TRIBHUVAN UNIVERSITY

INSTITUTE OF ENGINEERING

PULCHOWK CAMPUS

THESIS NO: M-49-MSMDE-2018-2022

Study of Thermal Performance of Ventilation System with Flow Dynamics

by

Ashesh Ghimire

A THESIS

SUBMITTED TO DEPARTMENT OF MECHANICAL AND AEROSPACE ENGINEERING IN PARTIAL FULFILLMENT OF THE REQUIREMENT FOR THE DEGREE OF MASTER OF SCIENCE IN MECHANICAL SYSTEMS DESIGN AND ENGINEERING

DEPARTMENT OF MECHANICAL AND AEROSPACE ENGINEERING

LALITPUR, NEPAL

MARCH, 2022

COPYRIGHT

TRIBHUVAN UNIVERSITY

INSTITUTE OF ENGINEERING

PULCHOWK CAMPUS

DEPARTMENT OF MECHANICAL AND AEROSPACE ENGINEERING

The undersigned certify that they have read, and recommend to the Institute of Engineering for acceptance, a thesis entitled "**Study of Thermal Performance of Ventilation System with Flow Dynamics**" submitted by Ashesh Ghimire in partial fulfillment of the requirements for the degree of Master in Mechanical Systems Design and Engineering.

____

Supervisor, Associate Prof. Vishwa Prasanna Amatya

Associate Professor, Department of Mechanical and Aerospace Engineering, Pulchowk Campus

____

External Examiner, Roshan Kumar Thapa

MD C-ZONE HVAC INC

Kathmandu, Nepal

____

Committee Chairperson, Dr. Surya Prasad Adhikari

Head, Department of Mechanical and Aerospace Engineering, Pulchowk Campus

Date: 20th March, 2022

ABSTRACT

The thermal comfort is an abstract term. Mathematical representation of thermal comfort is very complex as different individual have different preference and stimulation to thermal conditions. The basic factors that define thermal comfort are air temperature, radiant temperature, air velocity, humidity, clothing insulation level and metabolic rate. These factors can be controlled accordingly to achieve the best thermal condition for maximum occupant. The previous studies have been focused on the finding the analytical solution to the thermal comfort problem. Some simulation studies have been performed on the space ventilation, but this research takes an integrated approach to studying ventilation performance with computational fluid dynamics tools along with the study from analytical/empirical method .This research gives an insight to identifying some of the input parameters such as temperature and velocity for the inlets air in a conditioned space and the arrangement of inlets and/or outlets that helps in positioning and sizing of boundary surfaces for related fluid domain.

ACKNOWLEDGEMENT

I would like to express sincere gratitude to my supervisor Associate Prof. Vishwa Prasanna Amatya for the valuable suggestions and guidelines throughout the course project study. His understanding, expertise, and continuous support have been very important in completing this work. It was a great honor for me to pursue this work under his supervision.

I would like to express my due respect to Prof. Dr. Laxman Poudel, Department of Mechanical and Aerospace Engineering for his valuable and kind support. I would like to thank all the professors, lecturers, and administrative staffs of the department for their precious suggestion and kind support throughout the work. I am also very much thankful to respected Roshan Kumar Thapa for helping to solve queries that was ahead during the research.

I am always grateful to my, families and relatives for their continuous support and motivation.

ABBREVIATION

EDT	: Effective Draft Temperature
CFD	: Computational Fluid Dynamics
IAQ	: Indoor Air Quality
MV	: Mixed Ventilation
DV	: Displacement Ventilation
CLTD	: Cooling Load Temperature Difference
AHU	: Air Handling Unit
CLTD	: Cooling Load Temperature Difference
CLF	: Cooling Load Factor
SC	: Shading Coefficient
SCL	: Solar Cooling Load Factor
UFAD	: Underfloor Air Distribution
CFD	: Computational Fluid Dynamics
TMY	: Typical Meteorological Year

TABLE OF CONTENTS

LIST OF FIGURE

CHAPTER ONE: INTRODUCTION

1.1 Background

Over the last couple of years, the consumption of energy of building regarding the cooling and heating purposes has increased significantly. This has also resulted in the load demand peaking, especially in the places that record colder temperatures compared to the other places with warmer temperature. One of the ways to reduce the load is to use the passive heating system instead as the passive heating system is known for its benefit of consuming significantly negligible energy compared to the conventional heating and cooling system (ASHRAE, n.d.). A proper analysis of the strategies used to design buildings that revolve around the cooling energy consumption needs to be conducted in order to decrease the load. Orientation, thermal mass, building sizes and the design of the windows along with two other direct cooling strategies are some of the different passive cooling techniques have been analyzed in this study. Shading and natural cooling are some of the ways to control passive cooling as well.

A lot of unique and contrasting design strategies have equally different and unique impact on the requirement of cooling energy and the study also shows the thermo-physical property of building has the most amount of effect on the energy a building consumes. The flow of air also differs as per the design strategy. The flow of air and its patterns also depend on the pressure exerted through unique ways. Indoor atmosphere significantly influences the health of humans with regards to different sorts of allergies, respiratory issues like Asthma, headaches and their general productivity(Fisk, 2002). Higher air temperature and the humidity of air result in behavioral actions like fanning one's self, consuming cold water or water stored in a cooler place, wearing light-colored clothes, increasing the rotation of the fan, going outdoors.

Unfortunately, this is not the permanent solution to the thermal comfort problem. Rather, the resolution to these problems can be the proper understanding of the change in climate and understanding several other factors like the velocity of the air, global temperature, and humidity. Natural Ventilation can help the indoor environment successfully achieve a satisfying thermal comfort. (Liping & Hien, 2007) signified two different techniques that can help the thermal comfort be better and more enjoyable in indoor environment.

1.2 An Introduction to Thermal Comfort

Thermal comfort is described as "the state of mind that releases happiness and comfort with the thermal environment," according to the international standard EN ISO 7730. In other words, it is a state in which a person's temperature is balanced, not too hot nor too cold.

Human thermal comfort cannot be stated in degrees and cannot be defined by an average temperature window or range. It is a matter of personal preference and a result of a variety of factors that determine how a person prefers to live in a given environment. Even though they all live in the same environment, it differs from individual to person. According to the Health and Safety Executive, acceptable comfort in terms of the thermal environment may be achieved when about 80 percent of persons living inside are pleased and comfortable. [1]

Factors that define the thermal comfort of a space:

1. Metabolic Rate

2. Clothing Insulation

3. Air Temperature (summer 23℃ and winter 20℃)

4. Radiant Temperature

5. Air Speed (summer 0.25m/s and winter 0.15m/s)

6. Humidity

It is possible for all these six factors to vary with time and this standard only address the thermal comfort in a steady state. Which means even if a person enters the room with these standard conditions he/she will not immediately feel comfortable if they have experiences different thermal environment just prior to entering this space (ASHRAE, n.d.).

1.3 Effective Draft Temperature

Effective Draft Temperature (EDT) is basically the measure of thermal comfort in a room. By integrating the physiological impacts of air temperature and air velocity on a human body, the Effective Draft Temperature gives a quantitative measure of comfort at a discrete spot in a room. (SimScale CAE Forum, 2016)

If the values of Equation below are between –3°F and +2°F, and the observed velocity at the site is less than 70 fpm (0.36 m/s), the point is deemed pleasant.

(Tx – Tc) – 0.07(Vx – 30) EDT

Where, EDT is the Effective Draft Temperature in degrees Fahrenheit

Tx = local airstream dry-bulb temperature, in degrees Fahrenheit

Vx = local airstream centerline velocity, fpm

Tc = average room dry temperature, (control) °F

The formula becomes: if the inputs are in SI units (Temp in Kelvin, Velocity in m/s),

EDT = [1.8(T - Tc) -0.07(196.85*Vx - 30)]

1.4 Thermal Simulation

How can simulation software be used to optimize thermal comfort?

Engineering simulation software may be used by anybody, whether they are civil engineers, mechanical engineers, or HVAC designers, to model ideal thermal conditions. The forms, sizes, and placements of inlet and outflow vanes may be tuned to save energy costs. Furthermore, the cloud-based SimScale CAE platform may be utilized for early virtual testing of various ventilation systems, fans, or even full building designs in order to readily visualize air flow and anticipate performance.

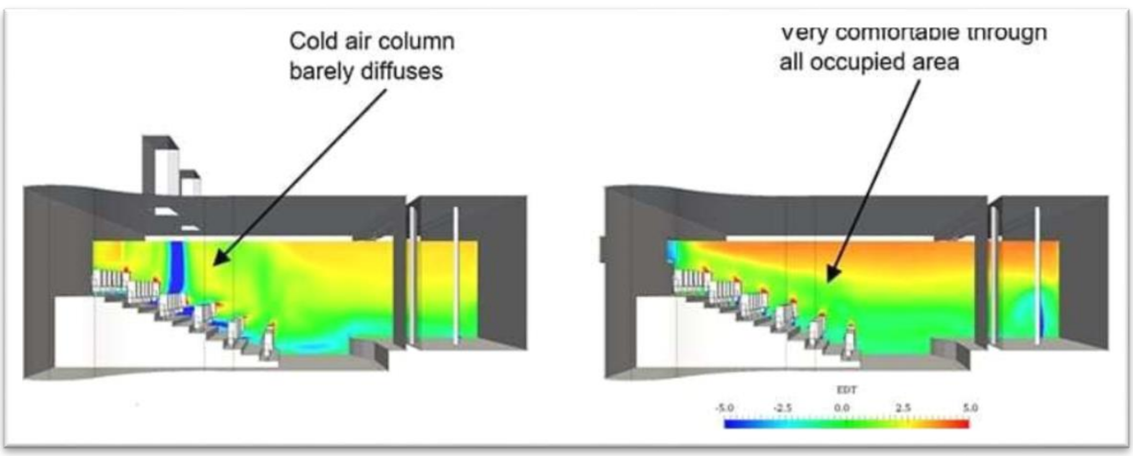


Figure 1: Simulations of the effective draft temperature/EDT in a movie theater using CFD (conducted with the SimScale cloud-based platform)

The optimization of HVAC systems revolves around CFD and thermal simulations. The distribution of the flow of the air and the dynamics of the air can be created in different spaces of a building. This starts from basic things like the infusion of fresh air and getting rid of the air that is stale. Removing the heat produced by the usage of electronic devices, office cubical, formation of the walls, and exposure through the doors and windows is also essential.

"How to Make an Office More Thermally Comfortable." This will take you to further information about the formation of ideal thermal comfort, other elements that influence it, and some other practical aspects of HVAC systems and building design.

Furthermore, many free templates can be found in the SimScale Public Projects Library, which can be copied and used to set up your own formation to anticipate thermal comfort in various building designs or optimize HVAC designs for various office spaces and buildings that are commercial, residential, or both.

1.5 Introduction to CFD

Computational Fluid Dynamics (CFD) is a process in which what is perceived as fluid flow is mathematically modeled and sorted by numerical application.

When an engineer is given the task of designing a new product as a winning race car, aerodynamics plays a vital role in the entire engineering process. However, the aerodynamic process cannot be so easily measured during the conceptual phase. Basically, the best and only way an engineer can improve his designs is to do some physical exercises on the prototypes of used or future products. With the rapid growth of computers and computing power (thanks to Moore's law!), Computational Fluid Dynamics has become a widely used tool that can produce fluid flow solutions, either in combination or without solid interactions. In the analysis of CFD software, testing of fluid flow relative to its properties such as pressure, density, speed, temperature, and viscosity is important. To produce a proper or direct response to a material connected to a liquid flow, these factors must also be considered.

For example, Navier-Stokes (N-S) statistics are described as a physical model. This also clarifies all the changes in all these apparent aspects of heat transfer and fluid flow. The mathematical model differs in relation to the content of the story such as heat transfer and weight transfer, chemical reactions and phase changes etc. Also, the reliability of CFD analysis depends largely on the overall structure of the whole process. The adjustment of the mathematical model is critical to the creation of an accurate problem-solving case. Besides, identifying the correct numerical methods is the key to producing a reliable and effective solution. CFD analysis is an important part of a well-designed product development process because there may be a decrease in the number of visual images.

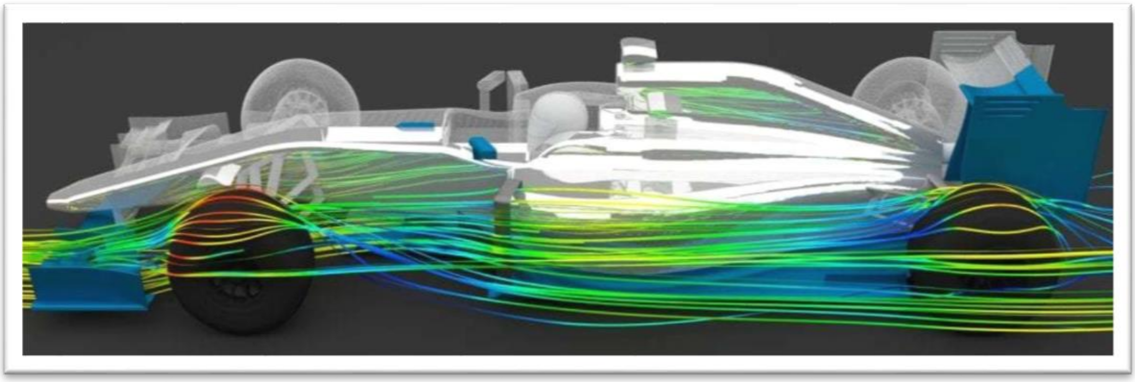


Figure 2: Airflow around the Formula One vehicle as depicted by N-S equations.

1.6 Governing Equations

The fundamental source of thermos-fluids is investigated using governing equations based on the conservation rule of the physical characteristics of fluids. The three primary laws of conservation are represented by these simple equations.

Mass Conservation: Equation of Continuity

Newton's Second Law of Conservation of Momentum

The First Law of Thermodynamics, sometimes known as the Energy Equation, states that all energy is conserved.

Quantity, momentum, and strength are the stable elements of a closed system in terms of these principles. They must be saved. There are certain structures on which all investigations of liquid flow with thermal changes depend. Three unknown ones to be detected simultaneously in these three basic conservation values are Velocity (v), Pressure (p) and total temperature (T). However, pressure and temperature are two important factors that represent thermodynamic variables. There are four other thermodynamic variables that are part of the final type of storage statistics. The other four include congestion (P), Enthalpy (H), Viscosity (μ) and Thermal Conductivity (K). Among them, the last two Viscosity and Thermal conductivity are also transport hubs. All four structures are specifically analyzed by the value of P and T.

Prior to designing any product that involves liquid flow, it is essential for liquid flow analysis to determine Velocity, Pressure, and temperature for each point of the flow system. Additionally, kinematic-based monitoring of water flow monitoring is a fundamental issue. The flow of fluid can be detected by Lagrangian or Eulerian methods. The Lagrangian definition revolves around the theory of following a particle large enough for the structures to be visible. The first-time links (t0) and the same particle links at the time (t1) should be checked and analyzed.

1.7 Statement of Problem

Hot comfort reflects the state of one's mind and usually refers to a situation in which a person feels very hot or very cold. There are many factors that must be considered in terms of individual and environmental factors when it comes to deciding what can make people feel at ease in the best possible way. This is also the reason why it is difficult to limit the hot comfort in a particular definition. These features also create a 'hot spot for a person'. People who work under conditions or in an unhealthy and very cold environment are more likely to behave unsafe as their decision-making ability and productivity when it comes to putting effort into it diminishes.

Following millions of different particles along the way is almost impossible. In the Eulerian way, a speed field is a test object instead of following any particles crossing the path as a time-dividing function. The following example of the arrows provides a detailed explanation of these two approaches.

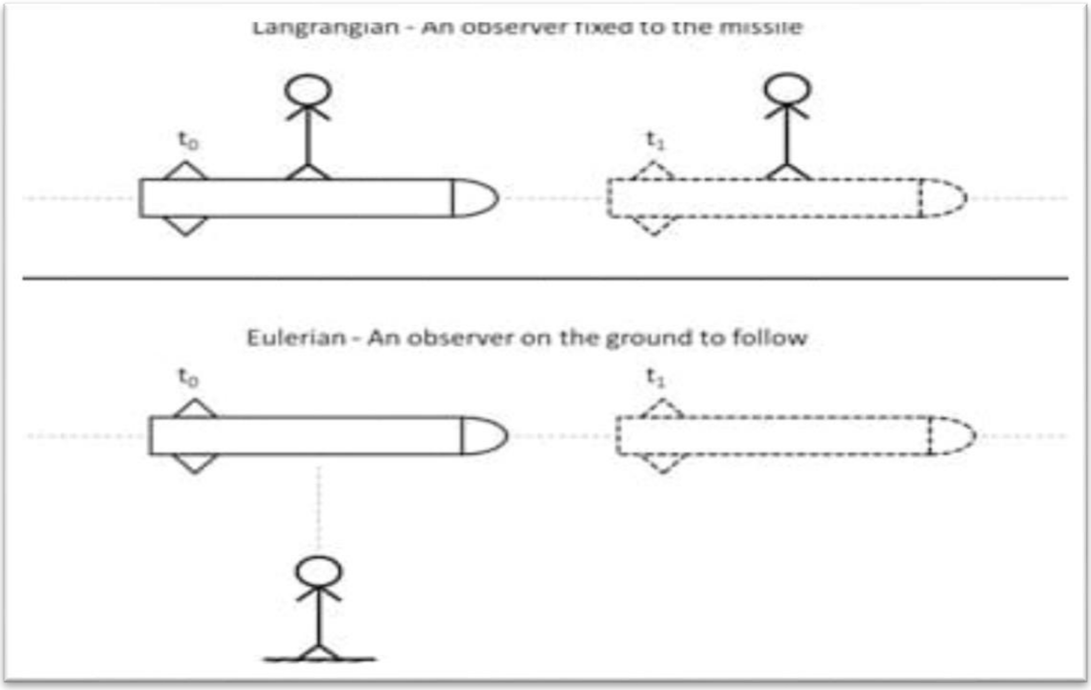


Figure 3: In the Lagrangian method, the person is stable with relation to the missile, however in the Eulerian approach, it is radically different.

Langrangian: From beginning to end, every domain point must be improved. Eulerian: We start with a single point (Control Volume) in a liquid and then look at the movement of particles inside that volume. The building of a Lagrangian is always time-dependent. The initial particle linkages are represented by the letters A, B, and C. Similarly, at the same moment, X, Y, and Z exhibit identical particle connections (t). The following is a definition of Lagrangian flow:

x = x (a, b, c, t)

y = y (a, b, c, t)

z = z (a, b, c, t) (1)

U, V, and W illustrate the speed components at distinct places (X, Y, Z) in time in the Eulerian approach (t). As a result, the unknown elements U, V, and W are functions of independent variables (X, Y, Z and t). The movement as the Eulerian flow at any given moment is described below.

u = u (x, y, z, t)

v = v (x, y, z, t)

$w = w(x, y, z, t)$ (2)

1.8 Objectives

There are two major sections in which the objectives of the study are divided. The following are the objectives.

1.8.1 Primary Objectives

The primary objectives of the research are listed below.

- To properly understand thermal behavior and flow dynamics of ventilation system.

1.8.2 Specific Objective

Besides the primary objectives, other secondary objectives are mentioned below:

- To successfully perform the parametric study and design optimization.
- To also analyze how the flow dynamics and the thermal comfort are related.
- To help identify the type of ventilation system that will be most efficient in term of providing thermal comfort to maximum number of occupants.

1.9 SCOPE AND LIMITATION

The scopes and limitations of the projects are talked about in the succeeding heading.

1.9.1 Scope

The various scopes of the research are:

- Case Based design and calculation will be performed.
- The study is preformed analytically and also numerically.
- The region of interest is the human occupant space of the hall which is subset of the fluid domain under study.

1.9.2 Limitation

The different limitations and rules for the projects are:

- The research locations that are mentioned will be the only locations where all the analysis, studies, and calculations will be done.
- Transient behavior is not considered in this study.
- Only thermal comfort related to indoor temperature will be considered,

humidity, infiltration and equipment load in relation with thermal comfort will be neglected.

- The study will be focused on the human occupant space and not the whole domain of study.
- The study is based on the full attendance of the occupants for cooling application only.

