Temperature and velocity magnitude comparison in test grid C4
Figure 66: Temperature and velocity magnitude comparison in test grid C5
ACKNOWLEDGEMENTS

I would like to express my deepest appreciation to Dr. Yitung Chen for his help and guidance in the completion of my thesis and throughout my graduate program. It has been very rewarding and satisfying experience to be able to work with a person of such vast knowledge and experience. I also thank my committee member, Dr. Huajun Chen for his assistance and timely guidance throughout my M.S program. I would also like to acknowledge Dr. Liangcai (Tom) Tan for providing experimental data. I would also like to thank my committee members, Dr. Boehm, Dr. Satish C. Bhatnagar for their support, patience, and effort in reviewing my thesis.

This thesis has been a challenging experience to me and was accomplished through the help of many people. Briefly mentioning their names does not completely express my gratitude towards them. I thank the Department of Energy for providing me financial support throughout the coursework and for providing timely support to finish my thesis. I would like to thank the students and staff at UNLV for all the help and support in completing this thesis.

Last but not the least, I would like to thank my family and friends, and all the other people whom I have not mentioned above but have helped me in some way through the completion of my master's degree.
CHAPTER 1

INTRODUCTION

1.1 Background

Over the past few decades HVAC systems have been applied widely to provide thermal comfort and maintain good indoor air quality (IAQ). These conditions are also essential for a quality, high performance environment. However, HVAC system design has been strongly influenced by increasing emphasis on IAQ, energy conservation, environmental effects, safety, and economics. The relative placement of system component can significantly affect the thermal comfort and energy performance of the air handling system. To design high energy efficiency with human comfort HVAC systems, it is necessary to gather the detailed information about the behaviors of the air flow in both the spaces and the rooms of the building.

It is necessary to get detailed and complete information, providing relevant information regarding velocity, pressure, temperature, concentration, and turbulence intensity throughout the domain of interest. The fundamental information concerning the flow comprises air velocity, temperature, relative humidity, and species concentrations which are important in assessing thermal comfort and indoor air quality. The conventional design of ventilation systems normally relies on valuable know-how, empirical formulas and past experience. Although practical knowledge and basic methods provide successful solutions, this type of engineering cannot take into account specific air
flow patterns which are affected not only by the positioning of openings and exhausts in a room, but by the distribution of objects and energy sources as well. Consequences related to the absence of these elements include over-designing and unnecessary cost.

In this project, a 3-D Computational Fluid Dynamics (CFD) analysis technique is used to simulate Ceiling Air Distribution (CAD) with variable air volume (VAV) and Under Floor Air Distribution (UFAD) system. The advantage of using 3-D computational fluid dynamics (CFD) analysis and design is that the numerical results can produce better, faster, and more economically useful information for the development of high energy efficiency with human comfort of CAD – VAV and UFAD systems and the design cycle can also be reduced drastically.

1.2 Basic Introduction

The beginning of the last century saw the origin of the heating, ventilating and air conditioning (HVAC) systems. Early in the latter half of the nineteenth century, W. H. Carrier [34] developed a system for humidity control. Since then, the HVAC system has become one of the most important building systems to provide thermal comfort environments.

The term “HVAC system” is used to refer the equipment that can provide heating, cooling, filtered outdoor air, and humidity control to maintain comfort conditions in a building or defined space. There are two types of HVAC systems, single zone system and multiple zone system. In a single zone system, there is a single air handling unit which can only serve more than one building area if the areas served have similar heating, cooling, and ventilation requirements. Areas regulated by a common control (e.g., a
single thermostat) are referred to as zones. A multiple zone systems can provide each zone with air at a different temperature by heating or cooling the air stream in each zone. The three HVAC functions, heating, ventilating, and air-conditioning, are closely interrelated [46]. All seek to provide thermal comfort, acceptable indoor air quality and reasonable installation, operation, and maintenance costs. The main purposes of HVAC systems are to provide thermal comfort and to maintain good IAQ which is essential for a quality, high performance environment for occupants.

According to ASHRAE Standard 55, human thermal comfort is the state of mind that expresses satisfaction with the surrounding environment. Indoor air quality is the content of interior air that could affect health and comfort of building occupants.

HVAC systems consist of two subsystems, CAD and UFAD systems. CAD system is ceiling air distribution system in which the cool air is delivered from the diffusers from the over head vents. The supply diffuser redirects the air blown out of the HVAC unit. There are different types of ceiling diffusers and one of them is a square cone diffuser which is used in this project.

(a) Front view [44]  
(b) Fixed to the ceiling [7]

Figure 1: Schematic of the square cone diffuser
Figure 1 shows the geometry of a square cone diffuser and Figure 2 shows the general pattern of the air which is diffused by the diffuser.

![Diagram of a square cone diffuser](image)

Figure 2: General pattern of the air from the diffuser [49].

Conventional HVAC systems are designed to promote complete mixing of supply air with room air, thereby maintaining the entire volume of air in the space (floor-to-ceiling) at the desired set point temperature and ensuring that an adequate supply of fresh outside air is delivered to the building occupants. Figure 3 shows the conventional CAD system which gives an idea how the air is distributed in the building.

![Diagram of a conventional ceiling air distribution system](image)

Figure 3: Conventional ceiling air distribution system [38].
Under Floor Air Distribution (UFAD) systems, in contrast to conventional air conditioning systems, deliver cool air from diffusers on the floor rather than from overhead vents. Figure 4 gives the geometry of the swirl diffuser [44] and the general pattern of the air from the swirl diffuser [49].

![Figure 4: Geometry and the general pattern of the air from the diffuser](image)

UFAD is a method of delivering space conditioning in commercial buildings that is increasingly being considered as a serious alternative to conventional ceiling-based air distribution systems because of the significant benefits that it can provide [36]. UFAD systems have several potential advantages over traditional overhead systems, including improved thermal comfort, improved indoor air quality, and reduced energy use. In contrast to the well-mixed room air conditions of the conventional overhead system, stratification of the UFAD systems is actually encouraged above head height where increased temperatures and higher levels of pollutants will not affect the occupants [36].

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
With UFAD systems, conditioned air from the air handling unit (AHU) is ducted into the under floor plenum where it typically flows freely to the supply outlets. UFAD systems are generally configured to have a relatively large number of smaller supply outlets, as compared to a conventional overhead system. Air is returned from the room at ceiling level. This produces an overall floor-to-ceiling air flow pattern that takes advantage of the natural buoyancy produced by heat sources in the building and more efficiently removes heat loads and contaminants from the space [36]. UFAD system space is divided into two zones, an occupied zone extending from the floor to head level, and an unoccupied zone extending from the top of the occupied zone to the ceiling. Under floor air distribution systems fall into two general categories in which the first type is a displacement ventilation system and the second type is hybrid underfloor systems which are distinguishable from one another by the temperature and velocity profiles they create in the occupied space [36].

In displacement ventilation systems air is delivered at floor level into the space at very low velocity, typically less than 50 feet per minute (fpm). At this velocity, the air
coming out of the diffuser can barely be felt. The system produces two distinct zones of air, one characterized by stratified layers of relatively cool and fresh air, the other by fairly uniform hot and stale air. The vertical flow profile in the lower zone can be generally described as upward laminar flow which is to displace the hot stale air into an area well above the breathing level of the occupants, giving occupants the benefit of breathing significantly higher-quality air. Figure 6 shows the displacement air flow in which the air is delivered from the floor in upward direction [37]. The thermal plume created by a heat source has the effect of enhancing the airflow around the source, thereby improving overall heat removal.

![Displacement ventilation airflow](image)

**Figure 6: Displacement ventilation airflow [48]**

The second general type of under floor air distribution system is the hybrid under floor system which can be called a combination of displacement ventilation and conventional mixing systems. Similar to the displacement ventilation system, even hybrid system attempts to condition the lower part of the space, producing two distinct zones of air, one cool and relatively fresh, the other hot and stale. But the hybrid underfloor system aims to reduce the stratification in the occupied lower portion by delivering air at
higher velocity (200 to 400 fpm) which is unlike the displacement ventilation system [36]. This results in a smaller temperature gradient and a more mixed and turbulent vertical low profile. Hybrid under floor systems can handle higher cooling loads than displacement ventilation systems [36]. While the hybrid under floor systems may more or less reduce the comfort problems associated with an excessive temperature gradient, they usually create small sub zones of excessive draft called “clear areas” that occupants need to avoid as shown in Figure 7.

Figure 7: Hybrid under floor air flow system [50].

The variable air volume system is one of the two types of air volume systems which maintain thermal comfort by varying the amount of heated or cooled air delivered to each space, rather than by changing the air temperature, whereas the constant volume systems, as their name suggests, generally deliver a constant airflow to each space [37]. CFD is a computer-based mathematical modeling tool which is accepted as the numerical solution, by computational methods, of the governing equations, which describe fluid flow, the Navier-Stokes equations, continuity and any additional conservation equations.
The increasing developments of computational fluid dynamics (CFD) in the recent years have opened the possibilities of low-cost yet effective method for improving HVAC systems in design phase, with less experimentation required.

Recent advances in computational fluid dynamics (CFD) and computer power make it possible to accurately predict the features of airflow within room. CFD models have been used to study IAQ problems, pollutant distributions, and performance of HVAC systems.

1.3 Objective

The research objectives of this project are:

- To predict optimal locations for different types of diffusers with variable air volume due to various thermal loads within a commercial building.
- To provide detailed and complete information regarding velocity, pressure, temperature, and turbulence intensity throughout the domain of interest.

1.4 Motivation

In the recent years, HVAC systems have been applied widely to provide thermal comfort and maintain good indoor air quality. In this project, a 3D CFD analysis technique has been used to design high energy efficiency and human comfort CAD with VAV and UFAD systems. Over past 10 years, there was large amount of work [2] [3] [6] [7] [8] [9] [11] [13] [14] [21] [24] [25] conducted on CFD analysis and numerical study of the air flow within room or space. Among them, most of the studies were on the CAD system rather than UFAD systems. The research to study UFAD systems for thermal
comfort and indoor air quality as a part of the project/program funded by Department of Energy through NATIONAL CENTER FOR ENERGY MANAGEMENT AND BUILDING TECHNOLOGIES (NCEMBT) gave motivation to this work.

1.5 Literature Review

Over the past few decades there were many studies and large amount of experimental work done on the CFD analysis and numerical simulations on the indoor environment. Though many studies were conducted on the CFD analysis in the room, this thesis focuses on the Computational Fluid Dynamics (CFD) analysis technique to design high energy efficiency and human comfort CAD-VAV and UFAD systems.

