ADDIS ABABA UNIVERSITY

ADDIS ABABA INSTITUTE OF TECHNOLOGY

(AAiT)

SCHOOL OF MULTIDISCIPLINARY ENGINEERING

GRADUATE PROGRAM IN RAILWAY ENGINEERING

CFD ANALYSIS OF AIR CONDITIONED AIR DISTRIBUTION IN PASSENGER CAR OF A TRAIN

By

Tadele Bayu

A Thesis submitted to the School of Graduate Studies of Addis Ababa
University in partial fulfillment of the requirements for the Degree of Masters of
Science in Mechanical Engineering
(Railway stream)

Advisor:-Dr.-Ing. Demiss Alemu

SCHOOL OF MULTIDISCIPLINARY ENGINEERING
ADDIS ABABA UNIVERSITY

June, 2016

ACKNOWLEDGMENT

First and above all, I give thanks to God for providing me this opportunity and granting me the capability to proceed successfully.

A special gratitude I give to my advisor, Dr.Ing. Demiss Alemu, whose contribution in stimulating suggestions and encouragement, helped me to finalize this paper.

Furthermore, I am also deeply thankful to my friends and colleagues. Their names cannot be disclosed, but I want to acknowledge and appreciate their help and transparency during my research.

Last but not least I am also thankful to the Ethiopian Railways Corporation (ERC) academic staffs for giving this opportunity.

ABSTRACT

Everyday people spend much time in vehicles. Either riding or driving has become a part of our life. A comfortable thermal sensation brought on occupants contributes a lot to our life and work. On the contrast, a bad and uncomfortable thermal environment may get human ill and even risk their life.

Air distributions in the passenger trains are crucial to thermal comfort and air quality. Computational fluid dynamics (CFD) has been playing an important role in evaluating and designing various air distributions. Recent advances in CFD approaches and turbulence models provide a great potential of improving predict on accuracy of air distributions in enclosed environments.

The objective of the present research was to investigate the air flow distribution inside the passenger train through the route of Addis Ababa –Djibouti as well as to show the terminate velocity (draft) below 0.7 m/s around the passenger's necks by use numerical simulations. For the study purpose, the passenger train coach was simulated by creating the model in CATIA & Ansys design modeller and assessed by CFD analysis using commercial code FLUENT (ANSYS14.5) software for solving, analysing and post processing the results.

Air flow pattern have been studied in the interior of passenger train within different angle of air diffuser arrangement which are at zero, 15 and 25degress in which their results have showed that the passengers 'neck area speed were an average 0.681,0.7146 and 0.7057m/s respectively. Its result obtained from the numerical simulation agreed well with the comfort region which is below 0.7 m/s so that the air diffuser arrangement at an angle zero degree is better than the 15 and 25 degrees.

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

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

Abbreviations

A.A	Addis Ababa
ASHRAE	American Society of Heating, Refrigerating, and Air-Conditioning Engineers
CAD	Computer Aided Design
CATIA	Computer Aided Three Dimensional Interpolate Aided
CFD	Computational Fluid Dynamics
CO_2	Carbon dioxide
CWB	Coincident Wet Bulb Temperature
DBT	Dry Bulb Temperature
DNS	Direct Numerical Simulation
E	East
ERC	Ethiopian Railway Corporation
Fig.	Figure
HVAC	Heating Ventilation and Air- Conditioning
IAQ	Indoor Air Quality
ISO	International Standards Organization
LES	Large-Eddy Simulation
MDR	Mean Daily Range
N	North
RANS	Reynolds-averaged Navier- Stokes
RH	Relative Humidity
RNG	Renormalizing Group
SIMPLE	Semi-Implicit Method for Pressure-Linked Equations
SST	Shear-Stress Transport
3D	Three dimensions

Symbols

A	Area
°C	Temperature in Celsius
dB	Decibel

$\overline{F}$	External body forces
I	Intensity of Turbulence
J_j	Diffusion Flux of Species
k_{eff}	Effective Conductivity
k	Turbulent kinetic energy
K	Temperature in Kelvin
Kw	Kilo watt
L	Length
l/s	Litter per second
m, m^2, m^3	Meter, Square meter, Cubic meter
P	static pressure
Pa	Pascal
$\dot{Q}$	Volume Flow Rate
s	Second
T	Stress Tensor
$\vec{V}$	Velocity Vector
V	Air Speed
W	Width
$\vec{g}$	Gravitational acceleration
ρ	Air Density
ε	Turbulent dissipation rate
$\overline{\tau}$	Stress tensor
ω	Specific dissipation rate
%	Percentage

Subscripts

g	Grills
d	Diffusers
T	Total
P	Total number of occupants
p	Person

APPROVAL

CFD ANALYSIS OF AIR CONDITIONED AIR DISTRIBUTION IN PASSENGER CAR OF A TRAIN

By

Tadele Bayu

A Thesis Submitted in

Partial Fulfillment of the

Requirement for the Degree of

MASTER OF SCIENCE in

Mechanical Engineering

Approved by:

Signature Date

Dr.Ing. Demiss Alemu ____________________ ____________________

Advisor

Dr. Yilma Tadesse ____________________ ____________________

Internal Evaluator

Mr. Yidnekachew Messele ____________________ ____________________

External Evaluator

Dr. Birhanu Beshah ____________________ ____________________
Railway Center Head

Addis Ababa Institute of Technology (AAiT)

School of Mechanical & Industrial Engineering

Railway Engineering

DECLARATION

I hereby declare that the work which is being presented in this thesis entitled "***CFD Analysis of Air Conditioned Air Distribution in Passenger Car of a Train***" is original work of my own, has not been presented for a degree of any other university and all the resource of materials used for this thesis been duly acknowledged.