CHAPTER TWO: LITERATURE REVIEW

With fewer people living in the same indoor environment, performance is gradually declining without a personal approach to temperature control. (Heschong, 2002) explains that students with students who do not receive much daylight or windows are able to earn a score of 7 or 12% compared to others who do not have access to those items. This also means that restaurants need open air and open spaces for daylight to pass. This will allow students to learn more even while in the restaurant or relaxing.

(Fisk, 2002) pointed out that the environment we find in our home has a profound effect on people's health when it comes to things like allergies, shortness of breath and shortness of breath such as asthma and other things like headaches and dysfunction.

Air permeability, frequent use of cold water or water stored at low temperatures, wearing light clothing (both by weight and color) are some of the functions caused by air humidity and high air temperature in humans. Finding a seat by the window is also a common task that starts with this situation. However, this is not a permanent solution to this and instead, understanding that climatic conditions such as air temperature, wind speed, global temperature, and related humidity function will be the best solution.

People can enjoy a comfortable warm indoor environment with the help of natural breathing. According to (Liping & Hien, 2007), there are two main ways in which thermal comfort can be developed in the home environment.

For night cooling, the use of an air conditioner is an effective method. (Jones & Kirby, 2012) explained that this approach may serve as a natural venture or part of this process. If the interior designs and conditions are carefully considered, it can ensure that the occupants have the best possible comfort. Despite the emotions involved, these thermal conditions are important aspects of architectural design. Therefore, the factors discussed in the next section need to be carefully considered in order to determine the thermal comfort of the building occupants.

When it comes to natural factors, there are 4 basic factors that can affect or influence thermal comfort.

We often find people who are sensitive to wind speed which is why wind speed can be the most important element of thermal comfort among the factors mentioned above. Fake air that can withstand or withstand a period of time can cause a decrease in productivity and cause less energy. Wind speed can improve heat loss through convection without the effect of such air temperature change. However, when the air temperature is too low, the convective heat loss is greatly increased. This can be found mainly in warm or humid conditions. Additionally, exercise can stimulate air movement as suggested by (Liping & Hien, 2007).

(Humphreys & Nicol, 1998) states that measurement of wind speed creates certain problems due to wind fluctuations and ambiguity.

The temperature of the air around the human body is called the temperature of the air (Ta). Air temperature is also an important factor in determining temperature pressure. Represents the part of the environment around a person that determines the flow of heat between the human body and the air around the human body. Temperatures vary from person to person but the temperature in the human body does not always mean the average temperature. The air temperature found very close to the body wearing the clothes does not represent this either as some “boundary conditions” affect the object. An example of this is the layer of warm air that surrounds the body even when a person is in a cold environment.

Air temperature is said to be the most important weather factor affecting the comfort of the heat. But there are many other factors to consider before making a decision. When heat is transferred from the maximum temperature range to the lowest possible temperature without creating insufficient space, Radiant Temperature is available.

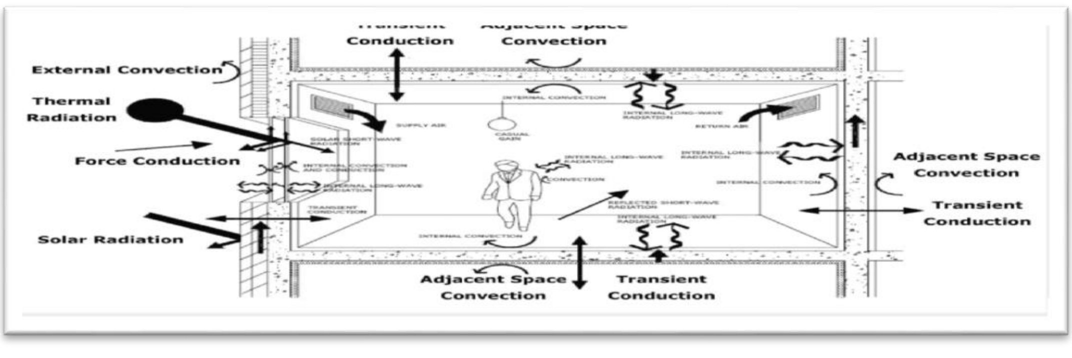


Figure 4: Space thermal transfer (Source: Energy Plus, 2009)

When a person is in a state of hot temperatures or balance, the person does not feel too cold and does not feel too cold. However, different people have different boundaries when considering heat and cold. This can be seen when one person wears clothes that look comfortable compared to another person who wears loose or extra clothing at the same temperature or in the same place. This does not change the fact that everyone responds in the same way as natural conditions change over time.

Heat transfer between people in a confined space can be analyzed to determine the level of activity. Those conditions are the balance of body temperature, the degree of sweating within the free range and the mean skin temperature. In order to provide thermal comfort to those living in buildings, all thermal comfort requirements must be met. (Muncey, 1979) states that in order to assess human comfort, air and temperature must be analyzed. This can also reduce the chances of the skin becoming unpleasant or wet and, too, eye comfort is well cared for. Other problems such as temperature stability, bacterial overgrowth, and respiratory-related illness are also avoided. More than 80% of citizens end up experiencing acceptable warm conditions if these conditions are considered continuously. According to (Olesen, B. W. Brager, 2004), two basic methods are available that can measure the need for comfort. One of those measures involves the conduct of a survey of residents, and the other revolves around natural features that place limits on luxury.

1.10 Types of ventilation system

1.10.1 Displacement and mixed ventilation

The waste air is driven to the ceiling and then out of the room through exhaust panels using displacement ventilation, which employs a slow-moving stream of new air from the floor to displace it. Displacement ventilation, which is based on temperature and concentration distribution, allows fresh air to replace dirty air in the workplace. However, mixing ventilation maintains the density of interior pollutants by bringing in sufficient outside air to uniformly dilute emissions from inside sources..(Editorial Simscale, 2017)

Displacement ventilation technology has been widely employed in the HVAC industry in Europe for decades, and it has also gained popularity in the United States, where it is frequently used in classrooms and workplaces.

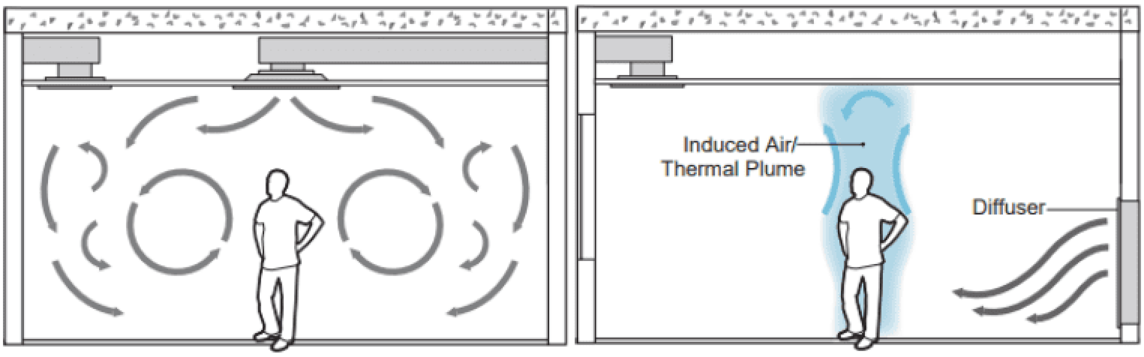


Figure 5: Mixed ventilation and displacement ventilation with a single occupant

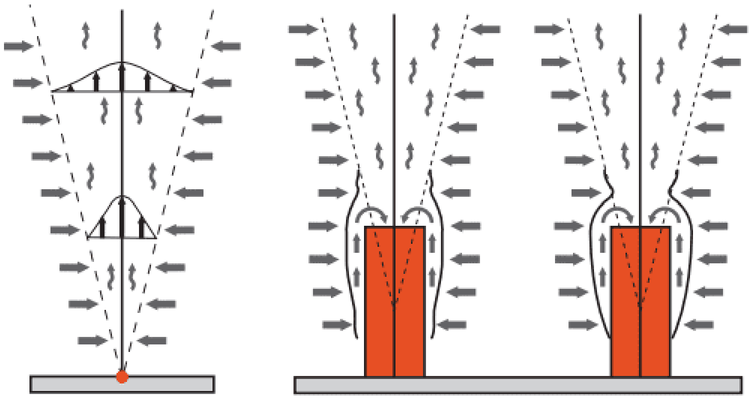


Figure 6: Draft induced thermal plume for point and cylindrical heat source

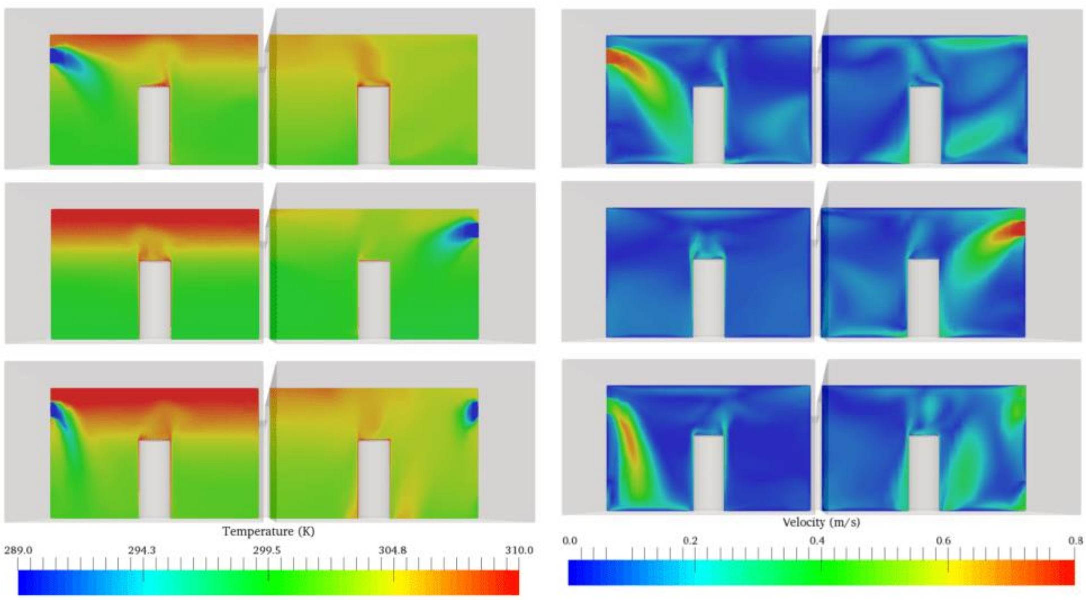


Figure 7: Profile of temperature and velocity for mixed ventilation system

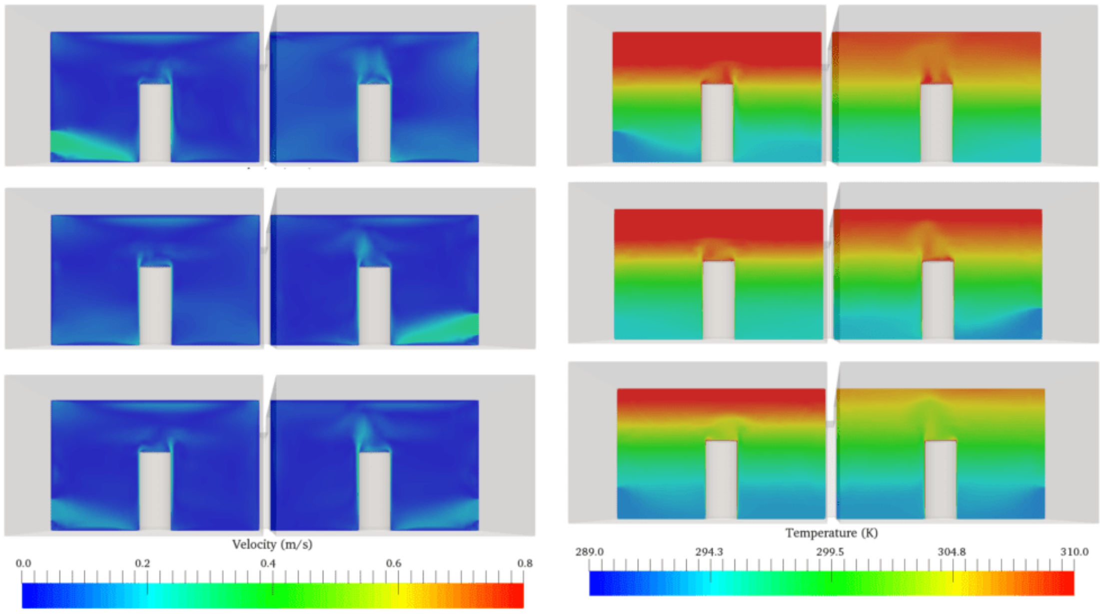


Figure 8: Profile of temperature and velocity distribution for displacement ventilation

1.10.2 Underfloor Air Ventilation (UFAD)

In offices and other commercial buildings, underfloor air distribution (UFAD) is a means of providing ventilation and space cooling. This approach is becoming a popular alternative to traditional ceiling-based air ducts in HVAC system designs. The open area between the structural concrete slab and the raised floor system is used by UFAD to route conditioned air into building zones. The most typical technique of distributing air is through supply outlets carefully positioned on floors.(Mateus & Da Graca, 2013)

Figure 9: Underfloor Air Ventilation

1.10.3 Diffuse Ceiling Ventilation

The open area between the structural concrete slab and the raised floor system is used by UFAD to route conditioned air into building zones. The most typical technique of distributing air is through supply outlets carefully positioned on floors(Zhang et al., 2014). As a result, low-velocity air flows into the occupied zone with no definite jet direction, thus the name "diffuse." Even when delivering cold air, this technology has the ability to decrease draught danger in the inhabited zone. The danger of draught in the occupied zone is caused by convective fluxes induced by heat sources such as people, equipment, or windows.

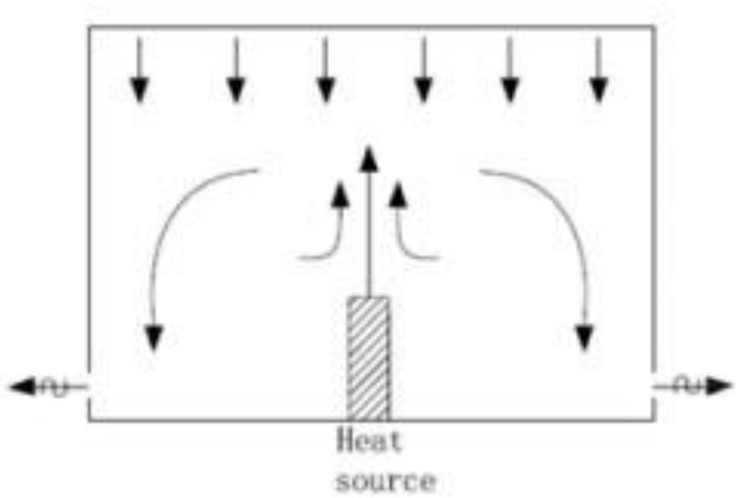


Figure 10: Ceiling Diffuse Ventilation

1.10.4 Stratum Ventilation

With a horizontal airflow at head level, a small and reverse temperature differential between the head and foot levels, and a high air distribution performance index, stratum ventilation may offer a thermally pleasant atmosphere (Zhang Lin, 2014).

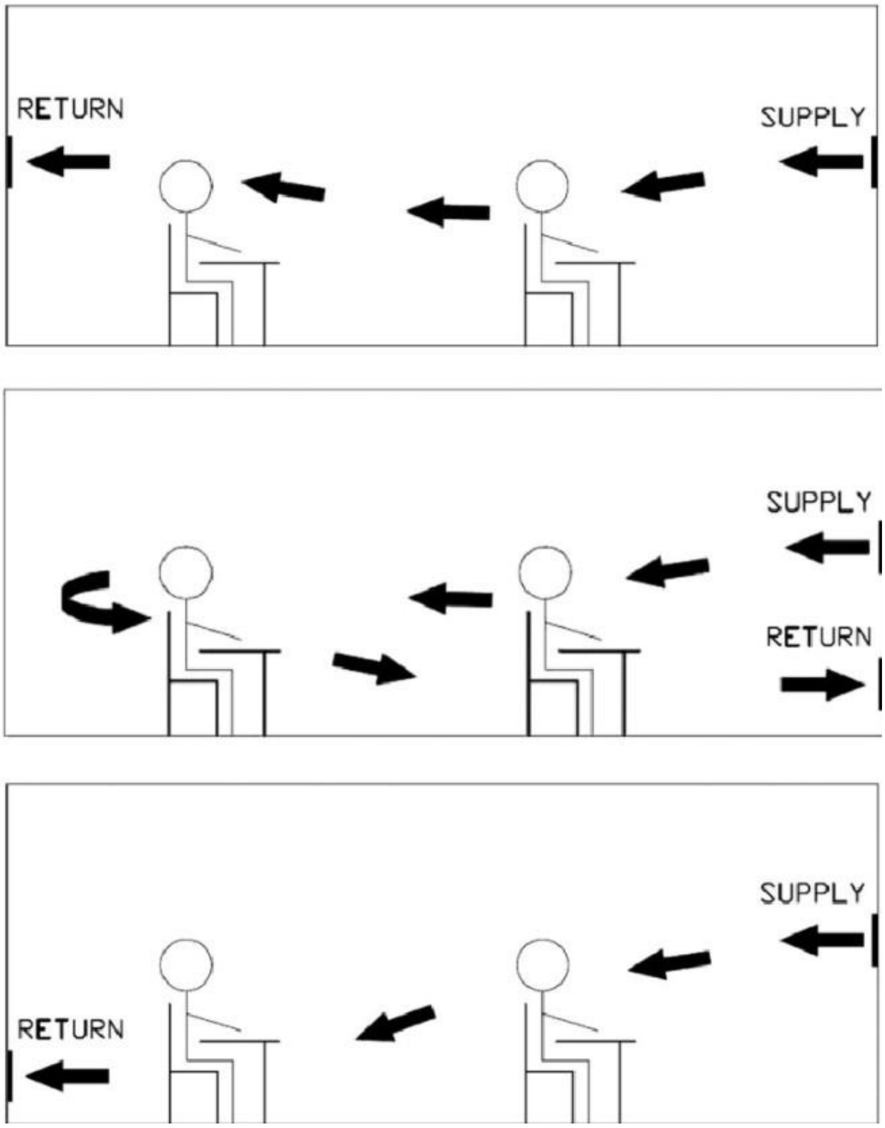


Figure 11: Stratum Ventilation

1.10.5 Impinging Jet ventilation

The impinging jet ventilation system was designed as a novel ventilation technology to alleviate the shortcomings of displacement ventilation. When the supply flow rate is low, it is known to cause a stratified state comparable to displacement during cooling operation. When the supply flowrate is high, it provides a similar situation with mixed ventilation. As a result, it's classified as a blend of displacement and mixed ventilation. The air is delivered to a room through a jet, and once it hits the floor, the provided air spreads horizontally across the space in a thin layer.(Yamasawa et al., 2021)

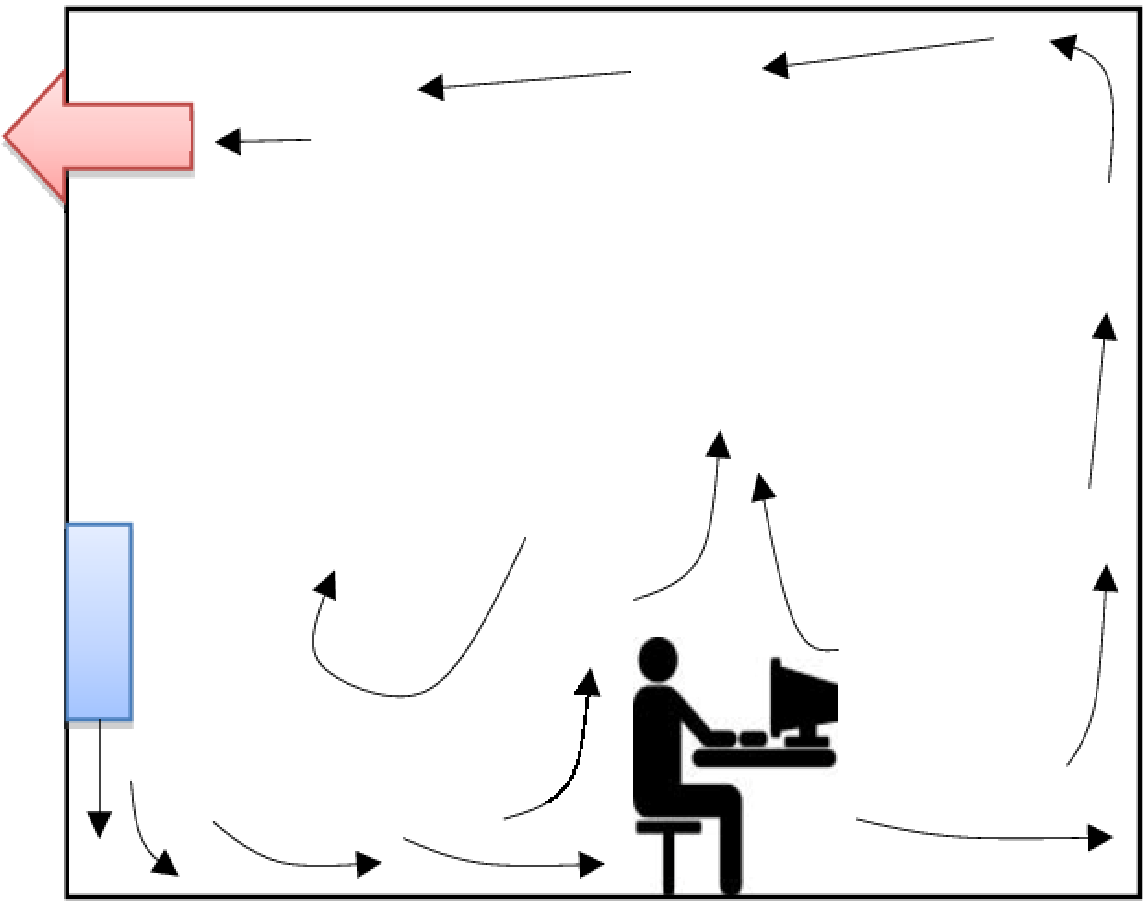


Figure 12: Impinging Jet Ventilation

The 6 common types of diffusers in HVAC are:

Directional Diffuser (most common): 4-way diffusers, 3-way diffusers, 2-way diffusers, and 1-way diffusers are examples of directional diffusers. Standard HVAC supply air diffusers include directional diffusers.