The main purposes of HVAC systems are to provide thermal comfort and to maintain good indoor air quality (IAQ). Bourhan Tashtoush, M. Molhim, M. Al-Rousan [14] describe a procedure for deriving a dynamic model of an HVAC system in particular; the interest is centered on control strategies to reduce energy consumption and improving the quality of the indoor environment. In this research project computational domains for CAD and UFAD systems are generated. Extensive literature reviews have been made on air flow simulations in a room, studies on diffusers, turbulent models, buoyancy flow in a room and general issues on numerical simulations in a room.

The research project is mainly divided into two parts, numerical simulation of CAD (ceiling air distribution) with VAV (variable air volume) system and numerical simulation of UFAD system.

CAD system is the ceiling air distribution system which is concerned with the distribution of air in the ceiling part of the room or space using four square cone diffusers.
Sun, and Smith [7] have presented a paper to examine the air flow characteristics of a room with square cone diffusers. Results show that offset and lips of the diffuser play an important role in air flow patterns. Juan Abanto, Daniel Barrero, Marcelo Reggio, Beno [8] conducted a study which concerns the numerical simulation of air flow and the prediction of comfort properties in a visualization room in which they concentrated on the four-way ceiling air supply diffuser.

To design high energy efficiency HVAC systems, it is necessary to get detailed and complete information, providing relevant information regarding velocity, pressure, temperature, concentration, and turbulence intensity throughout the domain of interest. A paper presented by S.L. Sinha, R.C. Arora, Subhransu Roy [26] deals with the velocity and temperature distribution in a room heated by a warm air stream introduced at various levels and finally presented that location of outlet at higher level than the inlet leads to better temperature distribution. J. Niu and J. van der Kooi [30] present two-dimensional numerical simulation results of the ventilation rates, indoor airflow fields and temperature distributions in an office room with open windows and auxiliary cooling devices. Based on the design supply air flow rate, the air flow inside the test space is turbulent. Lars Davidson [32] gives detailed information and explains all types of turbulent models. There are various types of turbulent models like zero-equation turbulence model, one-equation, two-equation models, large eddy simulation models. Bin Zhao, Xianting Li, Qisen Yan [3] published a paper with a simplified system for indoor airflow simulation based on the N-point air supply opening model, a zero-equation turbulence model and proved that the simplified methodology can predict indoor airflow quickly with satisfactory results. The work done by Guangyu Cao [33] showed that numerical
simulations of the prediction airflow distribution in an ordinary office room with some heat source, and computers with four different types of turbulent models.

In the past twenty years, various turbulence models have been proposed and among them the $k-\varepsilon$ model is the most popular of the two equation models and has produced qualitatively satisfactory results for a number of complex flows. Various experiments and numerical simulations were carried to obtain air flow patterns within a space. Qingyan Chen, Weiran Xu [4] used a new model with a new zero-equation model to predict natural convection, forced convection, mixed convection, and displacement ventilation in a room. The results agree reasonably well with experimental data and the results obtained by the standard $k-\varepsilon$ model. The paper which was presented by Youchen Fan [13] is about the $k-\varepsilon$ model in handling indoor air quality problem to simulate air flow patterns close to the boundaries of air and the stagnant component as well as the low air flow fluctuation elsewhere in a room. Shuzo Murakami, Shinsuke Kato’s [24] paper is concerned with the feasibility and validity of numerical simulation of room airflow. The results obtained from the numerical simulation are compared with model experiments concerned with velocity and diffusion fields. It may be concluded that 3-D numerical simulations using the $k-\varepsilon$ two equation model can predict turbulent recirculating flows in a ventilated room with sufficient accuracy from the viewpoint of engineering applications.

Improved developments of computational fluids dynamics (CFD) in recent years have opened the possibilities of low-cost yet effective methods for improving HVAC systems in the design phase, with less experimentation required. There has been considerable growth in the development and application of CFD to all aspects of fluid dynamics. In design and development, CFD programs are now considered to be standard
numerical tools, widely utilized within industry. Taeyeon Kima, Shinsuke Katob, Shuzo Murakamic, Ji-woong Rhod [1], [19] published two papers in which the first paper analyzed the performance of a cooling panel system installed in the vertical plane and finally concluded the cooling panel system was found to be very energy-efficient. And the second paper is about a computational fluid dynamics (CFD) simulation for analyzing indoor cooling/heating load which is coupled with a radiative heat transfer simulation and heating, ventilating, and air-conditioning (HVAC) control system in a room. In this paper, two types of HVAC systems are compared; i.e. radiation-panel system and all-air cooling system. This new method is able to analyze the indoor cooling load with changes of target thermal environments of a room and/or changing clothing conditions of occupants considering the temperature and air-velocity distribution in the room.

Posner, Buchanan, and Rankin [20] compare results from relatively simple 3-D numerical simulations with laser Doppler anemometry (LDA) and particle image velocimetry (PIV) experimental measurements of indoor air flows in a model room. Laminar, k-ε turbulence, and RNG k-ε turbulence numerical models are used. Results of the numerical simulations and velocimetry show obstructions can greatly influence the air flow and contaminant transport in a room. They finally concluded that RNG model most accurately predicts the flow in a portioned room, capturing the gross effects of a large flow obstruction.

CFD has almost become essential in every part of fluid dynamics. A paper published by Pietro Mazzei, Francesco Minichiello, Daniele Palma [18] deals with moisture control in buildings. So the dehumidification of the air is analyzed and there was a notable reduction of the power demand; a better control of ambient humidity.
Lu Lu, Wenjian Cai, Yeng Soh Chai, Lihua Xie [16], [17] presented two papers on the global optimization technologies for overall heating, ventilating and air conditioning (HVAC) systems. According to the characteristics of the operating components, the complicated original optimization problem for overall HVAC systems is transformed and simplified into a compact form ready for optimization. The second paper presents the solution for the global optimization problem for overall heating, ventilating and air conditioning (HVAC) systems using a modified genetic algorithm. Simulation studies for a pilot scale centralized HVAC plant by the proposed optimal method show that the proposed method indeed improves the system performance significantly compared with traditional control strategies. H. Xing, A. Hatton, H.B. Awbi [11] published a paper which is concerned with the difference in the air quality that is perceived by the occupants (breathing zone) and that existing in the occupied zone as a whole. CFD simulations were carried out for the purpose of flow visualization, the calculation of air velocity, temperature to study the effect of changing the air flow rate to the chamber and the position of air inlet to extend the range of parameters. The results from the CFD simulations were compared with those from measurements and good agreement was obtained in most cases. Guohui Gan, Hazim B. Awbi [25] developed the CFD program VORTEX which has been used for predicting the indoor environment in occupied spaces. The equations are solved for the 3-D Cartesian system using the SIMPLE algorithm. Results in the form of velocity vectors and contours for temperature, thermal comfort indices and CO₂ concentration are produced for the cases investigated.

M.M. Eftekhari, L.D. Marjanovic, D.J. Pinnock [5] has presented a paper to compare calculated and measured air flow distributions inside a test room for both winter
and summer. The predicted flow showed similar trends and the simulation results were in an agreement with the measured data.

One advantage of CFD modeling is that it allows specific entry details of a room that have relevant airflow. CFD models have been used to study indoor air quality (IAQ) problems, pollutant distributions, and performance of HVAC systems. K.W.D. Cheong, E. Djunaedy, T.K. Poh, K.W. Tham, S.C. Sekhar, N.H. Wong, M.B. Ullah [21] investigated the dispersion of contaminants in an office environment using empirical and modeling techniques and compared two layouts with a lower level of contaminant at the occupant’s breathing zones. BJarne W. Olesen, Makoto Koganei, G. Thomas Holbrook, James E.Woods [15] studied the effectiveness of a vertical displacement ventilation system was evaluated when contaminants were present. From the study they showed the air change effectiveness for the room varied from 126 to 145% and the contaminant removal effectiveness for the occupied zone varied from 80 to 700%.

Samirah Abdul Rahman, K.S.Kannan [6] used a CFD software called VORTEX as a tool to simulate air flow and thermal comfort in naturally wind ventilated classrooms of an educational institution and then recommendations will then be made on how to improve the ventilation of the least comfortable room, based on hypothetical simulation results.

CFD is now a widely accepted and validated engineering tool for industrial applications. Within the last few years, underfloor air distribution (UFAD) systems have become popular design alternatives to conventional air distribution (CAD) such as overhead air distribution systems for thermal and ventilation control.

Under Floor Air Distribution (UFAD) systems are originally introduced in the
In the 1950s in spaces having high heat loads (e.g., computer rooms, control centers, and laboratories), and subsequently introduced in office buildings in the 1970s, they have achieved considerable acceptance in Europe, South Africa and Japan for more than a decade. In UFAD systems, in contrast to ceiling air distribution systems, cool air is delivered from diffusers in the floor rather than from overhead vents.

UFAD systems can have significant impacts on room air stratification and thermal comfort in the occupied zone [36]. Halza [37] introduced the advantages of the UFAD system: improved air quality, lower life-cycle costs, as well as overhead system: better comfort, lower capital cost. Y.J.P. Lin, P.F. Linden [2] developed a simplified model of an underfloor air distribution (UFAD) system consisting of a single source of heat and a single cooling diffuser in a ventilated space and conducted laboratory experiments to simulate the flow.

1.6 Thesis Outline

This thesis presents a numerical solution of the HVAC design based on the dimensions of BTLab at UNLV using computational fluid dynamics (CFD) technique. Chapter 2 gives the details of the description of the problem, shows the schematic view. Chapter 3 describes the numerical method for solving the problem, governing equations and methodology of the project. Chapters 4 and 5 explain the CFD analysis and numerical methodology on the ceiling air distribution (CAD) system and Under Floor Air Distribution (UFAD) system. Results were presented and benchmark the result with the experimental work.
CHAPTER 2

PROBLEM DESCRIPTION AND METHODOLOGY

2.1 Problem Description

In this chapter the physical problem of the project is discussed. The numerical simulation of the ceiling air distribution (CAD) system and the Under Floor Air Distribution (UFAD) system based on the dimensions of BTLab at UNLV were studied using computational fluid dynamics (CFD) technique.

The CAD system of BTLab includes four square cone diffusers with four exhaust fans. The UFAD system has eight swirl diffusers and sixteen heaters and four exhaust fans. The results for the velocity and temperature of the air flow in the test space of BTLab for CAD and UFAD system have been analyzed.

The fundamental information about the UFAD experiment system in UNLV’s BTLab is collected. The analysis on the information of experiment system has been made and a simplified geometry of BTLab for UFAD system simulation was generated. The initial computational domain data for simulating the CAD and UFAD systems of the BTLab is shown are Figure 8.

Parameters for test space of the BTLab are

- Space length = 9.1 m
- Space width = 6.1 m
• Ceiling height = 2.7 m
• Space floor area = 55.7 m$^2$
• Air diffusion performance index $>80$
• South wall surface temperature, $T_{sw} = 297.05$ K
• North wall surface temperature, $T_{nw} = 297.05$ K
• East wall surface temperature, $T_{ew} = 297.05$ K
• West wall surface temperature, $T_{ww} = 297.05$ K

![Figure 8: Computational domain of the BT Lab](image)

Boundary conditions for CAD and UFAD systems will be presented in the next chapters.