Linear Slot Diffuser (Aesthetic): For airflow management, linear slot diffusers requires a rectangular plenum box with an opposed-blade damper.

Round Diffuser (Uniform Distribution): Diffusers that are spherical in form are comparable to directional diffusers. They distribute air even better than directed diffusers.

Swirl Diffuser (High Air Change): The supply air is "swirled" by swirl diffusers, which feature angled blades. They come in a variety of shapes, including round and square. Air is not supplied vertically by directional and spherical diffusers. Linear slot diffusers, on the other hand, do not feed air horizontally. Swirl diffusers, on the other hand, are a hybrid of directional and linear slot diffusers. They can fling air in both horizontal and vertical directions, and the volume of airflow in each direction may be controlled by adjusting the blades.

Double Deflection Diffuser (Large Airflow): Double deflection diffusers have several square-shaped holes, similar to a mesh. They are similar to linear slot diffusers but can handle more airflow.

Jet Diffuser (Long Distance Throw): Jet diffusers are circular supply air diffusers with a clean, smooth appearance. Unlike other diffusers with several holes, most of them simply have one large aperture for air to travel through.

CHAPTER THREE: RESEARCH METHODOLOGY

1.11 Research Question

The research will further be guided by finding out the answers to the following questions:

- How the flow dynamics is related to the thermal comfort?
- How the flow arrangements and properties (thermal and flow) of inlet influences performance of a building?
- How can highest degree of thermal performance be achieved for occupancy space?

1.12 Methodology

A tabular demonstration that showcases whole purpose and process of the study can be found below.

Particular	Purpose	Process
Literature Review	To collect the information regarding the research and identify the essential parameters required	Several published and unpublished report, journal report and text of relevant field will be referred
Field Study	To gain the practical and installation knowledge and collect the data for calculation and simulation	Questionnaire will be prepared and interview with experts will be conducted
Data Analysis and interpretation	Collected data need to be filtered and necessary parameters should be focused.	With the help of the literature review and suggestion provided by supervisor and other experts
Mathematical Calculation	To calculate the thermal load, design and develop the CAD model	Hand calculation will be used followed by Excel calculation.
CAD Model development	This acts as tool for simulation and it represents the calculated dimensions in simplified form.	With the use of Solid works Design tool
Simulations	To visualize the phenomenon undergoing while different ventilation system is in operations.	ANSYS Fluent will be used as simulation software
Result Validation	To validate the simulation result with the analytical solution.	ANSYS Fluent result compared to CLTD results.

1.13 Work Description

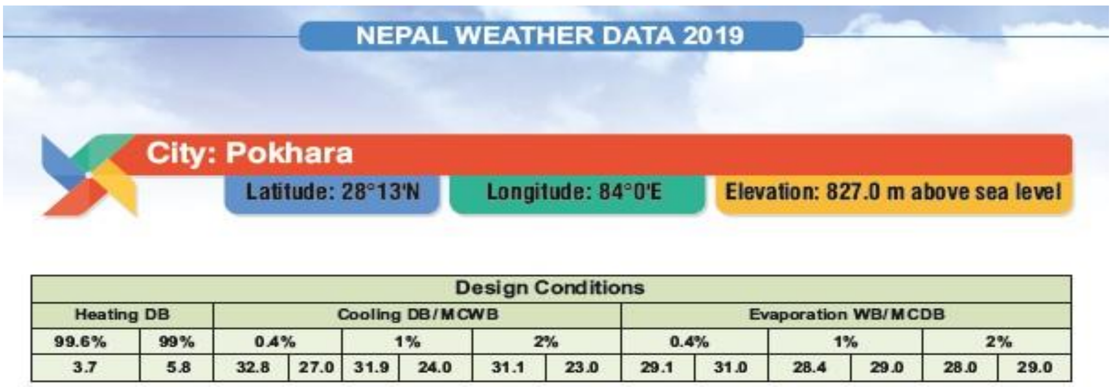
This study describes an application that uses simulation and CFD to prepare the ventilation system for the movie hall at Pokhara's Mid-town retail mall. Using the CLTD approach, the cooling load was calculated.

CFD simulations were used to predict detailed air flow velocity and space temperatures as well as to evaluate the accuracy of the different size and configuration of inlets and outlets (diffuser and exhaust) to achieve best temperature and velocity condition in the occupant region.

1.14 Building Cooling Load Calculation

Using weather data of Pokhara according ISHRAE, 2019, design ambient temperature is identified. For cooling load, ambient temperature of 32.8 degree Celsius is applied for calculation.

Also, from Accuweather, the maximum temperature was found to be 34 degrees Celsius over last year.



Design Conditions													
Heating DB		Cooling DB/MCWB						Evaporation WB/MCDB					
99.6%	99%	0.4%		1%		2%		0.4%		1%		2%	
3.7	5.8	32.8	27.0	31.9	24.0	31.1	23.0	29.1	31.0	28.4	29.0	28.0	29.0

Figure 13: ISHRAE Weather Data for Design Condition

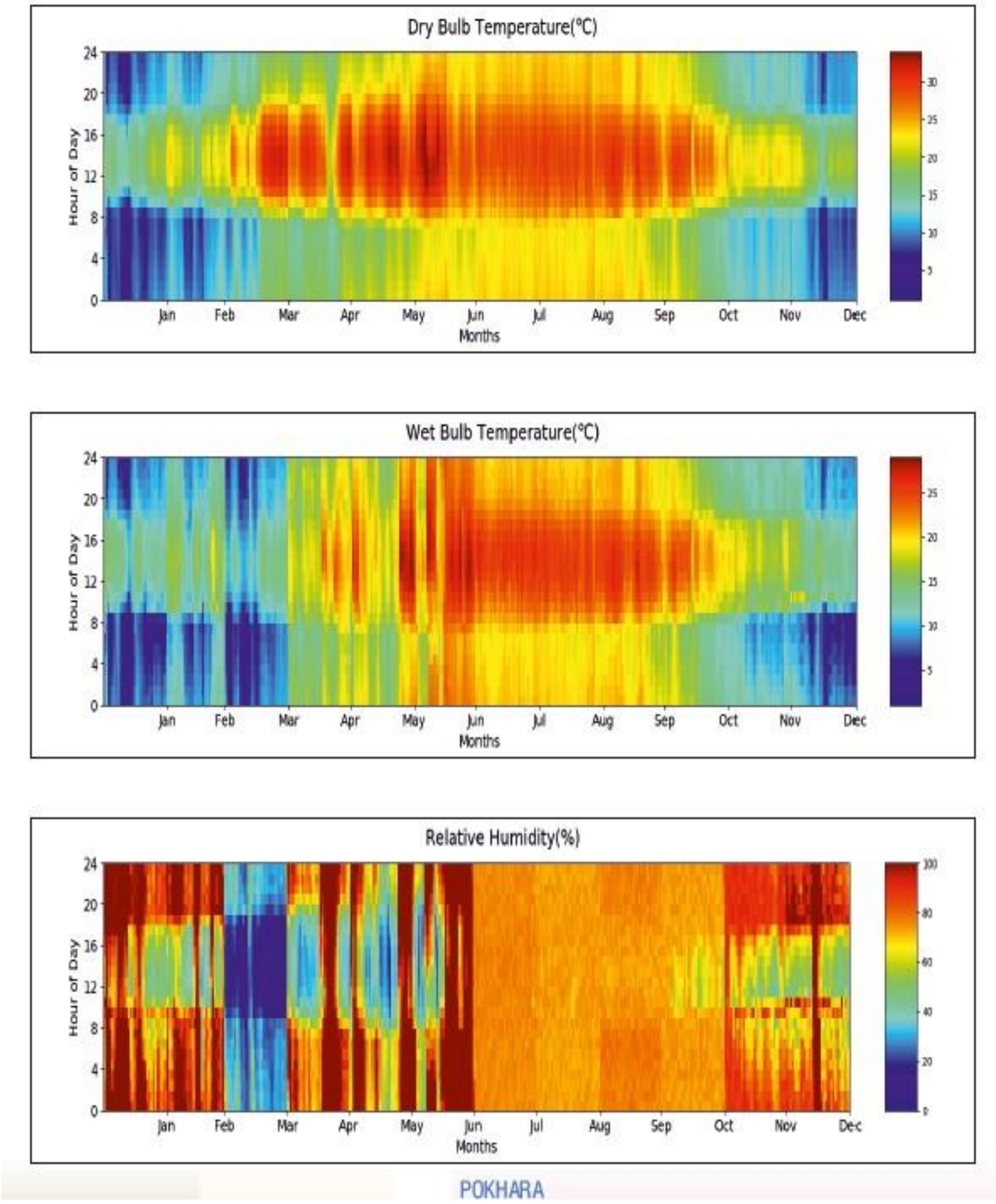


Figure 14: ISHRAE weather data for Pokhara

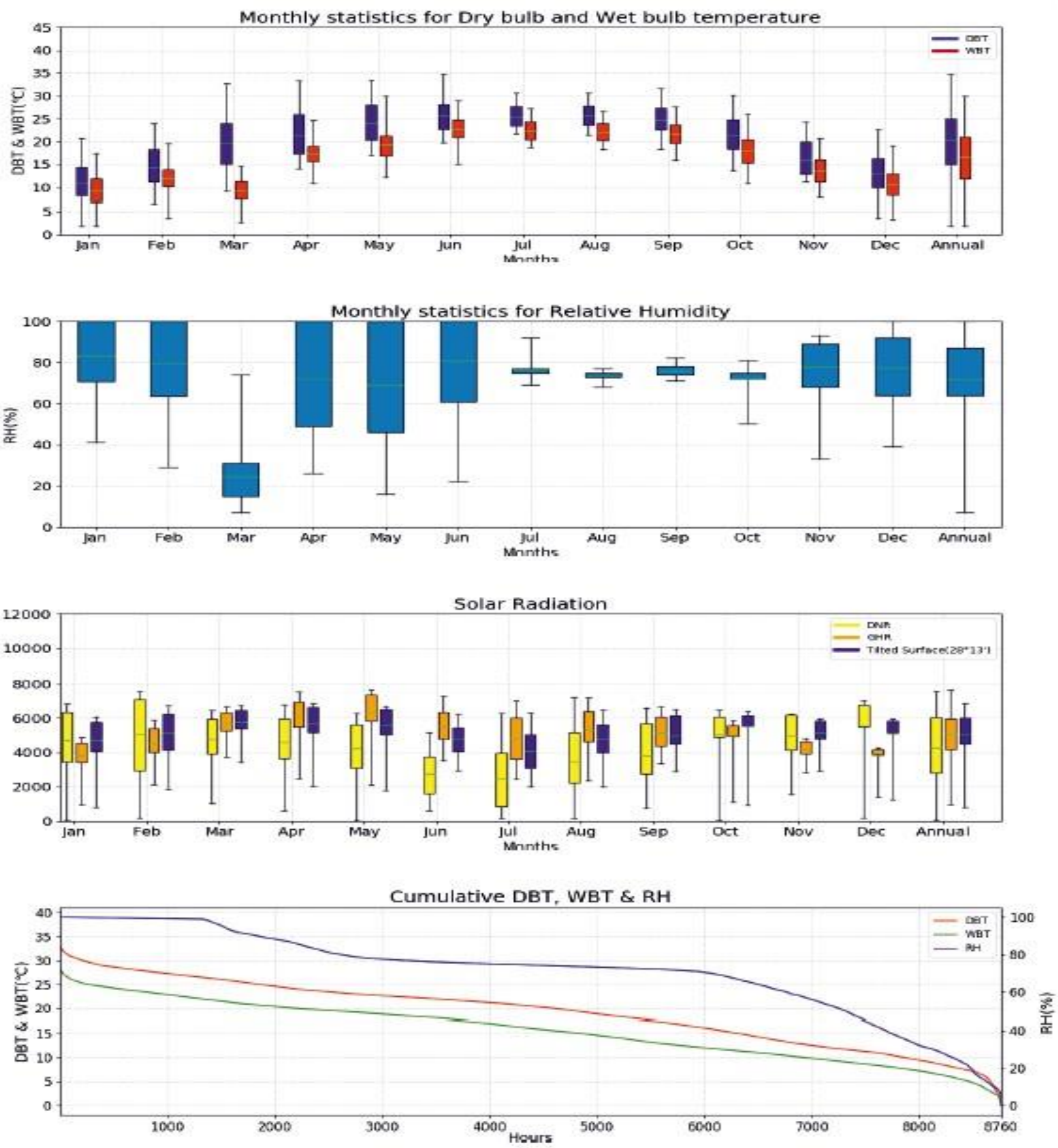


Figure 15: ISHRAE Monthly Weather Data for Pokhara

1.15 Desired Indoor Temperature

For indoor condition, temperature of 72-degree Fahrenheit for cooling application will be used considering section 5.3.2 of Thermal Environmental Conditions for Human occupancy of American Society of Heating, Refrigeration and Air Conditioning (ASHRAE) standard 55-2017.

1.16 Load Calculation Method

CLTD Method

This approach was created as an easier calculation alternative to the transfer function method and the Sol-air temperature method, which are both complex and clumsy.

When employing the CLTD/CLF/SCL approach, the error is often less than 20% over and less than 10% under.

Load Calculation Results

For a cinema hall of area 50ft*40ft and average height 20ft, the cooling load for AHU was calculated using CLTD method. To reach 23-degree Celsius room temperature while homogeneous distribution occurs, the AHU supply temperature should be less than 23 degree Celsius. We assume inlet temperature to be 28 degrees Celsius for our calculation.

Wall Load: 26165 Btu

Roof and floor Load: 17002 Btu

Lighting Load: 6280 Btu

Equipment Load: 4500 Btu

People Load: 67000 Btu

Infiltration Load: 427 Btu

Ventilation Load: 21384 Btu

Duct/Pipe Heat Load: 26280 Btu

Total Load: 282720 Btu

All the calculations are done in Excel. Formulae and mathematical relation that are used are mentioned above section. CLTD method is utilized for load calculation. Total cooling load of cinema hall was obtained to be 23.56 TR from our calculation with standard assumption and reference of actual blue print of the building.

The heat load calculation sheet is attached in the annex.

Ventilation Load

Ventilation load is a system load when outdoor air is routed to the return side of the space. An engineered ventilation load is a space load when a fan pulls outdoor air into a conditioned space.

Here,

Sample space volume = 35112 cubic feet

ACH (Air Change per Hour) = 10

Total required ventilation load = 35112x 10 = 351120 cubic feet per hour

Also, Ventilation load = 5852 cfm

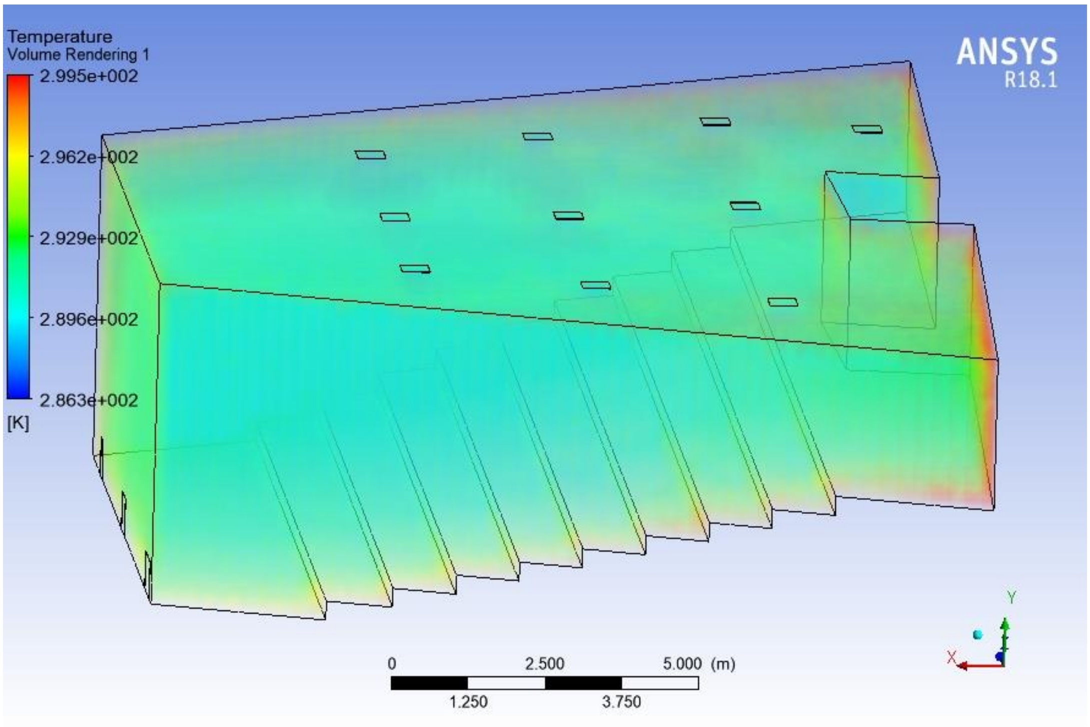
1.17 Thermal and flow Simulation

Thermal simulation of a hall with two design variation is studied for a 351112 cubic feet hall.