For this numerical study, it is really important to know whether the result is reasonable or not. The experimental phase of this project has been conducted by UNLV with which the results were compared. The experimental data are important for comparing the numerical results and vice versa. The numerical study of this project is necessary because the optical location of the diffusers can be predicted; it can be studied...
at different operating conditions which were difficult with the experimental setup as it is an expensive and time consuming process.

2.2 Experimental Setup

The University of Nevada Las Vegas (UNLV) is conducting the laboratory phase of this task and is in charge by Dr. Liangcai (Tom) Tan.

This experimental task will compare the performance and cost-effectiveness of UFAD and CAD systems. Information regarding the CAD and UFAD systems will be developed to determine under what circumstances UFAD systems should be specified. The actual applications of UFAD systems in commercial buildings have been assessed and analytical tools for evaluation and quantification of UFAD performance have been developed. A test protocol have been developed and implemented in the UNLV Center for Mechanical & Environmental Systems Technology (CMEST) laboratory test room, known as the BTLab, to test performance of UFAD systems compared to CAD systems, including comfort, energy use, IAQ.

Figure 9: Air Ducts of the BTLab
Figure 10: Condenser and Piping of the Daikin VRV System in the BTLab

(a) Square cone diffuser  (b) Swirl diffuser

Figure 11: Diffusers used in the BTLab

Air duct system, piping of the Daikin VRV system and the condenser used in the BTLab are shown in Figures 9 and 10. Figure 11 shows the square cone and swirl diffusers used in BTLab. The goals of this experimental project are to investigate HVAC design, comfort, and energy issues related to UFAD systems and to develop information that will provide the foundation for future development of design tools that can be used to
compare UFAD and CAD system applications. Figure 12 shows the traversing mechanism and sensor's pole and the data acquisition board and central control computer in the BTLab.

Stratification has not been observed in several installed UFAD systems; rather the systems appear to perform similarly to a CAD system. Therefore, the UFAD task force of the National Center of Energy Management and Building Technology (NCEMBT) recommended that the initial experiments of this task is to compare UFAD and CAD systems and specifically determine if stratification is present or not. The comparison will be made with both systems having been designed with an air distribution performance index (ADPI) of greater than 80%. The design of experiment (DoE) is based on the (null) hypothesis that "There are no significant differences in air velocity and thermal
distribution in interior office zones between CAD and UFAD systems that are designed to perform at an ADPI greater than 80%.” The null hypothesis implies that there is no thermal stratification in the UFAD system. Thus, the existence of stratification has been verified or disproved by this experiment.

The objective of the DoE is to compare a CAD and a UFAD system that are designed to the same performance level. Thus, the first challenge is to design a UFAD system that will result in an ADPI equal to or greater than 80%. Towards that goal the BTLab will be equipped to provide a full-scale simulation of a typical interior office zone. Both the CAD and UFAD systems have been tested in the same space. The experiment has been conducted based on ASHRAE Standard 113-1990 - Method of Testing for Room Air Diffusion.

2.3 Methodology

2.3.1 Software

Computational Fluid Dynamics (CFD) has grown from a mathematical curiosity to become an essential tool in almost every branch of fluid dynamics. CFD is the science of determining a numerical solution to the governing equations of fluid flow while advancing the solution through space or time to obtain a numerical description of the complete flow field of interest. CFD is a sophisticated analysis technique. It not only predicts fluid flow behavior, but also the transfer of heat, mass (such as in perspiration or dissolution), phase change (such as in freezing or boiling), chemical reaction (such as combustion), mechanical movement (such as an impeller turning), and stress or deformation of related solid structures. As a developing science, it has received extensive
attention throughout the international community since the advent of the personal
computer. There has been considerable growth in the development and application of
CFD to all aspects of fluid dynamics. In design and development, CFD programs are now
considered to be standard numerical tools, widely utilized within industry. As a
consequence there is a considerable demand for specialists in the subject, to apply and
develop CFD methods throughout engineering companies and research organizations.

CFD analysis shows parts of the system or phenomena happening within the system that
would not otherwise be visible through any other means. CFD gives a means of
visualizing and enhanced understanding of the designs. The foresight gained from CFD
helps to design better and faster. Better and faster design or analysis leads to shorter
design cycles. Time and money are saved. CFD is a tool for compressing the design and
development cycle.

FLUENT® 6.2 is a computational fluid dynamics (CFD) software package to
simulate fluid flow problems. FLUENT® is a state-of-the-art computer program for
modeling fluid flow and heat transfer in complex geometries. The FLUENT® package
includes FLUENT®, and GAMBIT® (the preprocessor for geometry modeling and
mesh generation). Over the last few years, many commercial CFD packages have become
available. Almost every industry that involves advanced engineering uses CFD. Its use is
rapidly expanding, and FLUENT® 's CFD software features accuracy, efficient meshing,
high speed and powerful visualization capability. FLUENT® is used to perform
simulations in this study. GAMBIT® is FLUENT®'s geometry and mesh generation
software. GAMBIT®'s single interface for geometry creation and meshing brings
together most of FLUENT®'s preprocessing technologies in one environment.
GAMBIT®'s combination of CAD interoperability, geometry cleanup, decomposition and meshing tools results in one of the easiest, fastest, and most straightforward preprocessing paths from CAD to quality CFD meshes. As a state-of-the-art preprocessor for engineering analysis, GAMBIT® has several geometry and meshing tools in a powerful, flexible, tightly-integrated, and easy-to-use interface. GAMBIT® can dramatically reduce preprocessing times for many applications. Most models can be built directly within GAMBIT®'s solid geometry modeler, or imported from any major CAD/CAE system. GAMBIT® also has an excellent boundary layer mesher for growing optimum grid cells off wall surfaces in the geometries for fluid flow simulation purposes.

2.3.2 Governing Equations

Basically indoor airflow can be viewed as turbulent flow. In turbulent flows almost all fluid physical parameters fluctuate and interact with each other over time and across space. In order to study the turbulent mixing on thermal comfort and indoor air quality, governing partial differential equations must be solved. The indoor air flow can be viewed as the incompressible turbulent flow and the numerical governing equations of air flow are based on the mass conservation, momentum conservation and energy conservation equations. These three conservation equations can illustrate the necessity of each parameter and show how they are combined to describe the total flow field.

Continuity equation:
\[
\frac{\partial \rho U_j}{\partial x_j} = 0 \quad (2.1)
\]

Momentum equation:
\[
\frac{\partial}{\partial x_j}(\rho U_j U_j) = -\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_i}\left[\mu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \rho u_i' u_j'\right] - \rho g_i \quad (2.2)
\]

Energy equation:
\[
\frac{\partial}{\partial x_j} \left( \rho U_j C_p T \right) = \frac{\partial}{\partial x_j} \left( \lambda \frac{\partial T}{\partial x_j} - \rho u'_j T' \right)
\]

where \( \rho \) is the density, \( \mu \) is the viscosity, \( P \) is the pressure, \( C_p \) is the specific heat capacity, \( \lambda \) is the thermal conductivity, \( u'_i u'_j \) and \( u'_j T' \) are the turbulent stress and heat flux, \( g_i \) is the gravitational vector in the \( i \)th direction. The turbulent stress and heat flux are determined by

\[
\rho u'_i u'_j = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho \delta_{ij} k
\]

\[
\rho u'_j T = \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_j}
\]

where \( \delta_{ij} \) is the Kronecker delta function, \( \delta_{ij} = 1 \) when \( i = j \) and zero when \( i \neq j \), \( k \) is the turbulent kinetic energy, \( Pr_t \) is the turbulent Prandtl number and taken as 0.9 in this study, and \( \mu_t \) is the turbulent viscosity, \( \mu_t = \rho C_{\mu} k^2 / \varepsilon \), where \( C_{\mu} = 0.09 \) and \( \varepsilon \) is the turbulence dissipation.

The above equations are difficult to solve directly for turbulent flows due to the practical complex natures of turbulence in theory and available computational capacity restrictions in practice. Therefore a statistical approach was used in 1890's and forms the basis of the turbulence models will be presented.

2.3.3 Standard k-\( \varepsilon \) Model

Turbulent flows are characterized by fluctuating velocity fields. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations.
that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities. There is no definition on turbulent flow, but it has a number of characteristic features such as [32]:

- **Irregularity:** Turbulent flow is irregular, random and chaotic. The flow consists of a spectrum of different scales (eddy sizes) where largest eddies are of the order of the flow geometry (i.e. boundary layer thickness, jet width, etc). At the other end of the spectra are the smallest eddies which are by viscous forces (stresses) dissipated into internal energy. Even though turbulence is chaotic it is deterministic and is described by the Navier-Stokes equations.

- **Diffusivity:** In turbulent flow the diffusivity increases. This means that the spreading rate of boundary layers, jets, etc. increases as the flow becomes turbulent. The turbulence increases the exchange of momentum in e.g. boundary layers and reduces or delays thereby separation at bluff bodies such as cylinders, airfoils and cars. The increased diffusivity also increases the resistance (wall friction) in internal flows such as in channels and pipes.

- **Large Reynolds numbers:** Turbulent flow occurs at high Reynolds number. For example, the transition to turbulent flow in pipes occurs that $\text{Re}_D \approx 2,300$ and in boundary layers at $\text{Re}_x \approx 100,000$

- **Three dimensional:** Turbulent flow is always three-dimensional. However, when the equations are time averaged the flow is treated as two-dimensional.

- **Dissipation:** Turbulent flow is dissipative, which means that kinetic energy in the small (dissipative) eddies are transformed into internal energy. The small eddies
receive the kinetic energy from slightly larger eddies. The slightly larger eddies receive their energy from even larger eddies and so on. The largest eddies extract their energy from the mean flow. This process of transferred energy from the largest turbulent scales (eddies) to the smallest is called cascade process.

- **Continuum**: Even though there are small turbulent scales in the flow, they are much larger than the molecular scale and the flow can be treated as a continuum.

In turbulent flow the variables are usually divided as one time-averaged part, which is independent of time (when the mean flow is steady), and one fluctuating part \( u' \), so that \( U = \bar{U} + u' \). One reason to decompose the variables is that the measure flow quantities are usually interested in the mean values rather that the time histories. Another reason is to solve the Navier-Stokes equation numerically it would require a very fine grid to resolve all turbulent scales and it would also require a fine resolution in time (turbulent flow is always unsteady). It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. 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.