Assumptions

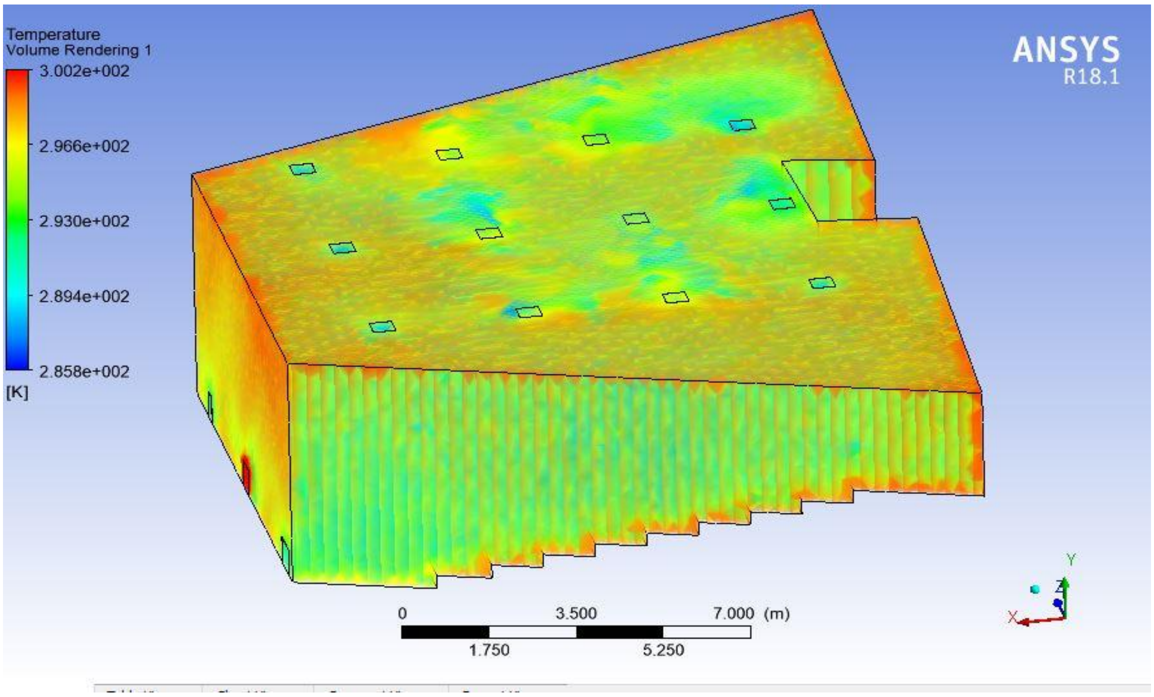
- Inlet temperature of air = 13 ˚C and wall temperature = 27 ºC
- The heat generated from the people inside the hall was considered and taken with reference to total volume i.e., 1 W/m^3
- The temperature profile is shown in pictures with case number
- Reference number for diffuser is taken from front i.e. from left side of pictures.

Case I



Case I contains 10 inlet diffusers where cfm of first six diffuser is 636 (velocity= 1.44 m/s) and cfm of last four diffuser is 509 (velocity= 1.15 m/s)

Case II



<u>Case II contains 12 inlet diffusers where cfm of all diffuser is 487 (velocity= 1.1 m/s) and total cfm is maintained as per requirement and is same as Case I</u>

1.18 Inference

- Case I has better temperature profile than case II.
- Average temperature in case I is approximately 18 °C where as in case II is 25 °C (inlet temperature is maintained constant for both). This signifies with equal energy consumption case I is better.
- Case I has homogeneous temperature distribution, comfort throughout the volume with not change. (People will be comfortable throughout the building)

Analysis

- The temperature profile of option1 and option2 are almost similar in pattern.
- If the velocity of option1 is considered 1.94 m/s (equal to the velocity in option 2 and last option) instead of 1.71 m/s (actual specification), the pattern shows change in 1 degree temperature with homogeneous distribution throughout the hall.
- In the last option, the air flow pattern is not so effective in two corners of the hall which makes it worst possible option among all.

CHAPTER FOUR: THERMAL SIMULATION

To analyze the indoor air conditions of each residential area, the simulation model includes simple geometry of a movie theater.

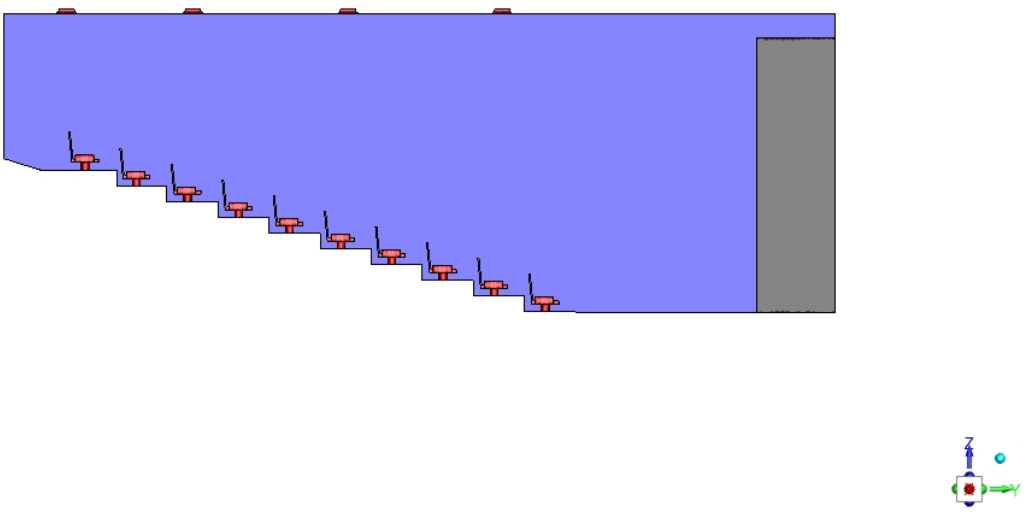


Figure 16- Hall simplified geometry

1.19 Sizing criteria

Typically, the TMY Pokhara weather file will be used to launch the simulation.

Additional source of weather data, based on airport weather data and ASHRAE measurement scenarios (Foundations of ASHRAE Handbook, 2009)

CHAPTER FIVE: CDF SIMULATION

Presently the case CFD is used to validate the CLTD cooling load calculation results, to assess the thermal comfort (including the air flow rate).

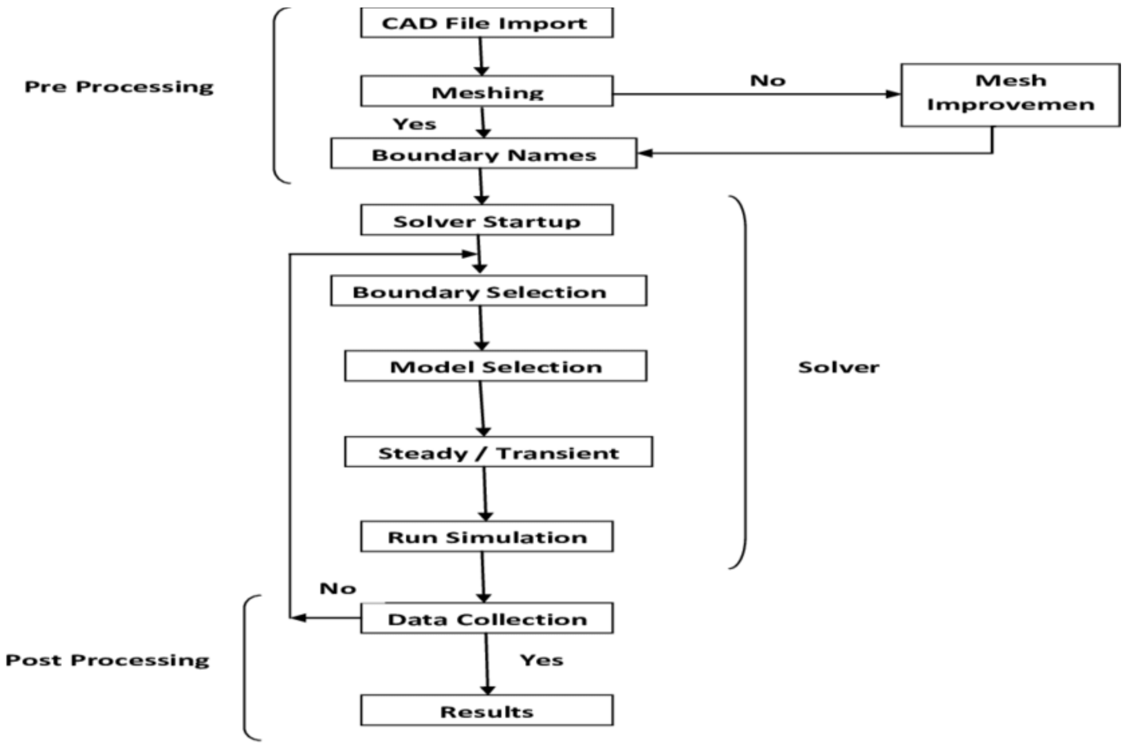


Figure 17 – CFD simulation flow chart

CFD can be used in residential trials in a theater with an environmentally friendly ventilated system. The common disadvantages of this ventilation strategy were obvious: excessive entry speed (average entry rate per seated person was 50% higher than the value used in the estimates presented in later sections of this paper) and consumer complaints of cold ankles. The latest CFD application in the construction of concert hall renovations presented by (Wakeham et al., 2014)

CFD simulation is a time-consuming process that should be simplified by introducing simplification that allows integrated simulation with sufficient time (preferably less than a day in each case).

1.20 CFD Process:

Pre-Processing: Modeling Geometry, Boundary identification and Meshing are the preprocessing part of an ANSYS simulation.

Solution: Solution of the CFD model is essentially the process of solving continuity, momentum, energy and turbulence equation on each cell for every iteration. The solution method are taken as shown below:

Solver: Pressure Based

Velocity Formulation: Absolute

Time: Steady

Gravitational Acceleration: 9.81 m/s^2

Model: Energy (on)

Viscous Model: Standard k-epsilon, Standard Wall function

Material: Fluid material was standard air properties

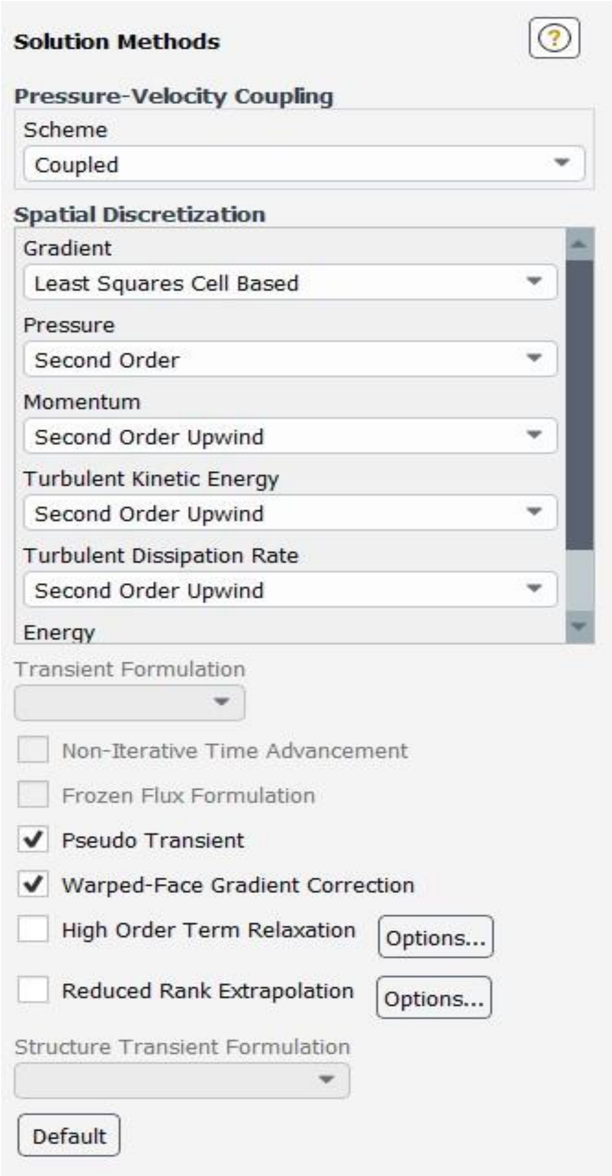


Figure 18: Solution Methods

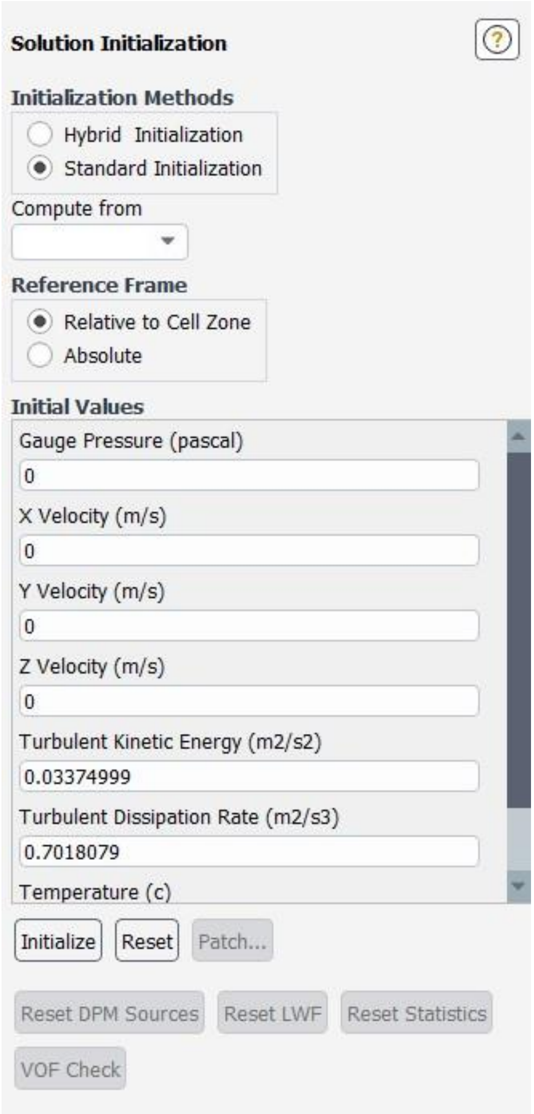


Figure 19: Solution Initialization

Post-Processing: The post-processing includes data and result displaying techniques which is shown below based on cases studied.

1.21 Geometry

The geometry used in the simulation is done in Solid works. Geometry is the preprocessing part of ANSYS simulation.

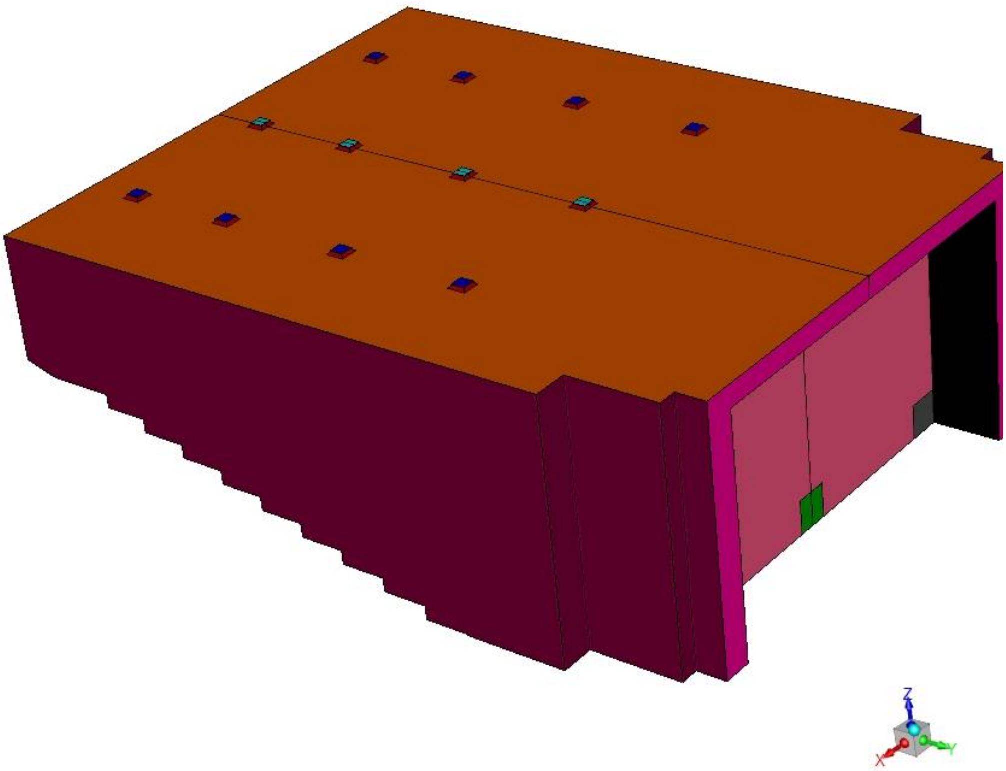


Figure 20: The symmetry of the hall corresponding to the longitudinal axis is enabled to use a single-component model of the room.

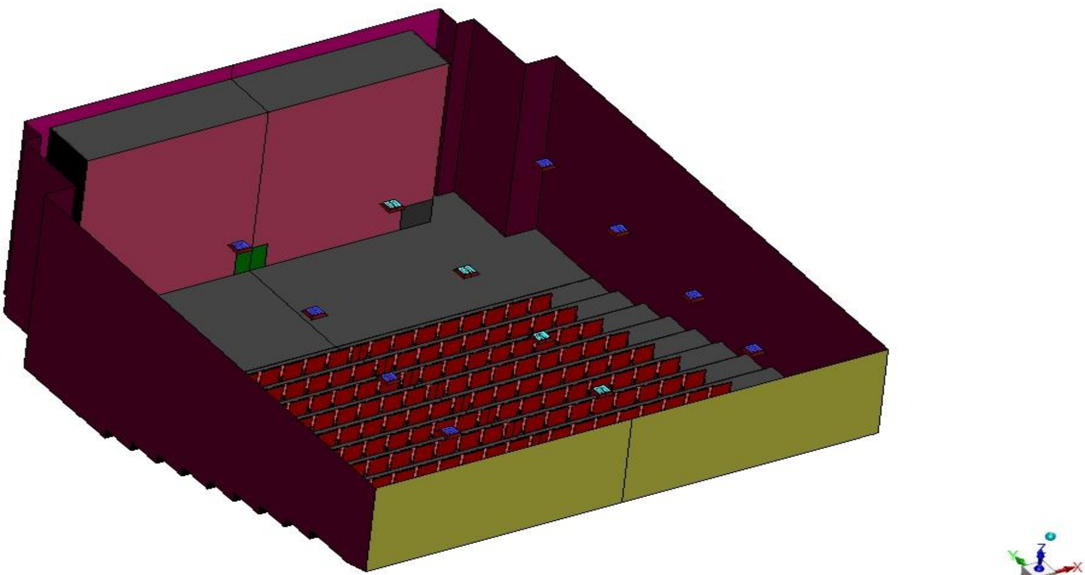


Figure 21: Cinema Hall with sitting arrangements

1.22 Meshing

The act of transforming irregular forms into more identifiable volumes called "elements" is known as meshing.

Meshes made out of triangular or quadrilateral cells (or a mix of the two) in 2D and tetrahedral, hexahedral, polyhedral, pyramid, or wedge cells (or a combination of these) in 3D can be used in ANSYS FLUENT. The mesh type you use will be determined by the application.

Meshing can influence the run time, computational expenses and numerical diffusion of the simulation.

We use tetrahedral meshing for our simulation. Initial cell count was 1.2 million. After polyhedral mesh was used the cell count was 1 million which reduced the computational time without compromising the accuracy.

Mesh Size:

Cells: 997218

Faces: 997218

Nodes: 4870738

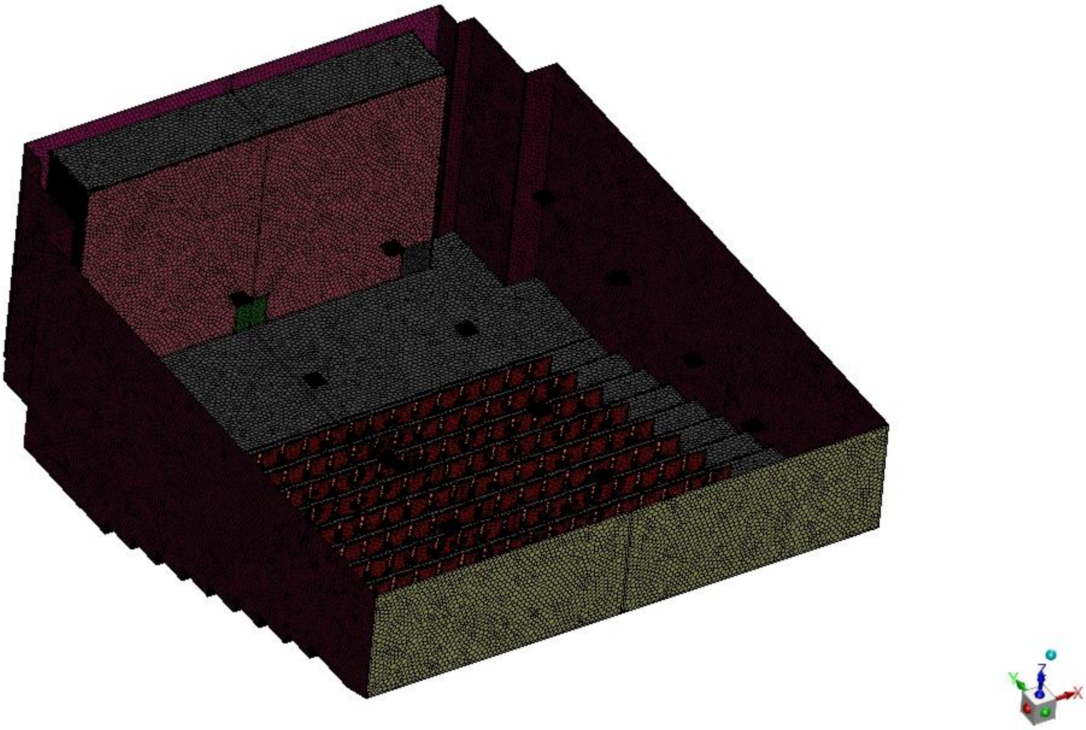


Figure 22 – CFD model mesh and grid generation.