The following choices of turbulence models:

- **Spalart-Allmaras model**
- **\( k-\varepsilon \) models**:
  - Standard \( k-\varepsilon \) model
  - Renormalization-group (RNG) \( k-\varepsilon \) model
  - Realizable \( k-\varepsilon \) model
• k-ω models:
  - Standard k-ω model
  - Shear-stress transport (SST) k-ω model

• \( V^2 - f \) model

• Reynolds stress model (RSM)

• Detached eddy simulation (DES) model

• Large eddy simulation (LES) model

Various turbulence models have been proposed in the past twenty years. The \( k-\epsilon \) model is the most popular and the simplest model of the two equation models and produced qualitatively satisfactory results for a number of complex flows and among them standard \( k-\epsilon \) model has been used in this project work. Even other models like realizable \( k-\epsilon \) model can also be used but standard \( k-\epsilon \) model requires less computational effort and memory and is sufficient for the present work. This technique uses wall functions to treat the near-wall sub-layers. The simplest models of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard \( k-\epsilon \) model in falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations. The most popular model is the standard \( k-\epsilon \) turbulent model. The standard \( k-\epsilon \) turbulent model is a semi-empirical model based on the model transport equations for the turbulent kinetic energy \( k \) and its dissipation rate \( \epsilon \). The model transport equation for \( k \) is derived from an exact equation, while the model transport equation for \( \epsilon \) was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. In the derivation of the standard \( k-\epsilon \)
model, it was assumed that the flow was fully turbulent. In the derivation of the k-ε model, it was assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k-ε model is therefore valid only for fully turbulent flows.

Transport Equations for the Standard k-ε Model: The turbulence kinetic energy, k, and its rate of dissipation, $\varepsilon$, are obtained from the following transport equations

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho ku_i) = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (2.6)$$

and

$$\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_t}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1e} \frac{\varepsilon}{k} (G_k + C_3 e G_b) - C_{2e} \rho \varepsilon^2 + S_\varepsilon \quad (2.7)$$

In these equations, $\mu_t$ is the turbulent viscosity, $G_k$ represents the generation of turbulence kinetic energy due to the mean velocity gradients, calculated as

$$G_k = -\rho u'_i u'_j \frac{\partial u_j}{\partial x_i} \quad (2.8)$$

$G_b$ is the generation of turbulence kinetic energy due to buoyancy, calculated as

$$G_b = \beta g_i \frac{\mu_t}{\Pr_t} \frac{\partial T}{\partial x_i} \quad \text{where} \quad \beta = -\frac{1}{\rho} \left( \frac{\partial \rho}{\partial T} \right) \quad (2.9)$$

where $\beta$ is the coefficient of thermal expansion, $\Pr_t$ is the turbulent Prandtl number for energy and $g_i$ is the component of the gravitational vector in the $i$th direction. For the standard k-ε models, the value of $\Pr_t$ is 0.85.

$Y_M$ represents the contribution of the fluctuating dilatation in compressible
turbulence to the overall dissipation rate. For high-Mach-number flows, compressibility affects turbulence through ‘dilatation dissipation’, which is normally neglected in the modeling of incompressible flows. In this model, the flow is considered to be incompressible. C_{1e}, C_{2e}, and C_{3e} are constants. \sigma_k and \sigma_e are the turbulent Prandtl numbers for k and \varepsilon, respectively. S_k and S_e are user-defined source terms.

Modeling the Turbulent Viscosity: The turbulent (or eddy) viscosity, \( \mu_t \), is computed by combining k and \( \varepsilon \) as follows:

\[
\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}
\]  

(2.10)

where \( C_\mu \) is a constant. The model constants \( C_{1e}, C_{2e}, C_{3e}, \sigma_k \) and \( \sigma_e \) have the following default values:

\( C_{1e} = 1.44, \ C_{2e} = 1.92, \ C_\mu = 0.09, \ \sigma_k = 1.0 \) and \( \sigma_e = 1.3 \)

These default values have been determined from experiments with air and water for fundamental turbulent shear flows including homogeneous shear flows and decaying isotropic grid turbulence. They have been found to work fairly well for a wide range of wall-bounded and free shear flows.

2.4 Numerical Procedure

The general purpose CFD code FLUENT® is used as a numerical solver for the present three-dimensional simulation. A non-staggered grid storage scheme is adapted to define the discrete control volumes. In this scheme the same control volume is employed
for the integration of all conservation equations and all variables are stored at the control
cell's cell center. The numerical scheme used in this study is a power law
differencing scheme and the solver used is a segregated solver. The segregated solver is
the solution algorithm. Using this approach, the governing equations are solved
sequentially (i.e., segregated from one another). Because the governing equations are
non-linear (and coupled), several iterations of the solution loop must be performed before
a converged solution is obtained. Each iteration consists of the steps outlined below:

1. Fluid properties are updated, based on the current solution. (If the calculation has just
begun, the fluid properties will be set based on the initialized solution.)

2. The u, v, and w momentum equations are each solved in turn using current values for
pressure and face mass fluxes, in order to update the velocity field.

3. Since the velocities obtained in Step 2 may not satisfy the continuity equation locally,
a "Poisson-type" equation for the pressure correction is derived from the continuity
equation and the linearized momentum equations. 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.

4. Where appropriate, equations for scalars such as turbulence, energy, species, and
radiation are solved using the previously updated values of the other variables.

5. When interphase coupling is to be included, the source terms in the appropriate
continuous phase equations may be updated with a discrete phase trajectory calculation.

6. A check for convergence of the equation set is made.

These steps are continued until the convergence criteria are met. The SIMPLE algorithm
is used to resolve the coupling between pressure and velocity. The governing equations,
discrete and nonlinear, are linearized using an implicit technique with respect to set of dependent variables. The algebraic equations are solved iteratively using an additive correction multi-grid method with the Gauss-Seidel relaxation procedure.

Finite volume method: In order to convert the differential equations into algebraic equations that can be solved numerically, some discretization strategies must be employed. The most common method is finite volume method. The commercial FLUENT® 6.2 software is in this numerical simulation, in which the discretization method is the finite volume method (FVM). The finite volume method divides space into finite number of control volumes. The transport equations are then applied to each of the control volumes, integrating the governing equations and yielding discrete equations that conserve each quantity on a control-volume basis. This method can employ the advanced grid generation techniques such as unstructured grid and body-fitted coordinates, and has become the most widely used discretization method. The FVM is a numerical method for solving partial differential equations (PDE) that calculates the values of the conserved variables averaged across the volume. The FVM uses the integral form of the conservation equations as its starting point. Assume $\phi$ is a generic conserved intensive property (for mass conservation, $\phi = \rho$; for momentum conservation, $\phi = \vec{u}$; for conservation of a scalar, $\phi$ represents the conserved property per unit mass), then the integral form of the generic conservation equation can be expressed as:

$$\frac{\partial}{\partial t} \int_{\Omega} \rho \phi d\Omega + \int_{\partial \Omega} \rho \phi \vec{u} \cdot \vec{n} dS = \int_{\Gamma} \Gamma u_{\phi} \partial \phi \cdot \vec{n} dS + \int_{\Omega} q_{\phi} d\Omega$$

(2.11)

For simplicity, steady-state is considered. Assuming the velocity field and all fluid properties are known, and then the generic conservation equation becomes:
\[ \int_S \rho \phi \overline{u} \cdot \overline{n} dS = \int_S \overline{\nabla \phi} \cdot \overline{n} dS + \int_{\Omega} q_\phi d\Omega \]  

(2.12)

The solution domain is subdivided into a finite number of contiguous control volumes (CVs) by a grid which defines the control volume boundaries, and the conservation equations are applied to each CV. At the centroid of each CV lies a computational node at which the variable values are to be calculated. Interpolation is used to express variables at the CV surface in terms of the CV center values. Typical 2D Cartesian control volumes with the notations are shown in Figure 13. For maintenance of conservation, it is important that CVs do not overlap; each CV face is unique to the two CVs which lie on either side of it.

![Figure 13: A typical control volume and the notation used for a cartesian 2D grid](image)

A typical 2D CV is taken as an example. Its surface is subdivided into four plane faces \((S_e, S_w, S_s, \text{ and } S_n)\) on which ‘e’, ‘w’, ‘s’ and ‘n’ represent the center of the surface; ‘ne’, ‘nw’, ‘sw’ and ‘se’ represent the corner nodes and P is the centroid. Thus, the net flux through the CV boundary is the sum of integrals over the four CV surfaces:
\[ \int_S f dS = \sum_k \int_k f dS \]  

(2.13)

Where, \( f \) is the component of the convective (\( \rho \phi \vec{V} \cdot \vec{n} \)) or diffusive (\( \Gamma \nabla \phi \cdot \vec{n} \)) vector in the direction normal to CV surface. As the velocity field and all fluid properties are known, thus quantity \( \phi \) is the only unknown in \( f \).

A typical CV face \( (S_e) \) is taken as an example to calculate the surface Eq. 2.13, the analogous expressions may be derived for all faces by making appropriate index substitutions. To calculate the surface integral on \( S_e \), an approximation must be introduced because only the CV center values of \( f \) are known. Approximation is best done using two levels: a) The integral is approximated in terms of the variable values at one or more locations on the cell face; b) The cell face values are approximated in terms of the CV center values.

The simplest approximation to the integral is the midpoint rule that can be expressed as:

\[ F_e = \int_{S_e} f dS \approx f_e S_e \]  

(2.14)

Where \( f_e \) is the value of \( f \) at the center of face \( S_e \).

Another second order approximation of the surface integral in 2D is the trapezoid rule, which leads to:

\[ F_e = \int_{S_e} f dS \approx \frac{S_e}{2} (f_{ne} + f_{se}) \]  

(2.15)

In this case the integrand at the CV corners are needed. For higher order approximation, the integrand is needed at more than two locations. For example, using the Simpson's rule, a fourth-order approximation of the integral over \( S_e \) is estimated as:
Here the values of $f$ are needed at three locations: the cell face center and the two corners. As the values of $f$ is unknown on $S_e$, the values of $f_e$, $f_{ne}$ and $f_{se}$ have to be expressed in terms of the values of CV centers by interpolation (if velocity field and other properties are known, then only the values of quantity $\phi$ needs to be approximated). Many interpolation methods have been developed, several of which widely used such as: The upwind interpolation, linear interpolation, quadratic upwind interpolation (QUICK) are used in this thesis.

(1) The upwind interpolation is also named as upwind differencing scheme (UDS), in which the value of $\phi$ on cell face, for example $\phi_e$ on $S_e$, is approximated by its value at the node upstream of ‘e’ using a backward or forward difference approximation for the first derivative:

$$
\phi_e = \begin{cases} 
\phi_p, & \text{if } (\nabla \cdot \bar{n})_e > 0 \\
\phi_E, & \text{if } (\nabla \cdot \bar{n})_e < 0 
\end{cases}
$$

(2.17)

The UDS approximation is a first order scheme because it retains only the first term of the Taylor series expansion. It will never be oscillatory, but it is numerically diffusive as its leading truncation error term is diffusive.