1.23 CFD simulation scenarios

Internal loads, such as occupants are modeled as blocks with a constant heat flux source. Parametric variation is checked for the identification of the best temperature and velocity scenario at the inlet.

Different Cases studied with ANSYS are as follows:

Case 1:

This is the case of mixing ventilation where the inlet from the ceiling is thrown by 12 diffusers and air exits through the exit at the floor level of the hall.

12 inlet diffuser and 3 outlets were used as is shown in the figure below.

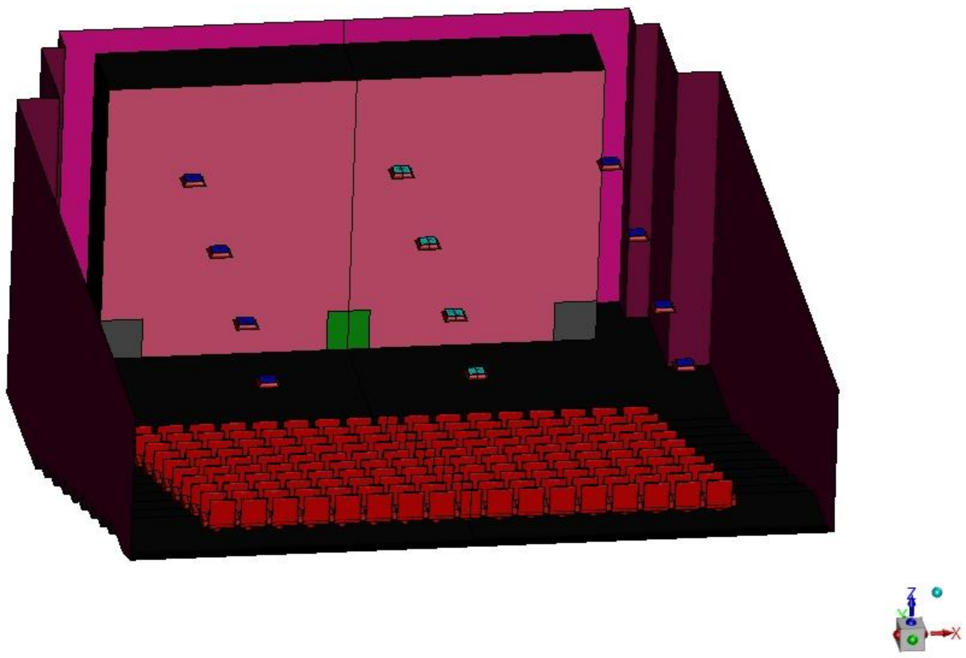


Figure 23: Geometry for case 1 and case 2

Inlet boundary condition:

Inlet temperature: 18℃

Inlet velocity: 3m/s

Simulation result:

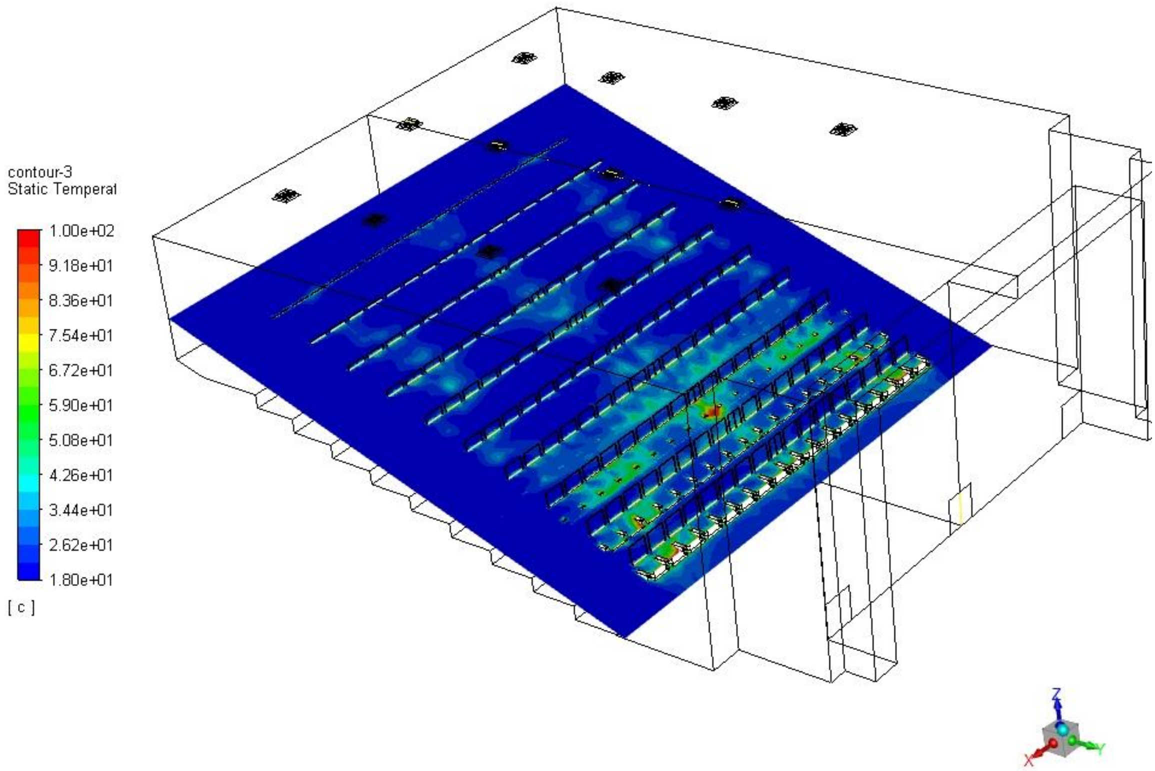


Figure 24: Temperature and velocity profile is studied at 1.8m height from floor

The result of simulation is focused on the region of interest i.e., the human occupant space of the hall. The inclined plane parallel to sitting chairs' plane and 1.8 m (average human occupied height) from floor level is considered for temperature and velocity study(Turner et al., 1979).

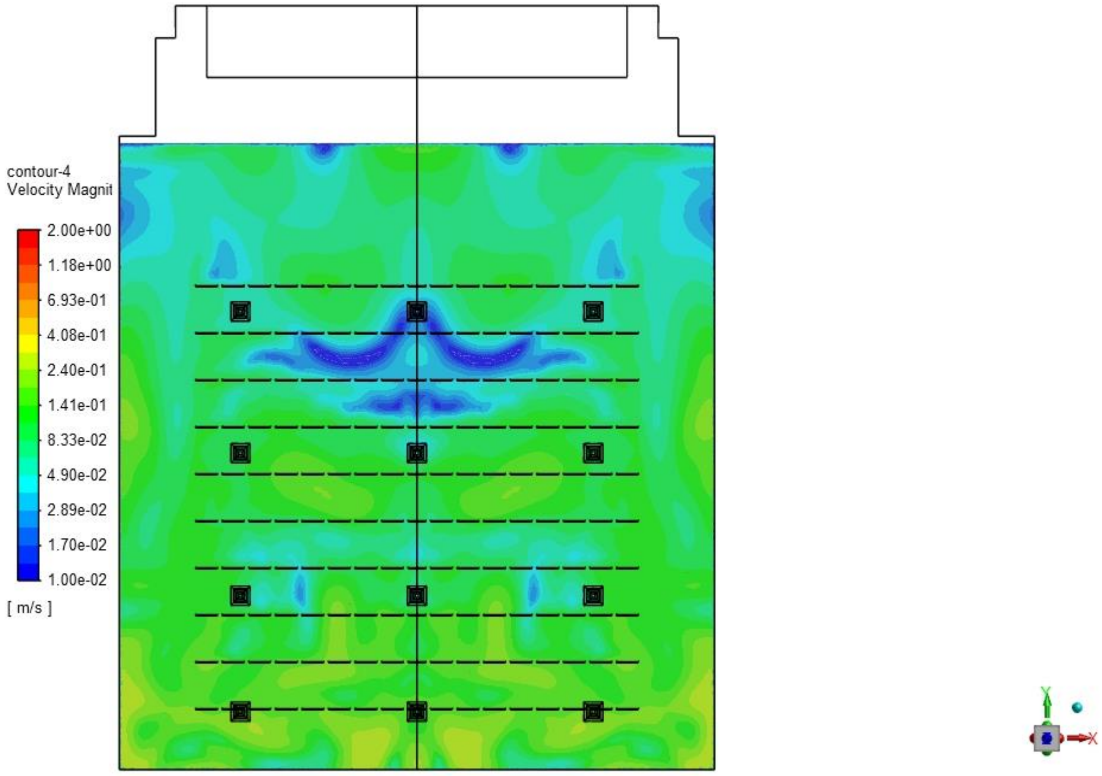


Figure 25: Average velocity in plane of interest: 0.0933 m/s

This average velocity is too low for summer as recommended by ASHRAE. The effect of velocity on the occupants will be negligible.

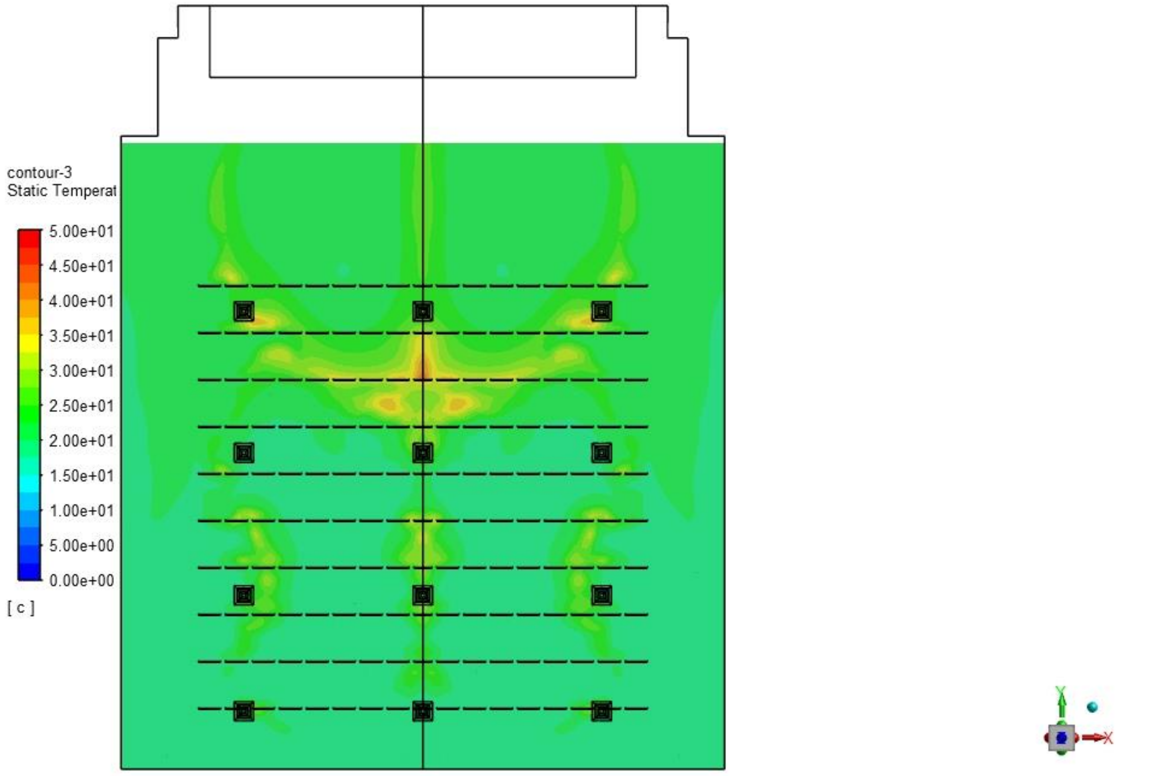


Figure 26: Average temperature in the plane of interest: 23.17℃

Although this temperature falls in the region as recommended by ASHRAE for summer cooling purpose, the lower velocity makes this configuration of ventilation less preferred.

Case 2:

Mixing ventilation arrangement as in the earlier case.

Same diffuser inlet and outlet arrangements (12 inlet diffuser and 3 outlets) are used as in case 1. Only the inlet temperature and velocity are altered.

Inlet temperature: 16°C

Inlet velocity: 4m/s

Simulation result:

Temperature and velocity are studied in the same plane of interest as earlier case.

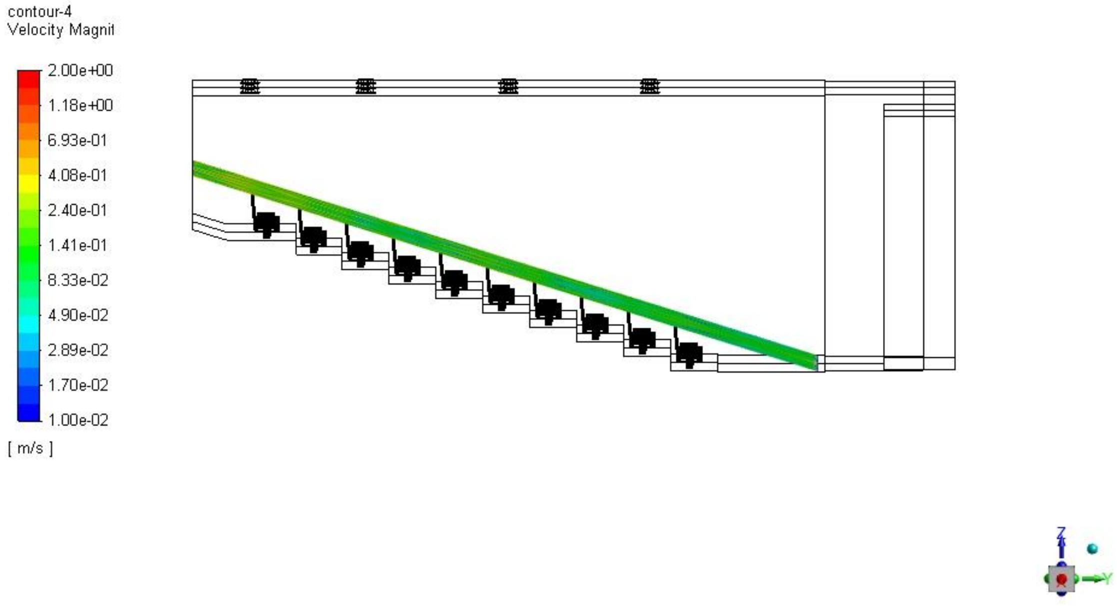


Figure 27: Plane in Z-direction where temperature and velocity distribution is under study

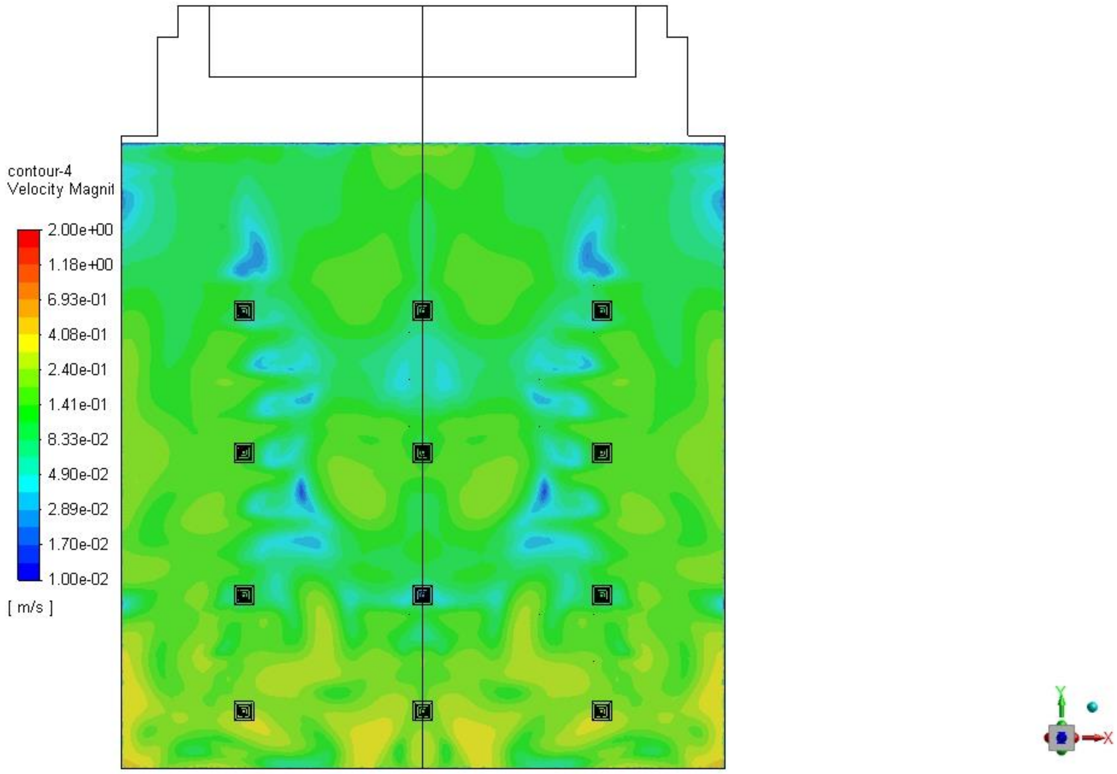


Figure 28: Average velocity in plane of interest: 0.124 m/s.

This average velocity is below the recommended air velocity range during summer according to ASHRAE.

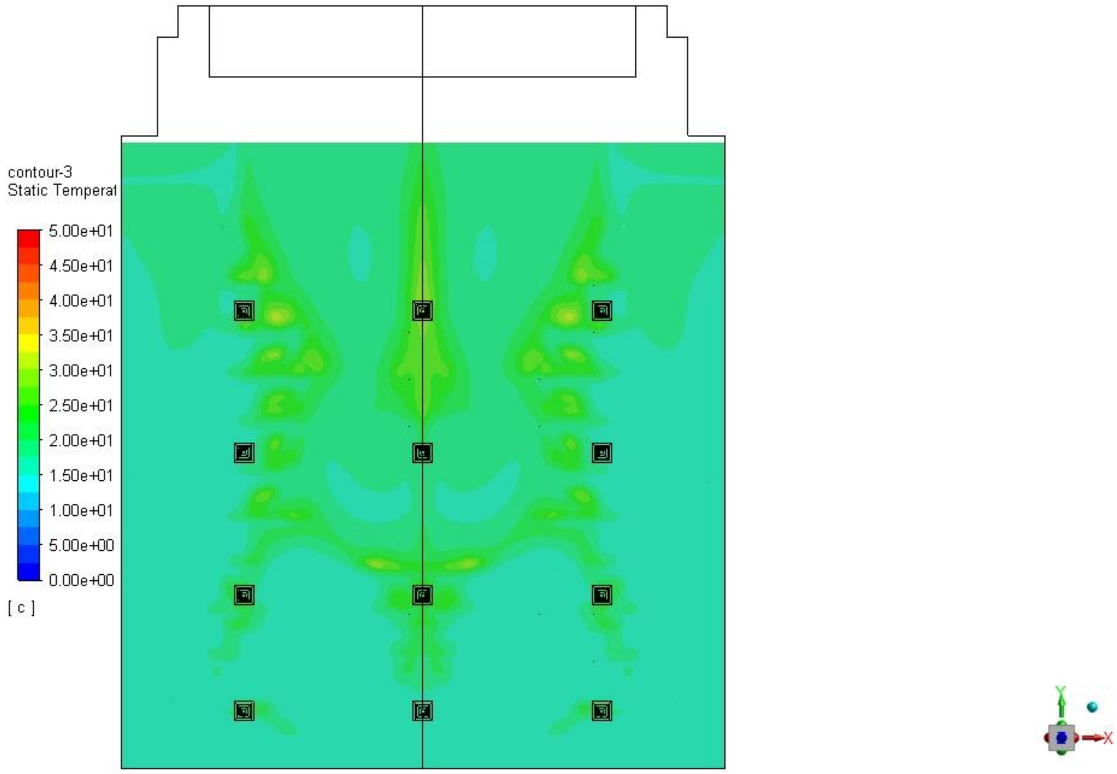


Figure 29: Average temperature in plane of interest: 20.35℃.