(2) Linear interpolation is another straightforward approximation for the value at CV face center between two nearest nodes. For example, at location ‘e’, $\phi_e$ is approximated by:

$$
\phi_e = \phi_E \lambda_E + \phi_P (1 - \lambda_E)
$$

(2.18)
Where the linear interpolation factor is defined as:

\[
\lambda_e = \frac{x_e - x_P}{x_E - x_P}
\]  

(2.19)

The linear interpolation retains the first two terms of Taylor series expansion. It is the simple second order scheme and is the one most widely used.

(3) The quadratically \textit{upwind interpolation}, also named as quadratic upwind interpolation for convective kinematics (QUICK), approximates the variable profile between P and E by a parabola instead of a straight line as in linear interpolation. To construct a parabola, data at one more point are needed, in accord with the nature of convection, the third point is taken on the upstream side, i.e. W if flow is from P to E (i.e. \( u_x > 0 \)) or EE if \( u_x < 0 \).

Thus:

\[
\phi_e = \begin{cases} 
  g_1 \phi_E - g_2 \phi_W + (1 - g_1 + g_2) \phi_P, & \text{for } u_x > 0 \\
  g_3 \phi_P - g_4 \phi_{EE} + (1 - g_3 + g_4) \phi_E, & \text{for } u_x < 0
\end{cases}
\]  

(2.20)

Where the coefficient \( g_i \) can be expressed in terms of the interpolation factors:

\[
g_1 = \frac{(2 - \lambda_{e,W}) \lambda_{e,P}^2}{1 + \lambda_{e,P} - \lambda_{e,W}}
\]  

(2.21)

\[
g_2 = \frac{(1 - \lambda_{e,P})(1 - \lambda_{e,W})^2}{1 + \lambda_{e,P} - \lambda_{e,W}}
\]  

(2.22)

\[
g_3 = \frac{(1 + \lambda_{e,W})(1 - \lambda_{e,P})^2}{1 + \lambda_{e,E} - \lambda_{e,P}}
\]  

(2.23)

\[
g_4 = \frac{\lambda_{e,P} \lambda_{e,P}^2}{1 + \lambda_{e,E} - \lambda_{e,P}}
\]  

(2.24)

This quadratic interpolation has a third order truncation error on both uniform and
non-uniform grids. Some terms in the transport equations require integration over the volume of a CV. The simplest second-order accurate approximation is to replace the volume integral by the product of the mean value and the CV volume:

$$Q_p = \int_Q q\,d\Omega \approx q_p \Delta\Omega$$  \hspace{1cm} (2.25)

where $q_p$ is for the value of q at the CV center. Since all variables are available at node P, this quantity is easily calculated, no interpolation is necessary.

An approximation of higher order requires the values of q at more locations than just the center. These values have to be obtained by interpolating nodal values or equivalently, by using shape function. Fluxes through CV faces coinciding with the domain boundary require special treatment, which must be either known or be expressed as a combination of interior values and boundary data. Since there are no nodes outside the boundary, these approximations must be based on one-sided differences or extrapolations.

One advantage of the FVM over finite difference methods (FDM) is that it does not require a structured mesh (grids are based on some regular distribution of the nodes), although a structured mesh can also be used. Furthermore, the FVM is preferable to other methods as a result of the fact that boundary conditions can be applied non-invasively because the values of the conserved variables are located within the volume element, and not at nodes or surfaces. The FVM is especially powerful on coarse non-uniform grids and in calculations where the mesh moves to track interfaces or shocks.
CHAPTER 3

NUMERICAL MODELING OF THE CAD (CEILING AIR DISTRIBUTION) SYSTEM

This chapter will present the numerical modeling for the CAD system with single and four square cone diffusers. The governing equations for combined turbulent convection heat transfer in the four-way diffuser are solved in the Cartesian coordinate system with a control-volume finite difference method. The general purpose CFD code FLUENT® is used as a numerical solver for the present three-dimensional simulation. Numerical methods are used to solve the governing integral equations for the conservation of mass and momentum, and energy. The segregated solver is used to solve the governing equations sequentially. The discrete and nonlinear governing equations are linearized using an implicit technique with respect to set of dependent variables. In this scheme the same control volume is employed for the integration of all conservation equations and all variables are stored at the control volume’s cell center. The segregated approach solves for a single variable field by considering all cells at the same time. It then solves for the next variable field by again considering all cells at the same time. The control-volume-based technique is used in segregated solver which divides the domain into discrete control volumes using a computational grid. Integration of the governing equations on the individual control volumes constructs algebraic equations for the discrete dependent variables such as velocities, pressure, and temperature.
Figure 14 shows the geometry and computational mesh system for the four-way square cone diffuser. The neck of the diffuser is 1.22 m in height. The width of bottom for the four-way diffuser is 0.61 m. In Figure 15, the angle at which the vanes is called as
lip angle and the vertical projection of the lips to the ceiling is the offset. The air flow patterns differ when the lip angle and the offset varies. To model the diffuser, the lip angle is taken as $18^0$ and offset is considered as zero. The offset and lip angle work together to determine the air flow patterns near the diffuser region. When the lip angle is less, the air flows more horizontal to the ceiling and as the lip angle increases, the air is directed downwards according to the lip angle.

3.1 CFD Analysis on the Single Diffuser Characteristics

The four-way diffuser located in the ceiling plays a key role in air flow distribution. To study the effects of the diffuser characteristics on the air flow patterns in the near diffuser region, a diffuser model is established. To simulate the small scale air flow inside the diffuser, 37,519 tetrahedron cells was used to model the diffuser.

Figure 16: Computational domain for the single diffuser study
The domain extents are as follows:

- x-coordinate: min (m) = -2.133600e+000, max (m) = 2.133600e+000
- y-coordinate: min (m) = -2.133600e+000, max (m) = 2.133600e+000
- z-coordinate: min (m) = -2.743200e+000, max (m) = 9.372600e-001

Volume statistics:
- Minimum volume (m³): 7.106405e-008
- Maximum volume (m³): 2.596498e-004
- Total volume (m³): 5.014861e+001

Face area statistics:
- Minimum face area (m²): 2.486274e-005
- Maximum face area (m²): 9.102224e-003

Boundary conditions are:

- The supply air flow for the single diffuser is 0.098 m³/s
- Temperature of the supply air flow is 285.95 K (12.8°C)
- Environment temperature is 300.05 K (26.9°C).

For the whole computational domain 724,351 tetrahedral cells are used. In the computational domain, the top surface, which connects the diffuser, is considered to be a wall and the other surfaces of computational domain are treated as pressure-outlet.

Figure 17 shows the velocity vector graph at the selected slice. Air has a strong vertical downward flow due to the buoyancy effect, as the temperature of supply air flow is lower than the environment temperature. Figure 18 gives the variation of velocity magnitude as a function of X at a selected slice (Y=0). From that Figure it is shown that the air from the diffuser affects in the range of -0.61 < X < 0.61 m.
Beyond 0.61 m it shows that the air is stagnant which is shown in Figure 18. The velocity magnitude of the points below the diffuser decreases as the height (Z value) increases.

Figure 17: Velocity vector graph at the center of the domain (X=0m)

Figure 19 gives the velocity magnitude profile of the single diffuser at a center of the domain (X=0). The flow of the air from the diffuser is almost horizontal to the ceiling as the lip angle of the diffuser is 18°. The larger the lip angle, the more horizontal will be the air flow to the ceiling.
Figure 18: Variation of velocity magnitude as a function of X at a selected slice (Y=0m)

Figure 19: Velocity magnitude at center of the domain (X=0m)
Figure 20: Temperature profile at the center of the domain (X=0m)

Figure 20 gives the temperature profile of the single diffuser. Figure 22 shows the path lines which are released from the inlet of the square cone diffuser which is colored by temperature. 225 path lines are tracked. These path lines are similar to the result of the temperature profile as in Figure 20. Similar to the variation of the velocity magnitude with X, the air temperature in the range -0.61 m < X< 0.61 m is influenced by significantly the diffuser. Figure 21 also shows that when the height is smaller than -0.61m (considering the center of the height is 0), air temperature almost has the same value with that of environment.
Figure 21: Variation of air temperature as a function of $X$ at a selected slice ($Y=0$)

Figure 22: Path lines released from the top inlet of the diffuser, colored by the temperature.
3.2 CFD Analysis on the CAD System of BTLab

The computational grid system and domain for the test space of BTLab is shown in Figure 23. The different sizes of computational mesh with 438,472 cells, 569,722 cells, and 876,482 cells have been used to check the grid independence of the numerical solution.

All the three cases are executed and results are almost similar. In order to save time and memory space, and since there is no big difference between the loose mesh and fine mesh, the results shown here is based on the computational mesh with 438,472 cells.

![Domain and computational grid system of the test space of BTLab with four square cone diffusers.](image)

Figure 23: Domain and computational grid system of the test space of BTLab with four square cone diffusers.

For the computational domain, the bottom and top walls are treated as adiabatic in this study. The constant temperature with 297.05 K (23.9°C) is taken on the side wall. The supply air flow for the each diffuser is 0.098 m³/s. When compared to a single diffuser, the path lines in the test space of BTLab with four diffusers is much more complicated as the air streams interact with each other when they flow out of the
diffusers. And path lines are also affected by the buoyancy force due to the high temperature on side wall. To check the flow behavior of the diffuser, the path lines released from the top inlet of diffuser are shown in Figure 24 which is colored by the air temperature. Figure 24 also indicates that the air inside the test space is also well mixed by the diffuser and the temperature in the region below the diffuser is quite uniform. Figure 25 shows the velocity vector graph at the selected slice (Y= 1.52 m). From the zoom view of A-A’, it is shown that two vortices can be found in the region below the diffuser. Since the side wall temperature is higher than that of supply air flow, the upward air flow can be found in the region near the wall. Figure 26 also shows the temperature distribution at the selected slice (Y = 1.52 m). Witness that the temperature distribution is quite uniform since the cold supply air is well mixed. Figure 27 illustrates the variation of average surface air temperature and velocity magnitude as a function of height. From Figure 27, the variation of the average surface air temperature with height is less than 1 K. The average temperature for the whole test space is about 289.05 K and almost 8 K less than the wall temperature. The curves for the average velocity magnitude show that the average air velocity magnitude is less than 0.1 m/s and increases as Z increases.
Figure 24: Path lines released from the top inlet of the diffuser. The path line is colored by the temperature. A total of 320 path lines are tracked.
Figure 25: Velocity vector graph at a selected slice
To guide the experimental study, the velocity and temperature at the test point have been studied. Figure 28 shows the test positions with temperature and velocity sensors. In the experiment, three test planes with 13 test columns are considered. Considering the symmetry of the geometry, the first six test column in test plane 1 and test plane 2 only were studied.
Figure 28: Test positions with temperature and velocity sensors at BTLab

Figure 29 shows the variation of air velocity magnitude at the test positions as a function of height. In the test plane 1, it can be found that the velocity magnitude is much larger than the other points for height, $Z > -0.305$ m since these points are near the diffuser. At the points $Z < 2.44$ m, the velocity magnitude is less than 0.1 m/s. Since the test plane 2 is far away with the diffusers, the velocity magnitude is more uniform than that in the test plane 1. Figure 30 shows the variation of air temperature at the test positions as a function of height. In the test plane 1, the air temperature in test point changes evidently for different location for $Z > 1.83$ m while the air temperature for $Z < 1.83$ m almost does not vary for different $X$. The similar variation results also can be found in the test plane 2.
Figure 29: The air velocity magnitude at the different test positions
3.3 Discussions

From the numerical study of BTLab for CAD system, it is shown that the air inside the test space is well mixed. To compare the numerical results with the
experimental study, the velocity and temperature variations for the three test planes have been studied. Considering the symmetry, the first six test points for test planes 1 and 2 are studied. As in the numerical study, the velocity magnitude is greater near the diffuser as compared to the lower part of the diffuser.
CHAPTER 4

NUMERICAL MODELING OF UFAD (UNDER FLOOR AIR DISTRIBUTION) SYSTEM

Underfloor air distribution (UFAD) systems have been proven to be an effective method of delivering conditioned air to localized diffusers in the occupied zone of the building. Recently, UFAD systems have become popular design alternatives to conservative air distribution (CAD) such as overhead air distribution systems for thermal and ventilation control. Underfloor air distribution systems are gaining ground and began to attract the interest in HVAC applications. This is due to the potential advantages of underfloor systems compared to conventional overhead air distribution systems include better indoor air quality, improved thermal comfort, and reduced energy use [36]. Among many practical design procedures and associated software tools to determine airflow characteristics, CFD simulation appears to be promising for the use in UFAD design systems.

Despite the fact that UFAD systems are being applied in the field in increasing numbers, there is a strong need for an improved fundamental understanding of several key performance features of these systems. Most of the potential performance advantages of UFAD systems over conventional air distribution systems are related to the fact that
conditioned air is delivered at or near floor level and is returned at or near ceiling level. In comparison to conventional air distribution systems that deliver air at low velocities, typical UFAD systems deliver air through floor diffusers with higher supply air velocities. In addition to increasing the amount of mixing, these are more powerful supply air conditions can have significant impacts on room air stratification and thermal comfort in the occupied zone. The control and optimization of this stratification is crucial to system design and sizing, energy-efficient operation, and comfort performance of UFAD systems. To investigate these issues, a series of full-scale laboratory experiments were performed to determine room air stratification (RAS) for a variety of design and operating parameters. Stratification of air is the unmixed air within a confined space is separated into thermal layers due to temperature variations.

The theoretical behavior of UFAD systems is based on plume theory for displacement ventilation systems. For displacement ventilation, cool supply air is heated as it flows across the floor and is then drawn upward primarily through entrainment by thermal plumes that develop over heat sources in the room [36]. A stratification level is established that divides the room into two zones (upper and lower) having distinct airflow conditions. The lower zone, below the stratification level, has no recirculation and is close to displacement flow. The upper zone, above the stratification level, is characterized by recirculating flow producing a fairly well-mixed region. The height of this stratification level primarily depends on the room airflow rate relative to the magnitude of the heat sources. In UFAD systems, the use of floor diffusers that introduce air with some momentum alters the behavior in the lower zone by increasing the amount of mixing and changing the temperature profile. If the diffuser throw is close to the
stratification height or already exceeds it, the throw will penetrate into the warm upper layer bringing warm air down into the lower region. The amount of air brought down influences the temperatures in the lower region. The amount of mixing in the lower layer influences the gradient. In the limit as throw and the amount of mixing is reduced, UFAD systems tend to approach the operation of displacement ventilation systems. Higher throws that penetrate above the stratification height will result in warmer temperatures and less gradient in the lower region.

4.1 Pre-processing

As an innovative concept, among all the existing literatures in HVAC, only a limited number deal with UFAD systems. Even more, to the best of our knowledge, no one has ever performed a detailed numerical simulation for UFAD with swirl diffusers. This is mainly due to the complex geometry and flow features involved.

4.1.1 Creating the Geometry

In this research, 3-D geometry modeling for swirl diffusers is drawn by using Pro/E and IGES files. 3-D meshing for swirl diffuser is generated by using GAMBIT®, by taking the IGES file as input. As there are no detailed dimensions available for swirl diffusers used in BTLab, all the dimensions used in Pro/E modeling were measured. The measurements are made as accurately as possible. The geometry of the Nailor floor diffuser is shown in Figure 31.
Figure 31: Geometry of Nailor floor diffuser [46]

Figure 32: 3-D Nailor floor diffuser

(a) Exploded view  (b) Assembled view
The exploded view and assembled view of 3-D geometry of Nailor diffuser modeled by Pro/E is shown in Figure 32.

4.1.2 Mesh Generation

For convenience the bottom part of the swirl diffuser is simplified when input to GAMBIT® for meshing which includes the top part of the computational domain and the cylinder. Figure 33 shows the solid part and the fluid part of the diffuser which is the correspondingly computational domain.

![Solid part and Fluid part of the diffuser](image)

Figure 33: Part of the diffuser.

The computational domain was meshed with tetradehral cells and the meshed domain of the swirl diffuser is shown in Figure 34.
The numerical simulation of BTLab with 8 swirl diffusers will incur big computational costs as incorporating 8 swirl diffusers is a complex procedure. The most reasonable way to handle is to obtain the flow pattern of one swirl diffuser first, and then use the simulation results as the input boundary conditions for the 8 swirl diffusers. Thus the numerical analysis of the BTLab with 8 swirl diffusers is studied.

4.2 CFD Analysis on Single Swirl Diffuser

Figure 35 shows the computational domain and mesh system for the single swirl diffuser study analysis. Figure shows the zoom view of the swirl diffuser. For meshing the computational domain, 1,724,351 hexagonal cells are used. And the swirl diffuser is meshed using tetrahedral cells of size 0.05.
The domain extents are as follows:

x-coordinate: min (m) = -1.350000e+000, max (m) = 1.350000e+000

y-coordinate: min (m) = -1.350000e+000, max (m) = 1.350000e+000

z-coordinate: min (m) = -9.999999e-002, max (m) = 2.010000e-000

4.2.1 Numerical Procedure and Boundary conditions

The experimental value for the mass flow rate going into the 8 swirl diffusers are 0.392 m³/s (830 cfm), inlet flow rate for each swirl diffuser is 0.049 m³/s (103.75 cfm). The supply air temperature is set at 291.3K (18.3°C). The ambient air temperature is given as 300K. The underfloor is treated as an adiabatic boundary condition.

Figure 36 shows the modeling of the swirl diffuser and the direction of the arrows represents the air flow pattern from the inlet of the swirl diffuser.
In the numerical study, standard k-ε model is used, which is suitable for simulating indoor air flow, where swirl existed. It requires less computational effort than the other models and it is the simplest method which is sufficient for this problem. FLUENT® based on finite volume method; in our study a power law differencing scheme and segregated solver are chosen. The SIMPLE algorithm is used to resolve the coupling between pressure and velocity. First order upwind scheme is used for momentum, turbulence kinetic energy, turbulence dissipation rate and energy.

From the previous publications [50] it is found that there exists a clear zone for the swirl diffusers. The swirl diffusers are designed to provide rapid mixing with the room air and thus minimize any high velocity air movement, except within clear zone. The clear zone is approximately 1.2 m (4 ft) high and 0.6 m (2 ft) in diameter, directly above the floor diffuser. Hence, one of the criteria to examine this model is based on if we can get the clear zone in our model.
The air from the inlet of the swirl diffuser enters into the mixing zone. The zone above the clear zone is displacement zone where the air gets displaced in the upper direction and this upward movement of air in the room takes advantage of the natural buoyancy producing a vertical temperature gradient.

4.2.2 Single Swirl Diffuser Performance Analysis

The results for the single swirl diffuser analysis are shown. In Figure 38, the velocity magnitude at half way cross the swirl diffuser is shown. The velocity magnitude of the air from the inlet of the swirl diffuser is shown. Figure 39 shows the injection of air flow from the swirl diffuser is helical and the clear zone is easily identified. The clear zone is around 0.8 m in diameter and 1.2 m in height, hence the present numerical model is reasonable.
Figure 38: Velocity magnitude contour distribution at a selected slice (X=0m)

Figure 39: Clear zone is observed.
Figure 40: Temperature profile at a selected slice (X=0m)

Figure 40 shows the temperature distribution at center of the computational domain. Since the cold supply air is discharged from the diffuser the temperature at the inlet of the diffuser is cooled and is at the supply air temperature (291.3 K). Figure 41 shows the path lines released from the inlet of the diffuser and a total 227 are tracked.

Figure 41: Path lines released from the inlet of the swirl diffuser which are colored by the temperature (K)
The path lines released from the inlet of the diffuser are identical with the inspection of the actual air flow for the diffuser with the temperature profile.

Figure 42: Iso-surfaces at different temperature values.

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
4.3 CFD Analysis on the UFAD System of BTLab without Thermal Load

The simulation of the whole UFAD system with thermal loading using parallel computing technique was performed and 4 servers have been used. The numerical data of the UFAD system obtained from the numerical simulation was analyzed and qualitatively benchmarked with the experimental data. And also the UFAD system of BTLab under the different test point was analyzed.
Figure 44: Schematic of the UFAD system of the BTLab

About five million cells have been used in the present numerical model. It requires more than 3 GB of memory. However, for a 32-bit processor, it only provides addressing up to 4 GB of memory. There is a 2 GB memory limitation per processor. Hence, we need to deal with this model with parallel processing. In FLUENT®, the parallel solver allows us to compute a solution by using multiple processors that may be executing on the same computer, or on different computers in a network. Figure 45 illustrates the parallel FLUENT® architecture. Parallel processing in FLUENT® involves an interaction between FLUENT®, a host processor, and a set of compute-node processors. FLUENT® interacts with the host processor and the collection of compute nodes using a utility called cortex, that manages FLUENT®'s user interface and basic graphical functions. In these calculations, four CPUs with 3.4 G Hz and the memory with
12 GB are used in the same computers. The computational time requires approximately 4 days to finish the calculation.