This average air temperature is a little bit cold and also because the air velocity is below the recommended range, this arrangement is less preferred.

Case 3:

Displacement ventilation arrangement is used with three inlets at the front floor level and three outlets at the back ceiling level of the hall.

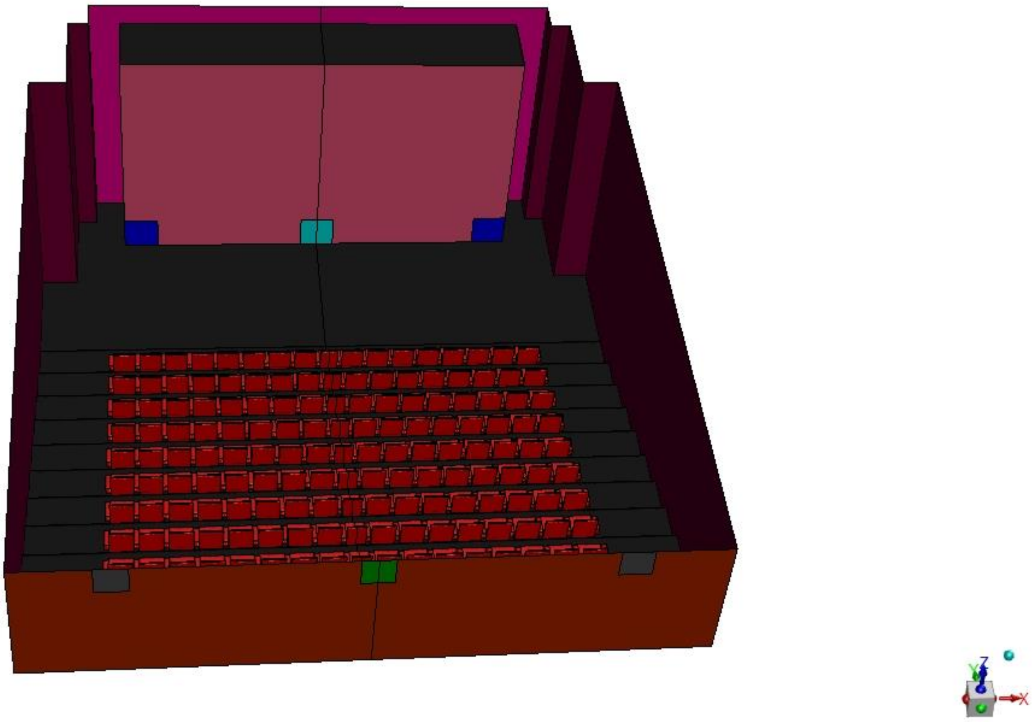


Figure 30 : Geometry and diffuser arrangement for case 3 and case 4

Inlet temperature: 16℃

Inlet velocity: 4m/s

Simulation result:

Temperature and velocity are studied in the same plane of interest (the region from floor to human head level).

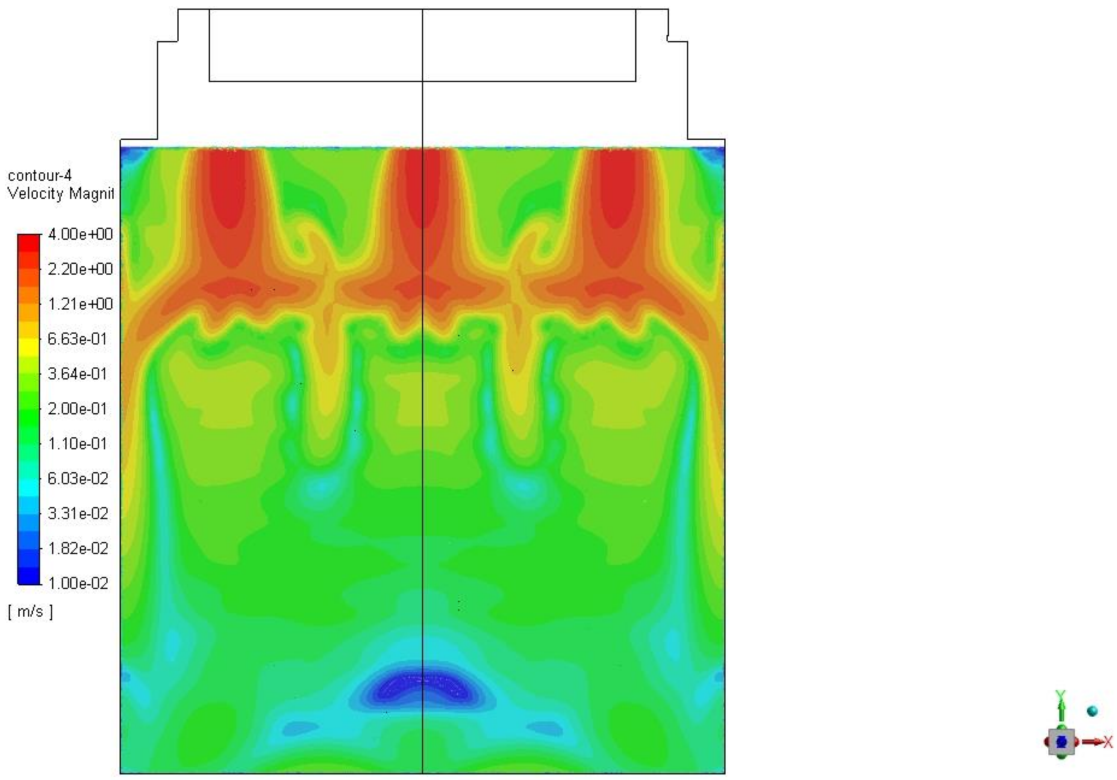


Figure 31: Average 1velocity in the plane of interest: 0.49 m/s.

This air velocity is very higher than as recommended by ASHRAE for summer cooling applications.

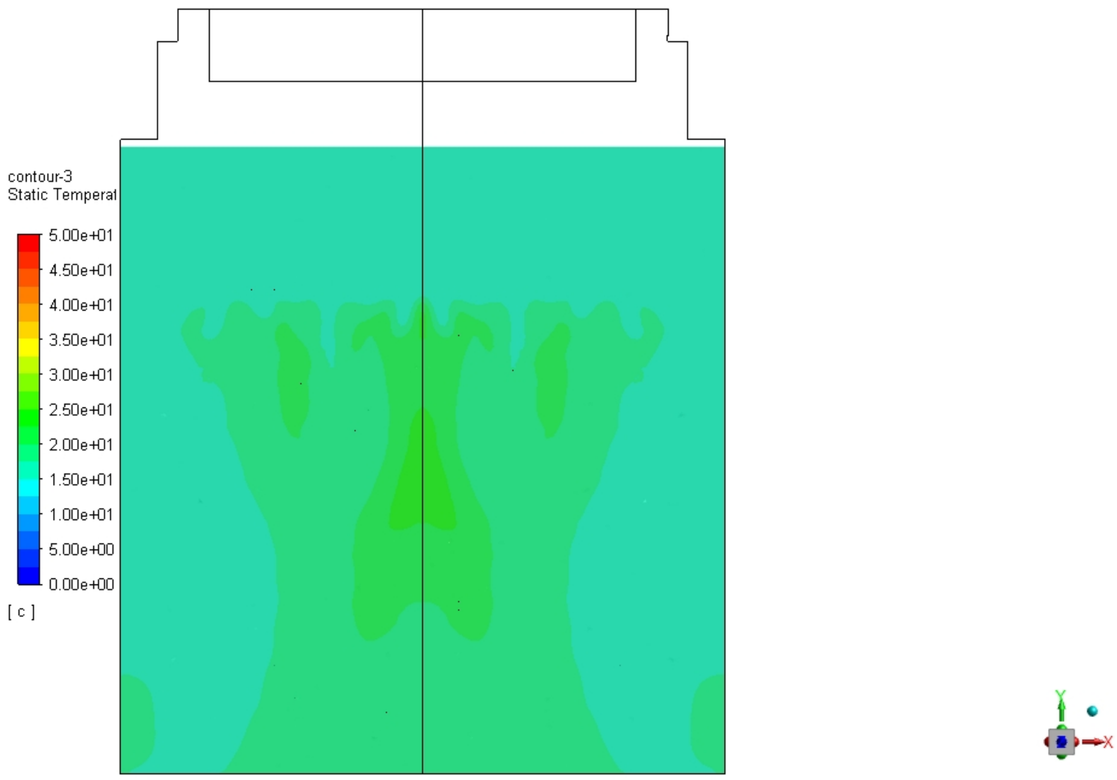


Figure 32: Average temperature in plane of interest: 17.7℃.

This average temperature is less than the recommended temperature range by ASHRAE for summer cooling application. In fourth case we will decrease the inlet velocity and increase the inlet temperature and see if this will make the average temperature and velocity in the region of interest will fall in the recommended range for summer cooling application.

Case 4:

Displacement ventilation arrangement is used with three inlets at the front floor level and three outlets at the back ceiling level of the hall as was in case 3.

Inlet temperature: 18°C

Inlet velocity: 2m/s

Simulation result:

Temperature and velocity are studied in the same plane of interest (the region from floor to human head level).

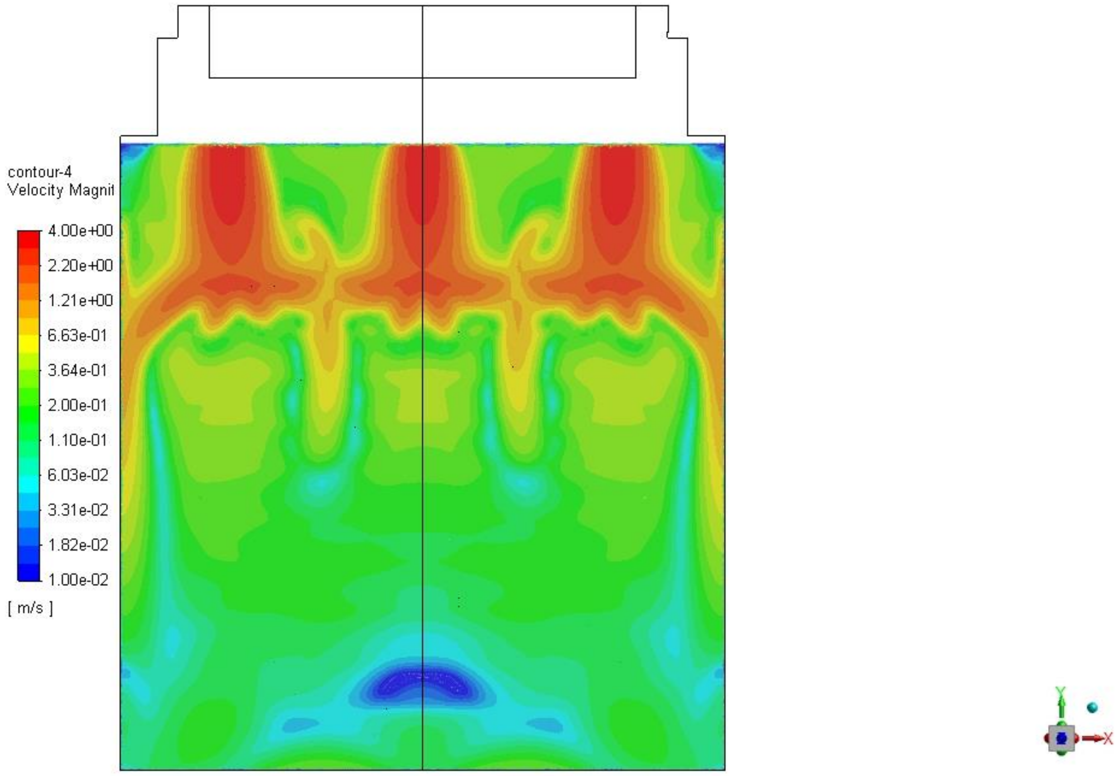


Figure 33: Average velocity in the plane of interest: 0.24 m/s

This average air velocity is close to ASHRAE recommended range for summer cooling application. But the variation of the air velocity is evident at the front and end rows of the occupants/chairs. Also the average air velocity seems to be quite higher than the value recommended by ASHRAE chart for the corresponding inside temperature. This problem is addressed in the next case.

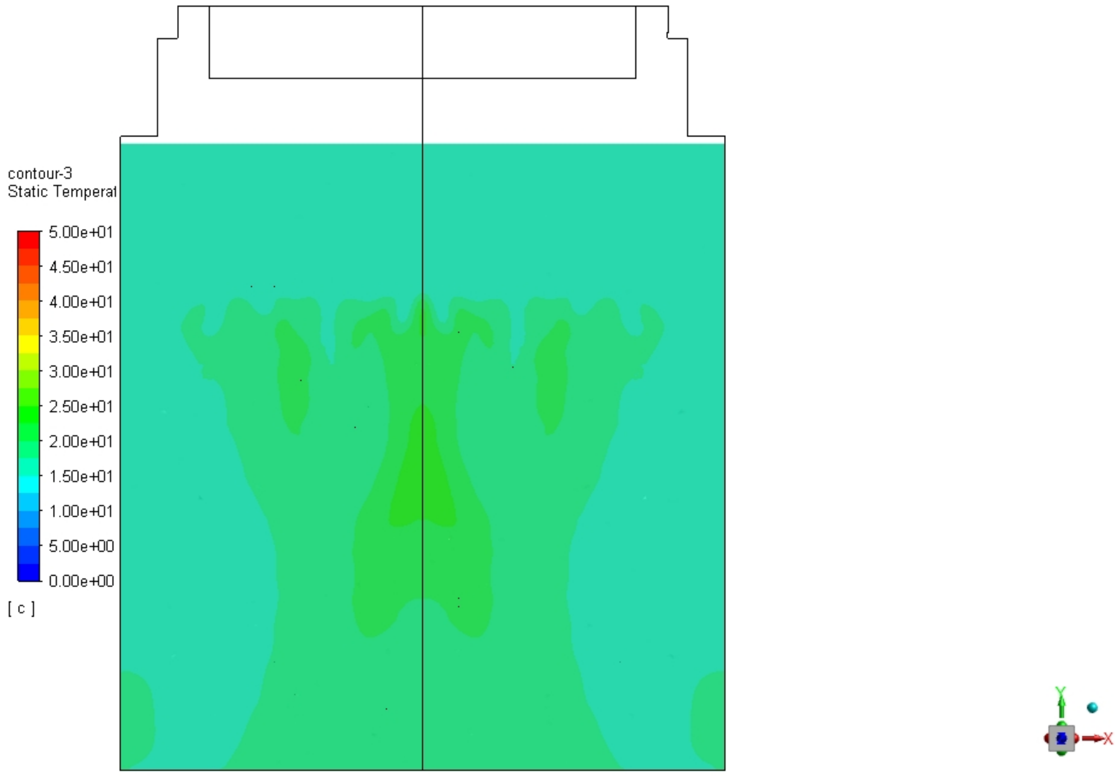


Figure 34: Average temperature in plane of interest: 23.56 ℃.

This average temperature and velocity is good according to the ASHRAE recommendation. Thermal comfort for the given hall will be maximum for displacement ventilation arrangement with this inlet parameter in terms of average velocity but the local air velocity for the front row audiences is higher and it creates local discomfort.

Case 5:

The configuration for this system is as shown in the figure below:

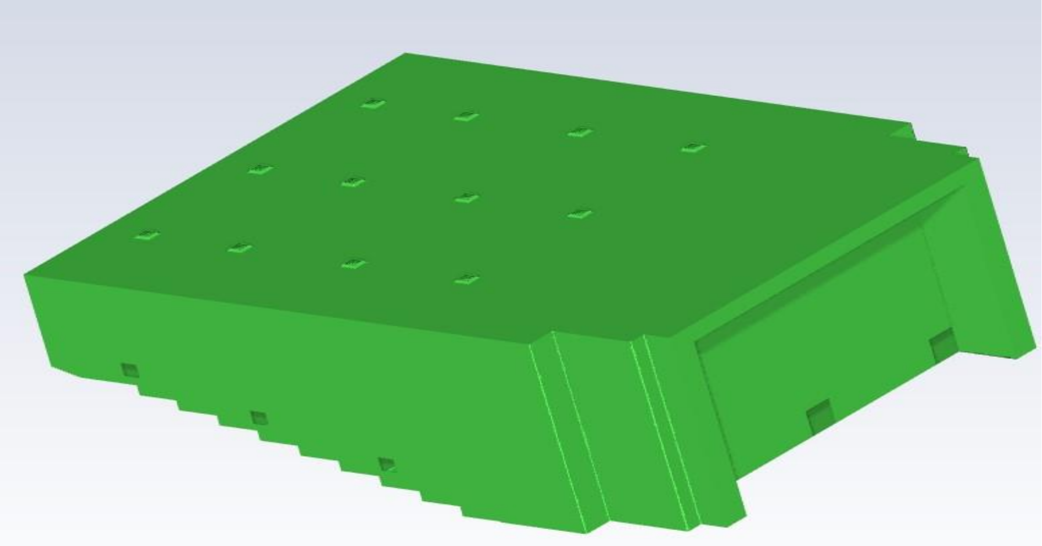


Figure 35: airflow configuration for case 5

The air flow inlet to the hall is from the from the four column and three row arrangements of 12 ceiling diffusers, and the air outlet is free stream exit from the three exists in each side walls and three exits at the floor level below the screen.