Figure 45: Parallel FLUENT® architecture [41]

Compared to the whole space of the BTLab, the scale of the swirl diffuser is much smaller. To characterize the helical fluid flow of the swirl diffuser, a small size mesh needs be used for the spray slot of the diffuser. The 3-D geometry generated by SOLID WORKS® package was imported to the GAMBIT® and then meshed by using the tetrahedral mesh. For meshing the UFAD system with eight swirl diffusers, two kinds of configurations of mesh system were used. As shown in Figure 46, a very fine mesh was used for the swirl diffuser.
Figure 46: Computational domain and mesh system of the BT Lab

Domain extents are as follows:

x-coordinate: min (m) = 0.000000e+000, max (m) = 9.144000e+000
y-coordinate: min (m) = -6.096000e+000, max (m) = 0.000000e+000
z-coordinate: min (m) = -1.100054e-001, max (m) = 2.743200e+000
4.3.1 Numerical Procedure and Boundary Settings

To investigate the performance of UFAD system with different thermal loads, the CFD model with considering the floor heater has been developed, as shown in Figure 46. Total 12 heaters locate on the floor to supply as heat source. The total power for the heaters is 4192W. To simulate the heater, two kinds of boundary conditions can be used for this calculation. The boundary condition of heater can be taken as constant temperature while it also can be applied as a constant heat flux boundary condition. In this study a constant heat flux boundary condition was considered. In the calculation, all these wall temperatures come from the experimental data.

- Mass flow rate going into the 8 swirl diffusers are 0.392 m³/s (830 cfm).
- Inlet flow rate for each swirl diffuser is 0.049 m³/s (103.75 cfm).
- The supply air temperature is set at 291.3K.
- The ambient air temperature is given as 300K.
- The under floor is treated as adiabatic boundary conditions.
- East wall temperature = 297.7K
- West wall temperature = 297.6K
- North wall temperature = 297.7K
- South wall temperature = 297.3K
- Ceiling temperature = 297.8K
- Floor supply diffuser airflow rate = 0.0771Kg/s per each diffuser
• Heat flux for the heater surface = 1253.55 W/m².

Figure 47 shows the plane views of schematic BTLab with different test point. Numerical procedure for the UFAD system of the BTLab is similar to the procedure followed for the numerical study of the single swirl diffuser.

Figure 47: Schematic view BTLab with thermal load
In the numerical study, the standard k-ε model is used, which is suitable for simulating indoor air flow, where swirl existed. FLUENT® is based on finite volume method; in this study a power law differencing scheme and segregated solver are chosen. The SIMPLE algorithm is used to resolve the coupling between pressure and velocity. First order upwind scheme is used for momentum, turbulence kinetic energy, turbulence dissipation rate and energy.

4.3.2 Numerical Simulation Results

Comparing with the four way diffuser, the fluid flow for the swirl diffuser is more complicated. As shown in Figure 47, there exist three zones for the UFAD system of BTLab. Along the height of BTLab, the first zone is mixing zone (approximately 0-0.9m). In this zone, the supply air flow has been sprayed out helically from the swirl diffuser and mixed with ambient environment. The clear zone of swirl diffuser also locates inside this zone. The secondary zone is the uniform mixed zone (from 0.9 m to 1.8 m). The buoyancy air flow will be mixed uniformly in this zone and the air flow velocity decrease obviously. The third zone is the stagnant zone (from 1.8 m to 2.75 m). In this zone, the air flow velocity becomes very low and the average temperature is uniform. In this case, since there are floor heaters, the heater can induce the buoyancy force which moves the air flow upwards. In the present model, the effect of lighting is not considered and is without thermal load. The heat flux boundary condition was applied.
<table>
<thead>
<tr>
<th>Temperature(K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>310</td>
</tr>
<tr>
<td>308.73</td>
</tr>
<tr>
<td>307.46</td>
</tr>
<tr>
<td>306.2</td>
</tr>
<tr>
<td>304.93</td>
</tr>
<tr>
<td>303.66</td>
</tr>
<tr>
<td>302.4</td>
</tr>
<tr>
<td>301.13</td>
</tr>
<tr>
<td>299.86</td>
</tr>
<tr>
<td>298.6</td>
</tr>
<tr>
<td>297.33</td>
</tr>
<tr>
<td>296.06</td>
</tr>
<tr>
<td>294.8</td>
</tr>
<tr>
<td>293.53</td>
</tr>
<tr>
<td>292.26</td>
</tr>
<tr>
<td>291</td>
</tr>
</tbody>
</table>

(a) Temperature contour distribution of the swirl diffusers at a selected slice (Y=1m)

(b) Temperature distribution slices of the swirl diffusers (Y=1m) and heaters at (Y=3m)

Figure 48: Temperature contour distribution
To check the flow pattern, the path lines released from the bottom inlet of the swirl diffusers are shown in Figure 49. Here, the path lines are colored by path line ID.
and total 5280 path lines are tracked and in this model thermal load was not considered. As shown in the Figure 49, it can be found that the air flow injected from the swirl diffuser is highly twisted and the helical flows are formed. Unlike the single swirl diffuser case, both of the interactions among the diffusers and the effect of heaters make the fluid flow more complicated. Figure 48 shows the temperature distribution in the slice (Y=1m). As seen in Figure 48, the air temperature is much lower inside the clear zone. Outside the clear zone, it can be found that the temperature distribution is stratified along the height (Z direction). As the height increases, the air temperature decreases gradually. As seen in Figure 50, the air temperature is much lower inside the clear zone.

![Figure 50: Velocity magnitude at a selected slice (Y=1m)](image)

To show the clear zone of each swirl diffuser, the velocity magnitude contours in the slice (Y=1m) are shown in Figure 50. As illustrated in Figure 50, the clear zone from the present model is in the range of the definition [50] Here, the height of the clear zone
for each diffuser is around 1 m and 0.6 m in diameter. Hence, it can be concluded that the present numerical model is reasonable. In the future, the present model will be benchmarked with experimental data.

Figure 51: Velocity vector graph at a selected slice (Y=1m)
Figure 51 shows the velocity vector graph in the slice $Y=1m$. Since the slot of the swirl diffuser is very small, high air velocity can be found near the slot, and the flow pattern is very complicated. To analyze the velocity magnitude and temperature distribution, these are given as a function of $x$ for different heights at the slice ($Y=1m$) and are plotted in Figures 52 and 53. As indicated in Figures 52 and 53, the velocity magnitude is much higher around the swirl diffuser, while the temperature is relatively lower. Figure 54 illustrates the average temperature and velocity magnitude as a function of height. As the height increases, the average temperature increases gradually, while the average velocity magnitude decreases along the height ($Z$ direction).

According to the results from the present model, it can be concluded that the present numerical model is reasonable.

![Figure 52: Velocity magnitude as a function of x for different heights at the slice (Y=1m)](image)
Figure 53: Temperature as a function of x for different heights at the slice (Y=0m)

Figure 54: Average temperature and velocity magnitude as a function of height
The return air temperature is about 4°F lower than that from the experimental data. The main difference was caused by the heater boundary condition setting. Another reason for the difference is that the lighting is neglected in present model.

4.4 Discussion

The UFAD system of the BTLab is studied numerically considering heat flux boundary condition for heater and the effect of lighting was not considered. From the results, the evident characteristic is that the fluid flow is helical. It is seen that the flow from the diffuser is highly helical and twisted. And a clear zone is formed for each of the diffusers as per the previous publications which can be concluded that the obtained results are reasonable. And these results were compared with the experimental values by considering test position, which showed a good agreement.
CHAPTER 5

INVESTIGATION OF SPRAY ANGLE ON SWIRL DIFFUSER PERFORMANCE

As one of important thermal loads, the effect of lighting on the air flow and temperature can not be neglected in the numerical model since there is approximately 1100W from the light power. To investigate the performance of UFAD system with considering the effect of lighting, the CFD modeling on the BTLab coupled with radiation has to be developed. But considering the time factor, BTLab with radiation model can not be completed in this study. In the present model the effect of lighting is not considered. In the calculation, all of wall temperatures come from the experimental data. The wall temperatures are 297.7 K for the east wall, 297.7 K for the south wall, 297.6 K for the west wall and 297.3 K for the north wall. The ceiling temperature is taken as 297.8 K. The temperature of supply air flow is 291.3 K. The schematic of the BTLab with UFAD system was shown in Figure 44. The heat flux boundary condition for heater was applied while the effect of lighting was not considered.

As a critical part in UFAD system, the geometry modeling of swirl diffuser, especially spray angle of slot on from the top cover, is one crucial concern on the air flow distribution in the test space in BTLab. To investigate the effect of different spray angles on the air flow pattern, a parametric study on the geometry modeling of swirl diffuser with different spray angles has been made in this quarter. As shown in Figure 55, the swirl diffusers with angles from 3° to 7° have been studied.
The geometry model has been created and meshed in the GAMBIT®. In this simulation, the spray flow from one single swirl diffuser has been considered. The inlet flow rate for the swirl diffuser is 0.049 m³/s (103.75 cfm). The supply air temperature is set at 291.3K. The ambient air temperature is given as 300K. Figures 56 to 59 show the flow pattern and temperature distribution results. From the results, it is noted that the flow patterns strongly depend on the spray angle, as shown in Figure 56. For a small spray angle, the cold air flow has been injected directly toward the upside and only a small region within the diameter of the swirl diffuser has been affected. From this type of flow pattern, it is expected that the mixing effect of cold air is only limited to the small region above the swirl diffuser and the temperature distribution in the room is not uniform. When the spray angle increases, the mixing zone becomes larger gradually and a clear zone has formed. According to the previous publication [50], the clear zone for a classical swirl diffuser is approximately 1.2 m (4 ft) high and 0.6 m (2 ft) in diameter, directly above the floor diffuser. In the outside of clear zone, room air velocities will be less than 0.25 m/s (50 fpm). From the Figures, it can be found that when the spray angle is about $5^0-5.3^0$, the
clear zone from the present model is approximately within the range of the definition. Hence, it is recommended that the spray angle in geometry model of swirl diffuser should be taken as around 5.3°. As the spray angle increases, the air flow pattern will obviously change. Since the spray angle is very large, the air flow from swirl diffuser is highly twisted and moves along the underfloor. To visualize the air flow pattern, the path lines released from the bottom inlet of swirl diffuser under different spray angles are shown in Figure 57. After the air flow is injected from the swirl diffuser, it has been highly twisted and a helical flow pattern is formed. With different spray angles, the intensity of mixing effect of the helical flow is different. Figures 58 to 59 show the temperature distribution and the temperature isosurface. Since the heat transfer from hot air flow is controlled by forced convection, all of the phenomena in the temperature distribution with different spray angle can be explained by the air flow pattern. As the spray angle increases from 5.3°, the air flow is almost parallel to the floor.

(a) $\alpha = 3^0$

(b) $\alpha = 4^0$

Figure 56: Velocity magnitude (m/s) contour at a selected slice
Figure 57: Path lines released from the bottom inlet of the swirl diffuser (colored by path line ID)
Figure 57: Path lines released from the bottom inlet of the swirl diffuser (colored by path line ID) (contd...)
Figure 58: Temperature iso-surfaces with different spray angle

Figure 58: Temperature iso-surfaces with different spray angle

(a) $\alpha = 3^\circ$  
(b) $\alpha = 4^\circ$
(c) $\alpha = 5^\circ$  
(d) $\alpha = 5.3^\circ$
Figure 58: Temperature iso-surfaces with different spray angle (contd...)
Figure 59: Temperature (K) contour distributions at different heights (Z-plane)
Figure 59: Temperature (K) contour distributions at different heights (Z-plane) (contd...)
Figure 59: Temperature (K) contour distributions at different heights (Z-plane) (contd...)

(c) $\alpha = 5^0$
Figure 59: Temperature (K) contour distributions at different heights (Z-plane) (contd...)

(d) $\alpha = 5.3^0$
Figure 59: Temperature (K) contour distributions at different heights (Z-plane) (contd...)

(e) $\alpha = 5.7^0$
Figure 59: Temperature (K) contour distributions at different heights (Z-plane) (contd...)
Figure 59: Temperature (K) contour distributions at different heights (Z-plane) (contd...)
5.1 Benchmark Results

Benchmarking is important in research, especially in numerical simulation. It provides the validation of the tools and is the basis for the further effort. In this study, benchmark studies have been done to validate the effectiveness of FLUENT® in the numerical analysis of the UFAD system with zero thermal load.

To our knowledge among all the existing literatures in HVAC, only a limited number deal with UFAD systems. Due to the complex geometry and the flow properties, as far as known, no one has ever performed a detailed numerical simulation for UFAD with swirl diffusers. The laboratory phase of this task has been conducted in the UNLV (University of Nevada Las Vegas) Center and the numerical results for the UFAD system are compared with the experimental data.
Table 5.1: Operating Conditions for UFAD System with zero Thermal Load

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter Name</th>
<th>Symbol</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Average airflow rate during the test</td>
<td>$\dot{V}$</td>
<td>m$^3$/s(cfpm)</td>
<td>230 (0.109)</td>
</tr>
<tr>
<td>2</td>
<td>Average airflow rate per unit of floor area</td>
<td>$\dot{V} / A$</td>
<td>m$^3$/s·m$^2$(cfm/ft$^2$)</td>
<td>0.384(0.00195)</td>
</tr>
<tr>
<td>3</td>
<td>Average supply air outlet temperature</td>
<td>$T_{de}$</td>
<td>K (°C)</td>
<td>293.75 (20.6)</td>
</tr>
<tr>
<td>4</td>
<td>Average test zone</td>
<td>$T_{ac}$</td>
<td>K (°C)</td>
<td>296.85 (23.7)</td>
</tr>
<tr>
<td>5</td>
<td>Maximum recorded mean speed in the test</td>
<td>$v_{n-max}$</td>
<td>m/s (fpm)</td>
<td>46.6 (0.237)</td>
</tr>
<tr>
<td>6</td>
<td>Minimum recorded mean speed in the test zone</td>
<td>$v_{n-min}$</td>
<td>m/s (fpm)</td>
<td>4.7 (0.024)</td>
</tr>
<tr>
<td>7</td>
<td>Average heat load from floor heaters during the</td>
<td>$q$</td>
<td>W (Btu/h)</td>
<td>0 (0)</td>
</tr>
<tr>
<td>8</td>
<td>Average heat load from floor heaters per unit of</td>
<td>$q/A$</td>
<td>W/m$^2$ (Btu/h·ft$^2$)</td>
<td>0.0 (0.0)</td>
</tr>
<tr>
<td></td>
<td>Average return air temperature during the test</td>
<td>$T_{cl}$</td>
<td>K (°C)</td>
<td>297.45 (24.3)</td>
</tr>
<tr>
<td>9</td>
<td>Average air temperature in the return duct</td>
<td>$T_e$</td>
<td>K (°C)</td>
<td>296.85 (23.7)</td>
</tr>
</tbody>
</table>

Table 1 gives the UFAD system parameters without considering the thermal load. These results are compared with the experimental data and are benchmarked. The test grids for benchmark are marked with a pink box. The velocity magnitude and temperature are compared for the planes B and C as shown in Figure 60.
Test grids for benchmark:

Figure 60: Schematic of test grid in BTLab. The grids in the region with pink box have been benchmarked.

Figure 60 gives the test grid of the BTLab and the grids B3, B4, B5, C3, C4, C5 are benchmarked and are marked above.

Figures 61 to 63 give the benchmarked results of the B plane in which the temperature of numerical and experimental data almost coincide with each other but the velocity magnitude seems to be little higher in the experimental data than in the numerical data. The air flow rate of the square cone diffuser may not be accurate from the
experimental data which may be a possible reason for the result in Figure 61. And in the numerical study, in UFAD system, the airflow rate for swirl diffusers is considered to be uniform but in practical applications, the airflow rate may not be same for all the diffusers. This may be one of the causes for the higher velocity magnitude in numerical data.

Figure 61: Temperature and velocity magnitude comparison in test grid B3.
Figure 62: Temperature and velocity magnitude comparison in test grid B4.

Figure 63: Temperature and velocity magnitude comparison in test grid B5.
Figures 64 to 66 give the benchmarked results of the C plane. The temperature and velocity magnitude for the test grids C3, C4 seemed to be qualitatively valid. For the test grid C5, the velocity magnitude of the numerical data is a little higher than the experimental data. The boundary conditions may not be exact in both the cases, which may give slight variations in the results. But finally when the numerical data are compared with the experimental data, it can be concluded that the result is reasonable and showed a good agreement.

![Graphs of Temperature and Velocity Magnitude Comparison](image_url)

Figure 64: Temperature and velocity magnitude comparison in test grid C3.
Figure 65: Temperature and velocity magnitude comparison in test grid C4.

Figure 66: Temperature and velocity magnitude comparison in test grid C5.
From the above Figures 61 to 66, it can be concluded that the error is not more than 10% when temperature was compared and is not more than 25% with velocity magnitude comparisons.
CHAPTER 6

CONCLUSIONS AND SUGGESTIONS

6.1 Conclusions

This research study was carried out to inspect the influence of supply diffusers on the thermal environments and indoor air temperature stratification on energy consumption in the CAD and UFAD system. With the employment of the standard k-ε turbulence model and using FLUENT®, the thermal comfort environment of the CAD and UFAD system are investigated and compared with the experimental values. With the powerful CAD software, numerical study on the square cone and swirl diffusers was shown and validated with the experimental setup. It can be concluded that proper inlet conditions from the supply diffusers are essential for accurate CFD simulation using the standard k-ε turbulence model. It showed the good ability of FLUENT® to calculate the flow features.

In the numerical study of BTLab for the CAD system, we can conclude from the temperature and velocity profiles that the air inside the test space is well mixed. To compare these results with the experiment study, the velocity and temperature studies for the three test planes have been studied. Considering symmetry, the first six test points for test planes 1 and 2 are studied. As in the numerical study, the velocity magnitude is greater near the diffuser as compared to the lower part of the diffuser. From these results, we can conclude that the numerical results show good agreement.
In the numerical study of UFAD system, the spray angle of the swirl diffuser is considered to be the crucial part for the air flow distribution on which a parametric study has been made. The spray angles are considered from $3^\circ$ to $7^\circ$ in which the spray flow for single swirl diffuser is studied. From the velocity and temperature profiles, and pathlines of this study, it can be concluded that the flow strongly depends on the spray angle and approximated spray angle for the swirl diffuser is around $5.3^\circ$.

After the study of the spray angle, the UFAD system of the BTLab is studied numerically considering heat flux boundary condition for heater and the effect of lighting was not considered. From the results, the evident characteristic is that the fluid flow is helical. It is seen that the flow from the diffuser is highly helical and twisted. And a clear zone is formed for each of the diffuser as per the previous publications which can be concluded that the obtained results are feasible. It is observed that as the height increases, temperature decreases gradually and the temperature distribution is stratified in Z-direction. And these results were compared with the experimental values by considering test position, which showed a strong agreement.

Finally these results are benchmarked with the experimental results.

6.2 Future Work

- The radiation model for the lighting of the UFAD system: The research work is done for CAD and UFAD systems of the BTLab. In the UFAD system, the heat flux boundary condition was considered and the effect of lighting was not considered. The effect of lighting on the air flow is considered to be one of the important thermal loads and temperature can not be neglected in the numerical model since there is
approximately 1100W from lighting power. The radiation model for the lighting of the UFAD system needs to be developed in the future.

- The detailed comparisons of temperature and velocity fields for different thermal load with experimental data and the CFD modeling of CAD and the UFAD system model for the whole test space with zero load, half load and full load need to be performed as future work.

- Thermal comfort modeling: In the future, the detailed study on the humidity needs to be performed and then the thermal performance index will be calculated. From the results, the thermal performance of the UFAD and CAD systems need to be developed.

- Modeling of contaminate species transport: A discrete particle model is developed for indoor air quality study and we assume that the air contaminants are small particle and the effect of the particles on air flow has been neglected. For the discrete particle model we simulate a discrete second phase which consists of spherical particles dispersed in the continuous phase. In the future study, the detailed investigation with different contaminant species will be made under the UFAD system in the BTLab

- Real life simulation: In the above research study, the numerical analysis was performed in an empty room with diffusers, heaters and exhaust fans only. But in a real life simulation a desktop, a table, a few chairs and a human being should be included and the numerical study performed.
REFERENCES


9. V. Dorer, A. Weber, Air, contaminant and heat transport models: integration and

107

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.


30. Gordon Scott, Philip Richardson, The application of computational fluid dynamics in the industry.


33. Lars Davidson, An Introduction to Turbulence Models, Publication 97/2

34. Guangyu Cao, Indoor air flow prediction by means of computational fluid dynamics, Publication 97/2


39. www.cbe.berkeley.edu/underfloorair

40. C:\FLUENT.Inc\help\index.htm

41. Xiaobo Yang, and Mark Hayes, Application of grid techniques in CFD field, Cambridge eScience Centre, University of Cambridge
42. FLUENT Inc, FLUENT 3.1 User’s Guide, 2003, Lebanon, NH


44. www.madehvac.com

45. www.commercial.carrier.com

46. www.nailor.com

47. www.wikipedia.org

48. www.energydesignresources.com

49. www.kuken.com

VITA

Graduate College
University of Nevada, Las Vegas

Bhanu Rekha Bandhakavi

Home Address:
4235, Cottage Circle, Apt #4
Las Vegas, Nevada 89119

Degree:
Bachelor of Engineering, Mechanical Engineering, (2004) Acharya Nagarjuna University, Guntur, India

Thesis Title: Application of Computational Fluid Dynamics for High Energy Efficiency Design with Human Comfort of CAD – VAV and UFAD Systems

Thesis Examination Committee:
Chairperson, Dr. Yitung Chen, Ph. D
Committee Member, Dr. Boehm F. Robert, Ph.D., P.E.
Committee Member, Dr. Huajun Chen
Graduate Faculty Representative, Dr. Satish C. Bhatnagar, Ph.D.