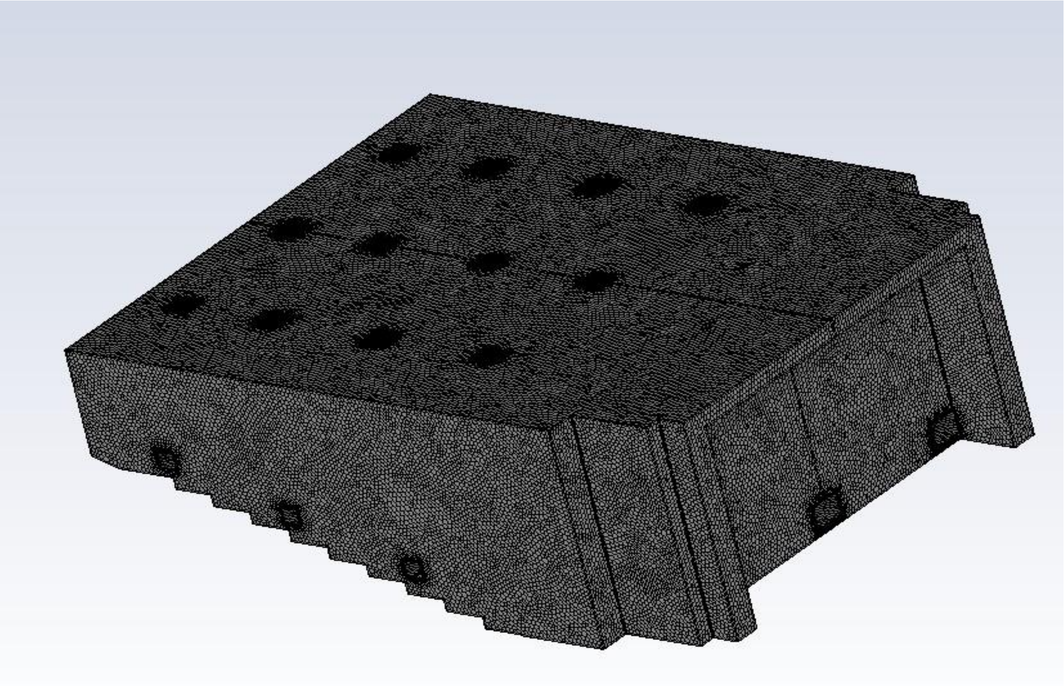


Figure 36: mesh display for case 5

Inlet Temperature: 20℃

Inlet Velocity: 4m/s for the two front columns of diffusers

3m/s for the last two columns of diffusers

CFD Results:

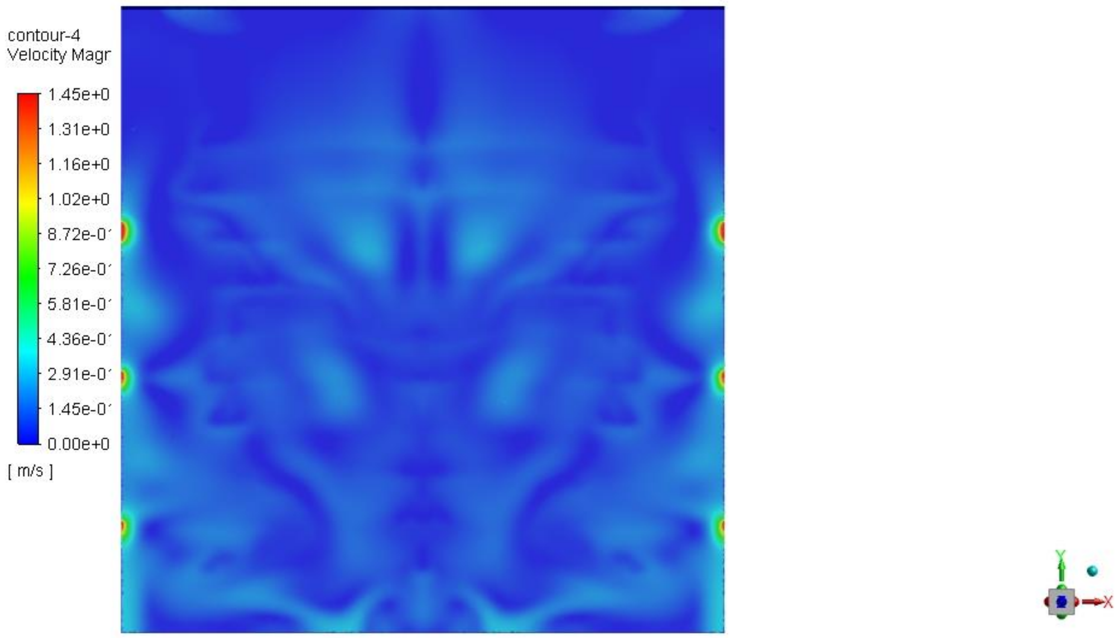


Figure 37: velocity distribution for case 5

As we can see from the velocity distribution above, the velocity at the plane of interest i.e. 1.8m above the floor inclined parallel to the sitting inclination, the average air speed is 0.157m/s and the variation for all occupant region is less. Hence, this velocity configuration is highly preferred. From the ASHRAE standard chart it is evident that this velocity is the optimum velocity for achieving higher thermal comfort.

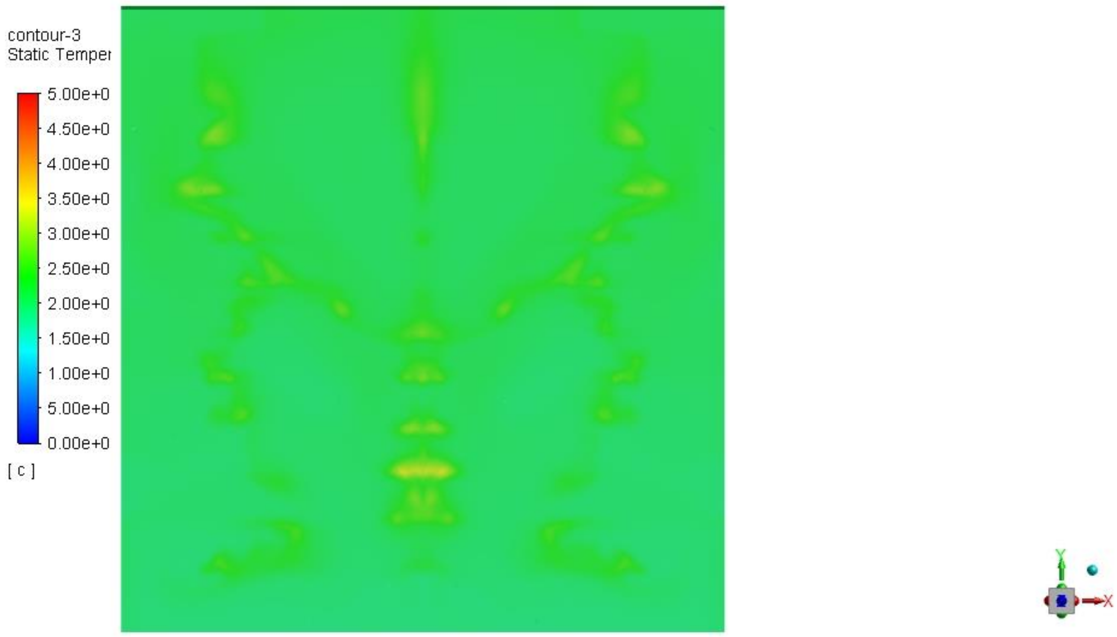


Figure 38: temperature distribution at the plane of interest

The above temperature distribution shows there is low variation. The average temperature at 1.8m above the floor level is 22.3°C which is fine according to ASHRAE.

The temperature at other planes at the ankle and head for the sitting posture as specified by ASHRAE 0.1m and 1.1m above the floor is also studied to identify the percentage of occupants dissatisfied due to vertical temperature difference at the head and feet level.

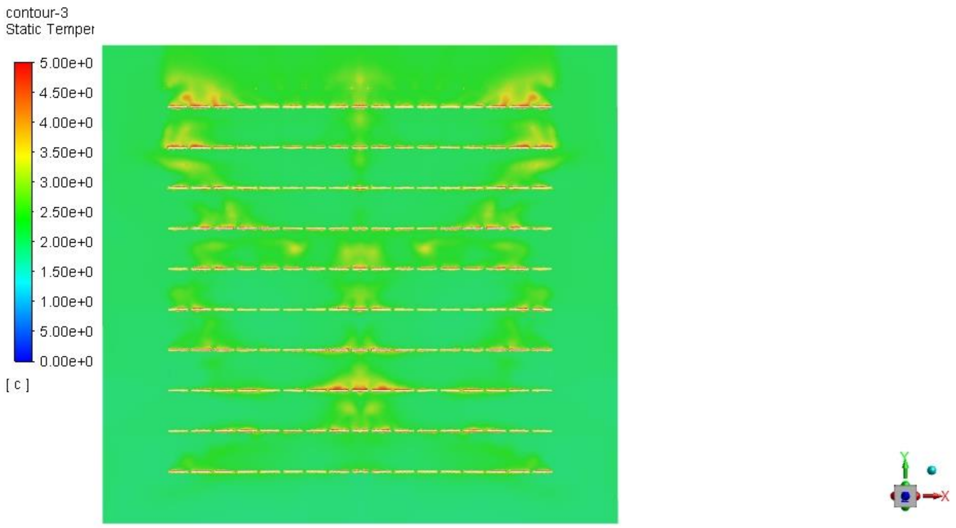


Figure 39: temperature profile at the head level with average temperature 22.69℃

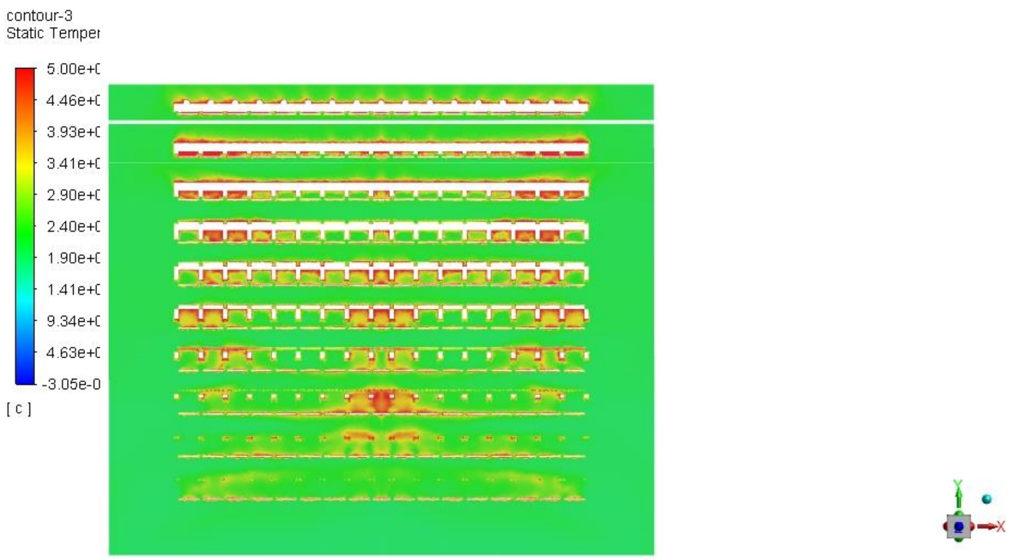


Figure 40: temperature profile at the feet level with average temperature 23.14℃

High vertical temperature gradient is a sign of higher number occupants being dissatisfied as recommended by ASHRAE. The temperature difference for this case is less than 2℃ which means less than 5% of the occupants are dissatisfied hence this arrangements and configuration is highly preferred.

1.23.1 CFD Results

Cases	Average velocity (m/s)	Average Temp (℃)
Case-1	0.09	23.17
Case-2	0.124	20.35
Case-3	0.49	17.7
Case-4	0.24	23.56
Case-5	0.157	22.30

The average temperature and velocity at the plane of interest is shown in the table above. The best case in terms of temperature and velocity at the human occupied space seems to be the fifth one based on standards established by ASHRAE.

Not only average velocity, local discomfort occupant percentage is also low for the last case i.e. low variation in temperature and velocity distribution. Vertical temperature difference is also in the acceptable region as discussed above. Hence this method can be adopted.

CHAPTER SIX: RESULTS AND DISCUSSION

Selection criteria for the arrangements and parameter for the inlets are local temperature and air speed at the occupant region. These value can be compared with the ASHRAE standard and it there parameter fall in the acceptable range, the requirements regarding ventilation system design can be achieved

The result of the simulation indicates that parametric control to both designs can improve the thermal comfort to the occupant space for cooling application. The first case of the mixing ventilation is better than the second case in term of average temperature distribution but velocity is low.

The fifth case of ventilation arrangements is the most appropriate for the optimal comfort according to ASHRAE both in terms of temperature and velocity best suited for the occupants.

The efficiencies of the system may be better in the second and third case in terms of temperature dropped but this temperature is too low for sound thermal environment. If we take best thermal environment for human comfort to be the function of temperature and velocity of air, the fifth case is the best scenario. Human occupied region of the hall will have best thermal condition in the case of ventilation with the inlet temperature and velocity corresponding to the fifth case for the given hall.

During ventilation design for a space, best configuration and parameter for inlets can be optimized based on CFD results without the need for expensive and time consuming experiments.

.

CHAPTER SEVEN: CONCLUSION

Proper HVAC balancing is a process that incorporates different input parameter that can set final outcome. The current situation's measurement findings show that the accurate modeling of the air flow pattern and the specification of the design size have a major influence on system performance, which is maximum thermal comfort in the occupancy zone.

Interior flow fields and internal temperatures were simulated using CFD to explain how to pick a desirable design configuration. Temperature and velocity distribution was obtained for the simple simulation could help identify the diffuser size and AHU sizing and load.

Comparison of the predicted effects of temperature in the habitat showed a good agreement between mathematical calculation and CFD results.

CHAPTER EIGHT: RECOMMENDATION

Results of the simulation was commensurate with the analytical load calculated from CLTD (analytical/empirical method) with no consideration to humidity conditions which is integral part of human comfort in a closed environment. The simulation study could also consider the analysis based on infiltration and equipment load. Moreover, the study at different occupancy level (for low attendance to full-house conditions) can also be done.

The results could also have been tested with the experimental outcomes, which could further validate the result obtained in this research.

The confirmation from all experimental result is an expensive process, but it is recommended to conduct one.

REFERENCES

Thermal comfort in buildings, Designing Buildings Wiki, 2016

ASHRAE. (n.d.). *American Society of Heating Refrigerating and Air Conditioning Engineers (ASHRAE). (1992) Thermal environmental conditions for human occupancy. (ASHRAE Standard 55-1992).*

Editorial Simscale. (2017). *Displacement ventilation system*. http://www.google.com/patents/WO2012093938A1

Fisk, W. J. (2002). How IEQ affects health, productivity. *ASHRAE Journal*, *44*(5), 56–58.

Hangan, H., McKenty, F., Gravel, L., & Camarero, R. (2001). Case study: Numerical simulations for comfort assessment and optimization of the ventilation design for complex atriums. *Journal of Wind Engineering and Industrial Aerodynamics*, *89*(11), 1031–1045. https://doi.org/https://doi.org/10.1016/S0167-6105(01)00097-6

Heschong, L. (2002). Daylighting and human performance. *ASHRAE Journal*, *44*, 65–67.

Humphreys, M. A., & Nicol, J. F. (1998). Understanding the adaptive approach to thermal comfort. *ASHRAE Transactions*, *104*(Pt 1B), 991–1004.

Jones, B., & Kirby, R. (2012). Indoor Air Quality in U.K. School Classrooms Ventilated by Natural Ventilation Windcatchers. *International Journal of Ventilation*, *10*(4), 323–337. https://doi.org/10.1080/14733315.2012.11683959

Kavgic, M., Mavrogianni, A., Mumovic, D., Summerfield, A. J., Stevanovic, Z., & Đurović-Petrović, M. (2010). A review of bottom-up building stock models for energy consumption in the residential sector. *Building and Environment*, *45*, 1683–1697. https://doi.org/10.1016/j.buildenv.2010.01.021

Liping, W., & Hien, W. N. (2007). The impacts of ventilation strategies and facade on indoor thermal environment for naturally ventilated residential buildings in Singapore. *Building and Environment*, *42*(12), 4006–4015. https://doi.org/10.1016/J.BUILDENV.2006.06.027

Mateus, N., & Da Graca, G. C. (2013). Thermal and airflow simulation of the gulbenkian great hall. *Proceedings of BS 2013: 13th Conference of the International Building Performance Simulation Association*, 1835–1842.

Muncey, R. W. R. (1979). *Heat transfer calculations for buildings / [by] R. W.R. Muncey*. Applied Science Publishers.

Olesen, B. W. Brager, G. S. (2004). A better way to predict thermal comfort. *ASHRAE Journal*, *August*, 20–26. https://escholarship.org/uc/item/2m34683k%0Ahttp://repositories.cdlib.org/cedr/cbe/ieq/OlesenBrager2004_comfort/

Parsons, K. (2014). Human Thermal Environments: The Effects of Hot, Moderate, and Cold Environments on Human Health, Comfort, and Performance, Third Edition. In *Human Thermal Environments: The Effects of Hot, Moderate, and Cold Environments on Human Health, Comfort, and Performance, Third Edition*. https://doi.org/10.1201/b16750

SimScale CAE Forum. (2016). *Tutorial_ Thermal Comfort Parameters for HVAC Simulations - Using SimScale _ Post-processing - SimScale CAE Forum*.

Turner, S. C., Paliaga, G., Lynch, B. M., Arens, E. A., Aynsley, R. M., Brager, G. S., Deringer, J. J., Ferguson, J. M., Filler, J. M., Hogeling, J. J., Int-hout, D., Kwok, A. G., Levy, H. F., Sterling, E. M., Stoops, J. L., Taylor, S. T., Tinsley, R. W., Cooper, K. W., Dean, K. W., … Peterson, J. C. (1979). American society of heating, refrigerating and air-conditioning engineers. *International Journal of Refrigeration*, *2*(1), 56–57. https://doi.org/10.1016/0140-7007(79)90114-2

Wakeham, M. K., Kuhn, E. M., Lee, K. J., McCrory, M. C., & Scanlon, M. C. (2014). Use of tracheostomy in the PICU among patients requiring prolonged mechanical ventilation. *Intensive Care Medicine*, *40*(6), 863–870. https://doi.org/10.1007/s00134-014-3298-4

Yamasawa, H., Kobayashi, T., Yamanaka, T., Choi, N., Cehlin, M., & Ameen, A. (2021). Applicability of displacement ventilation and impinging jet ventilation system to heating operation. *Japan Architectural Review*, *4*(2), 403–416. https://doi.org/10.1002/2475-8876.12220

Yao, Y., Lu, Y., Zhang, F., Liu, L., & Liao, M. (2020). Analysis of Seasonal Daytime Urban Thermal Environment Dynamics in a Tropical Coastal City Based on the Spatiotemporal Fusion Model. *Advances in Meteorology*, *2020*, 8182676. https://doi.org/10.1155/2020/8182676

Zhang, C., Heiselberg, P., & Nielsen, P. V. (2014). Diffuse ceiling ventilation - A review. *International Journal of Ventilation*, *13*(1), 49–63.

Zhang Lin. (2014). Stratum Ventilation – A Solution to Meet Challenges to Contemporary Air Distribution. *REHVA Journal*, *4*, 40–43. http://www.rehva.eu/

ANNEX 1

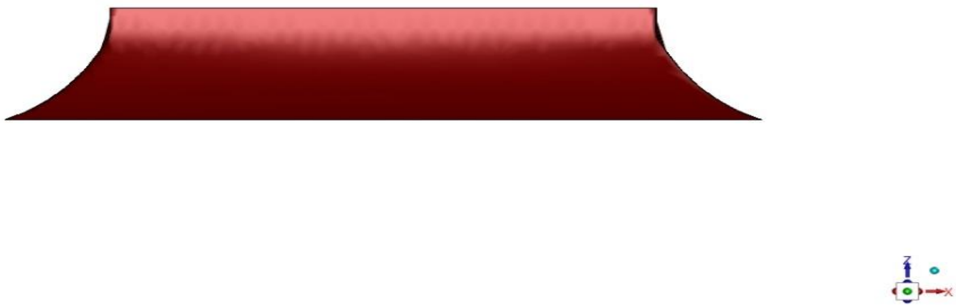


Figure 35: Diffuser used in the simulation.

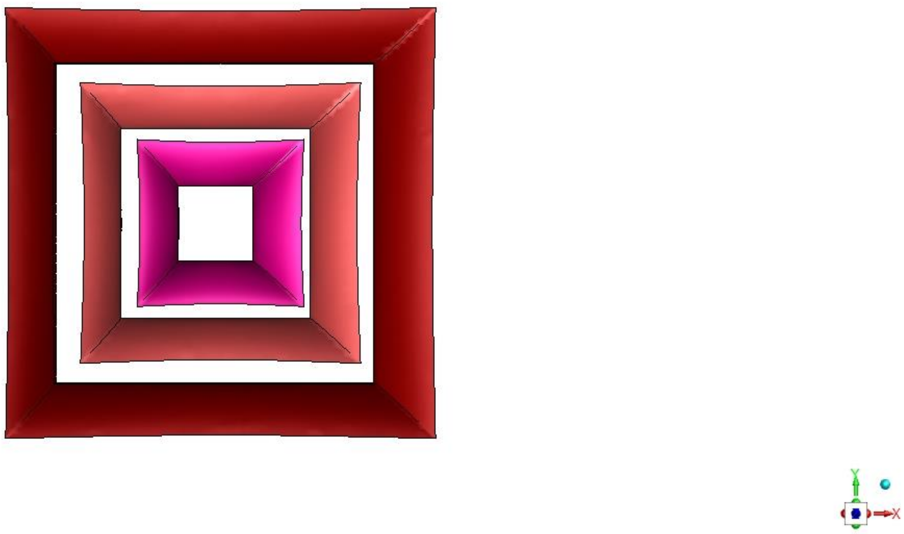


Figure 36: Diffuser top view

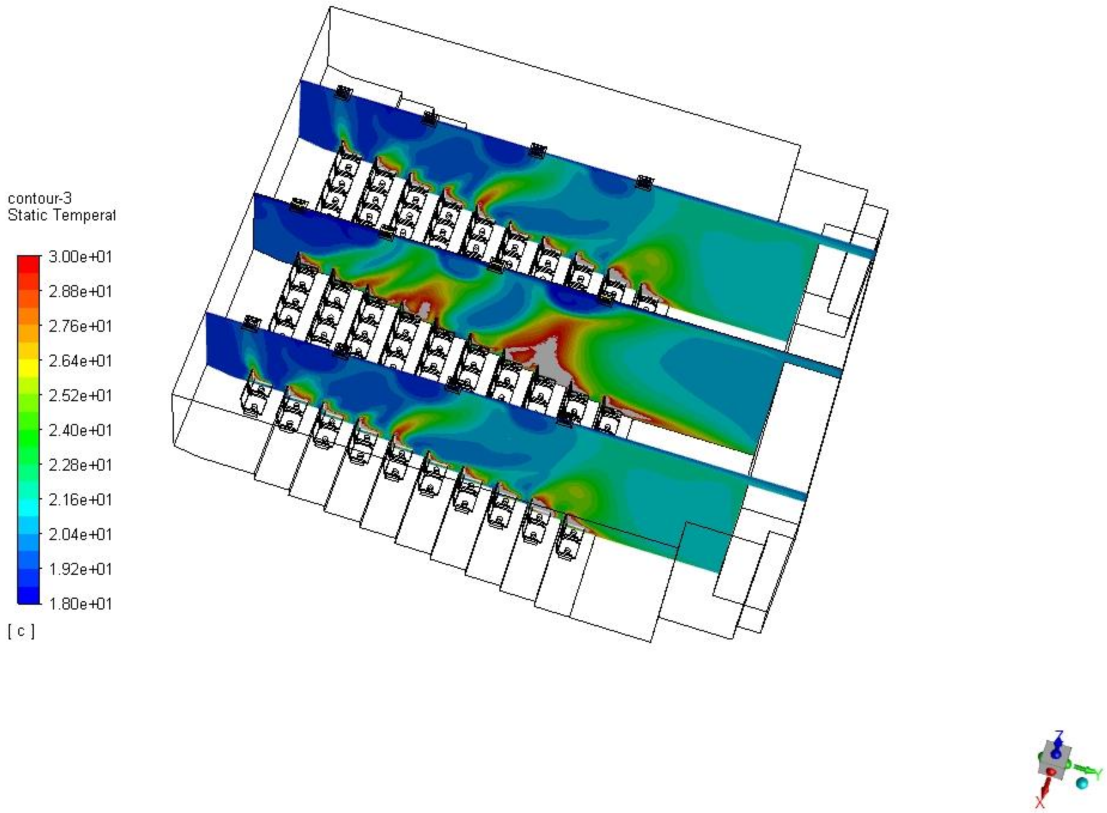


Figure 37: Temperature Distribution for three planes (isotherms) in x-direction of the mixing ventilation design's first case.

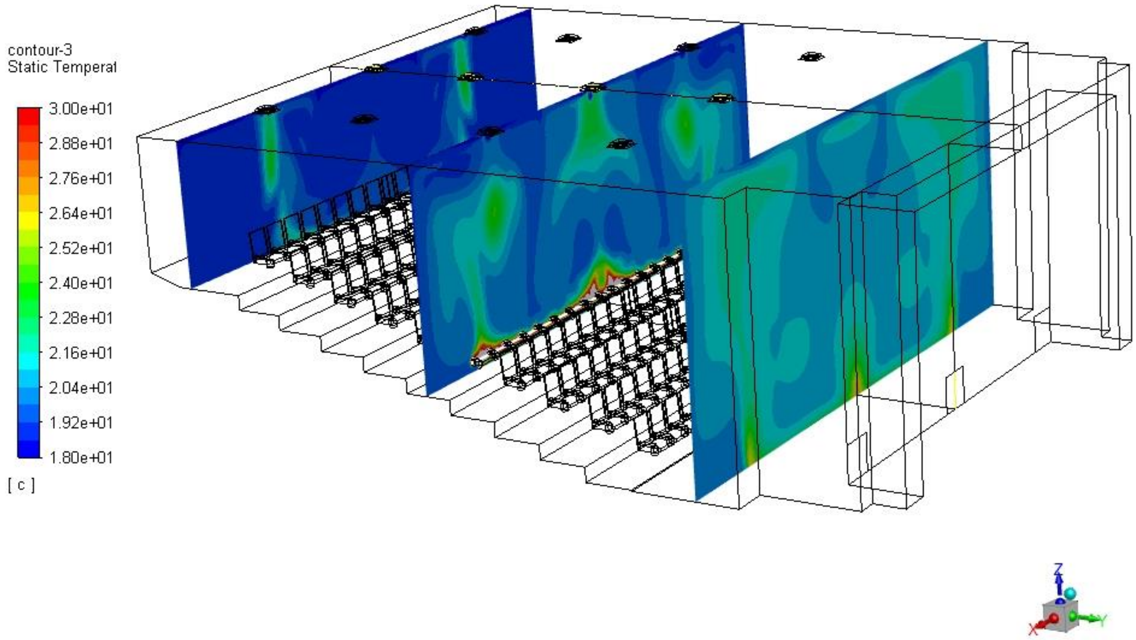


Figure 38: Temperature distribution in three planes(isotherms) for mixing ventilation design's first case.

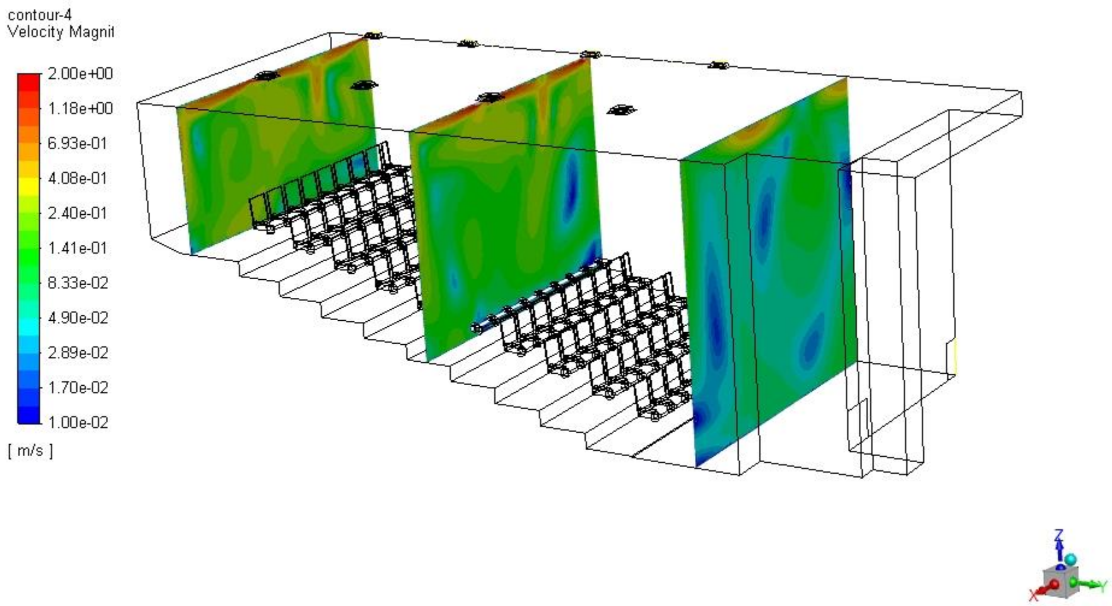


Figure 39: Velocity profile in three planes (isotherm) in Y-direction for the mixing ventilations design’s first case

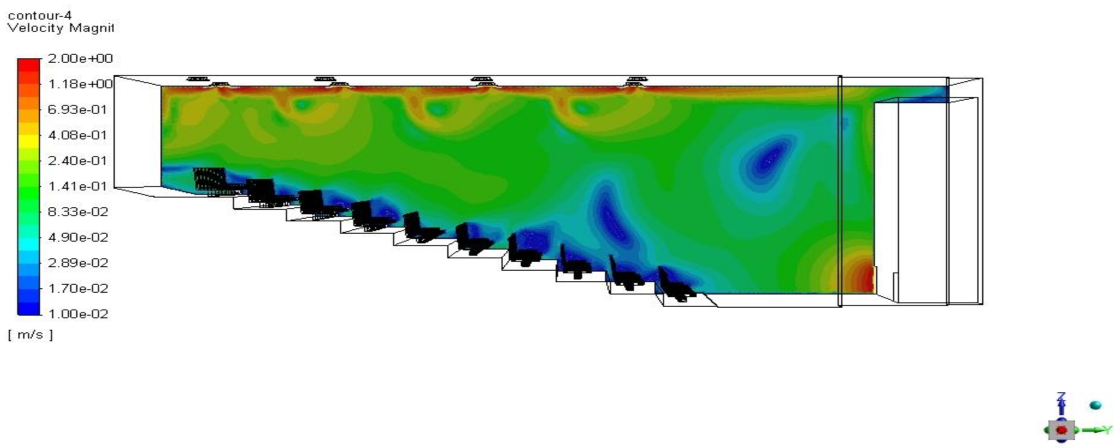


Figure 40: Velocity Distribution in the X- direction at the mid plane for mixing ventilations design’s first case

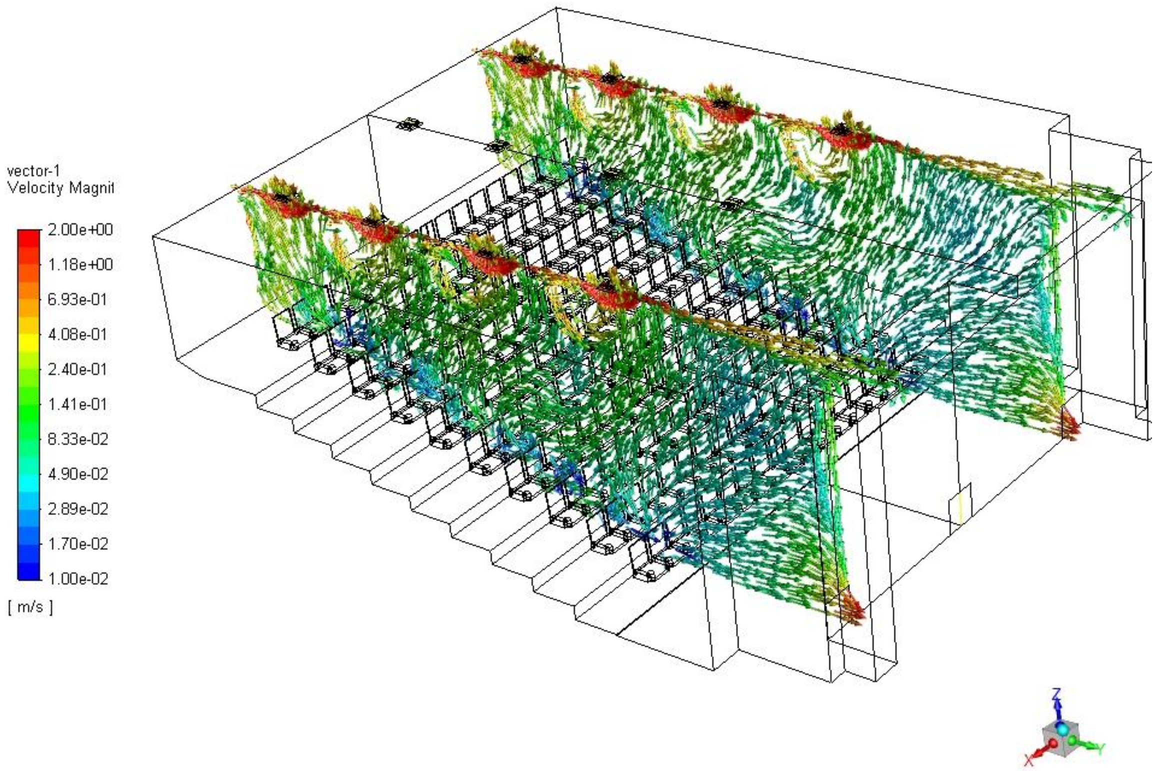


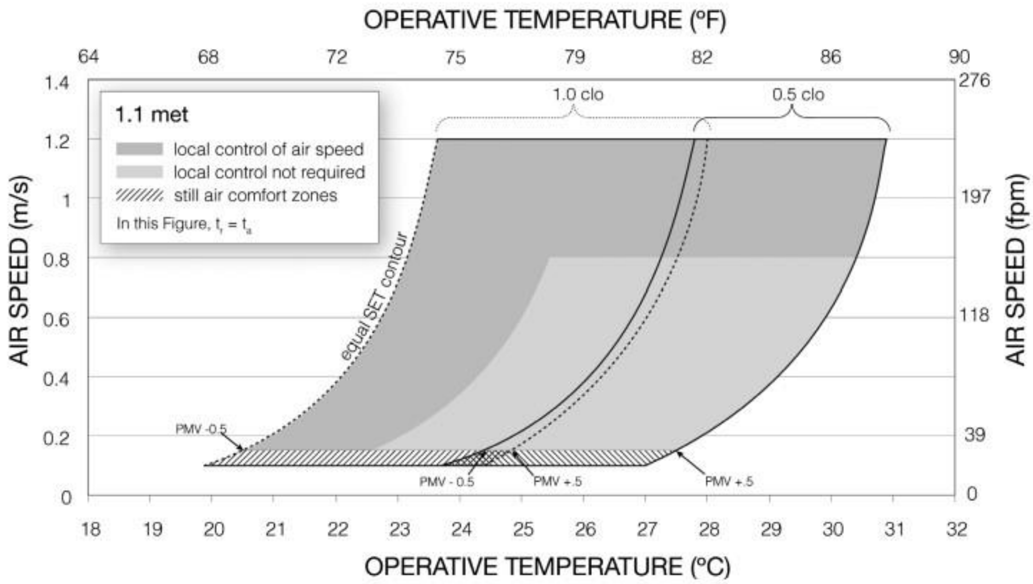
Figure 41: Velocity vector in two planes in X-direction for mixing ventilation design's first case.

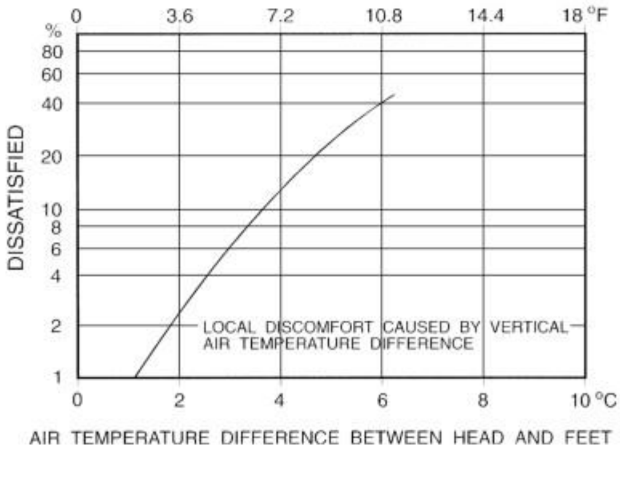
ANNEX 2

These table from ASHRAE 62 and 55 can help us estimate ventilation requirements.

TABLE 6-1 MINIMUM VENTILATION RATES IN BREATHING ZONE
(This table is not valid in isolation; it must be used in conjunction with the accompanying notes.)

Occupancy Category	People Outdoor Air Rate R_p		Area Outdoor Air Rate R_a		Notes	Default Values			Air Class
						Occupant Density (see Note 4)	Combined Outdoor Air Rate (see Note 5)		
	cfm/person	L/s·person	cfm/ft^2	L/s·m^2		#/1000 ft^2 or #/100 m^2	cfm/person	L/s·person	
Correctional Facilities									
Cell	5	2.5	0.12	0.6		25	10	4.9	2
Dayroom	5	2.5	0.06	0.3		30	7	3.5	1
Guard stations	5	2.5	0.06	0.3		15	9	4.5	1
Booking/waiting	7.5	3.8	0.06	0.3		50	9	4.4	2
Educational Facilities									
Daycare (through age 4)	10	5	0.18	0.9		25	17	8.6	2
Daycare sickroom	10	5	0.18	0.9		25	17	8.6	3
Classrooms (ages 5–8)	10	5	0.12	0.6		25	15	7.4	1
Classrooms (age 9 plus)	10	5	0.12	0.6		35	13	6.7	1
Lecture classroom	7.5	3.8	0.06	0.3		65	8	4.3	1
Lecture hall (fixed seats)	7.5	3.8	0.06	0.3		150	8	4.0	1
Art classroom	10	5	0.18	0.9		20	19	9.5	2
Science laboratories	10	5	0.18	0.9		25	17	8.6	2
University/college laboratories	10	5	0.18	0.9		25	17	8.6	2
Wood/metal shop	10	5	0.18	0.9		20	19	9.5	2
Computer lab	10	5	0.12	0.6		25	15	7.4	1
Media center	10	5	0.12	0.6	A	25	15	7.4	1
Music/theater/dance	10	5	0.06	0.3		35	12	5.9	1
Multi-use assembly	7.5	3.8	0.06	0.3		100	8	4.1	1


OPERATIVE TEMPERATURE (ºF)
64
68
72
75
79
82
86
90
1.1 met
local control of air speed
local control not required
still air comfort zones
In this Figure, $t_r = t_a$
1.0 clo
0.5 clo
equal SET contour
PMV -0.5
PMV - 0.5
PMV +.5
PMV +.5
AIR SPEED (m/s)
1.4
1.2
1
0.8
0.6
0.4
0.2
0
AIR SPEED (fpm)
276
197
118
39
0
18
19
20
21
22
23
24
25
26
27
28
29
30
31
32
OPERATIVE TEMPERATURE (ºC)


0
3.6
7.2
10.8
14.4
18 °F
%
80
60
40
20
10
8
6
4
2
1
DISSATISFIED
LOCAL DISCOMFORT CAUSED BY VERTICAL
AIR TEMPERATURE DIFFERENCE
0
2
4
6
8
10 °C
AIR TEMPERATURE DIFFERENCE BETWEEN HEAD AND FEET

ANNEX 3

COOLING LOAD CALCULATION							**Wall Group**	**CLTD**	**CLTD cor**
Project	Cinema Hall						**N**	15	28.44
Site	Pokhara						**S**	12	21.44
Latitude	28.20 N						**E**	19	31.44
Longitude	83.98 E						**W**	23	35.44
							Roof w/fc	50	63.44
							Glass	14	26.44
DESIGN CONDITION			**DB F**	**WB F**	**RH %**	**W**			
OUTDOOR			91.04	84.39	70	218			
	INDOOR		71.6	60	50	50			
					AREA				
CONDUCTION		**DIR**	**COL**	**U**	**GROSS**	**NET**	**CLTD/corrr**	**RSHG**	
(GLASS)		N	Rlf	0.56	0.0	0.0	26.44	0	
Double glazed		S	Rlf	0.56	0.0	0.0	26.44	0	
		E	Rlf	0.56	0.0	0.0	26.44	0	
		W	Rlf	0.56		0.0	26.44	0	
WALL		N		0.243	1000	1000.0	28.44	6911	
Group		S		0.243	1000	1200.0	21.44	6252	
d		E		0.243	800	800.0	31.44	6112	
		W		0.243	800	800.0	35.44	6890	
ROOF				0.134		2000.0	63.44	17002	
FLOOR				0.21		2000			
PARTITION				0.415		0.0	19.44	0	

SOLAR		**DIR**	**Sh**	**SHGF**	**A**	**SC**	**CLF**		
GLASS		N		40	0	0.94	0.86	0	
		S		58	0	0.94	0.71	0	
		E		219	0	0.94	0.31	0	
		W		219	0	0.94	0.35	0	
LIGHT		2000	W *	3.14		1	CLF *	6280	
PEOPLE		200	SHG *	200	N *	1	CLF *	40000	
		135	LHG *	200	N				27000
EQUIPMENT								4500	
INFILTERATION		1.1	*	20	CFM *	19.44	Tc	427.68	
		0.68	*	20	CFM *	168	gr/lb.		0
VENTILATION		1.1	*	1000	CFM *	19.44	Tc	21384	
		0.68	*	1000	CFM *	168	gr/lb.		114240
							TOTAL	**115758**	**141240**
							GRAND TOTAL	**256998**	
							TOTAL TR	**21.42**	
							Duct/pipe heat loss	**2.14**	
							Grand Total [TR]	**23.56**	