Tadele Bayu

Date

This is to certify that the above declared made by the candidate is correct to the best of my knowledge.

Demiss Alemu (Dr. Ing.)

Date

CHAPTER ONE

INTRODUCTION

1.1 Backgrounds

Rail travel has been a primary mode of transport for many people since the beginning of the 20th century. Ethiopian Railway Corporation (ERC) have a number plan to give rail transport service though out the country. In the rail industry has a number of standards regarding the thermal environment within the train carriage. International Standards Organization (ISO) details the various temperatures, air velocity and relative humidity ranges which are permitted within the train carriage. These are based on minimal empirical research conducted on trains and may not be suitable in optimizing passenger thermal comfort. [1]

The idea of air conditioning in public transport arises from the need to improve the well being of the people who use it daily to move one place to another place. The term 'Air Conditioning' was coined by Mr. S.W. Cramer in 1906 while he was making efforts in putting the air in a fit condition for the textile industry. The term has since come into use in its broader sense implying control of any or all of the physical or chemical properties of air within any enclosure. It is widely applied for the improvement of standard of living in human life. Air conditioning which deals with the comforts of human beings in an enclosed conditioned space is known as Comfort Air Conditioning. [2]Thermal comfort is a very subjective term since its perception to people has enormous variations in environmental and physiological parameters.

In the past decades there has been much effort to describe and measure thermal comfort. However, human thermal comfort is still far from fully understood and is therefore still under many investigation. There are six primary thermal comfort parameters which are air velocity air temperature, mean radiant temperature, humidity, metabolic rate and clothing insulation. Those parameters mainly influence body’s heat exchange with the environment.

The thermal comfort is a complicated problem because it is related to both psychological and physiological factors, the air-flow and temperature fields are identified as the most important factors. This makes it necessary to investigate the temperature and the air-flow field distributions inside the passenger compartment in the design process or to improve the thermal comfort conditions for the passengers. [3]

The rail passenger vehicle HVAC system is intended to provide a comfortable and pleasant interior thermal environment to the passengers during all expected external ambient environmental conditions. For that purpose, appropriately sized and controlled heating, cooling, and air supply components are integrated into the passenger car design. Based on the thermal load analysis and the specification the HVAC equipment configuration which may consist of single or multiple unitized HVAC units or split system components mounted beneath, within, or on top of the vehicle but in this thesis we use roof mounted HVAC units. In addition to the HVAC units and depending on the application, other HVAC components like ducting, dampers, diffusers, exhaust fans, booster fans, etc., may need to be integrated into the passenger car design.

This research is manly to address the effects of environmental parameters, temperature and air distribution inside the passenger compartment through the route from Addis Ababa to Djibouti port locally. This approach implies the use of powerful computer aided three dimensional interpolate aided (CATIA), computer aided design (CAD) and computational fluid dynamics (CFD) software. These obtained results, in combination with appropriate comfort models, are then used to predict the passenger’s comfort level.

Improving the comfort conditions of rail vehicles is an important factor in increasing the attractiveness of rail transportation system. In particular, providing air conditioning in the rolling stock can play a significant role in making public transport a viable alternative to the private car.

1.2 Air Distribution and Ventilation in Passenger Train

Air motion plays a role in the sensitivity of thermal comfort in the passenger train. In hot weather, as the body tries to cool itself, the flow of air across the body will assist evaporative cooling from sweating. When air has a high relative humidity, the air next to the sweating body may become saturated with moisture, but by moving the air next to the body away and bringing in fresh, lower-humidity air, the evaporation of sweat can continue. Mechanisms of convection can further move the heat generated by metabolic processes from the skin and into the surrounding air. All this leads to continued cooling, and the higher the velocity of air, the more effective is the process.

Air motion significantly affects body heat transfer by convection and evaporation. Air movement results from free (natural) and forced convection as well as from the occupants' body movements. A noticeable air movement across the body when there is sweat on the skin may be regarded as a pleasant cooling breeze. When the surrounding surface and inside train air temperatures are cool, however, it will probably be considered a chilly draft.

Cool air can impact on an occupant in two general cases: Air that is warm when introduced into the room may cool off before reaching the occupant, or the air is intended to cool the occupant under overly warm ambient conditions. In either case, when the temperature of the air impinging on an occupant is below the ambient temperature, the individual becomes more sensitive to air motion and may complain of drafts. Therefore, careful attention must be given to air distribution as well as velocity.

1.3 Overview of Computational Fluid Dynamics

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical analysis and algorithm to solve and analyze problems that involve fluid flows as well as how the gas or liquid affects objects as it flows past.[4] Computational fluid dynamics has been around since the early 20th century and many people are familiar with it as a tool for analyzing air flow around cars and aircraft. Recently simulation studies about ventilation by means of CFD have been developed. Nielsen (1974) was one of the pioneers in applying the CFD to predict the air movement inside the rooms. Since then, the technique has evolved and numerous works indicate the great applicability of the CFD to the simulation of the inner air movement. CFD has also become a useful tool in the data center for analyzing thermal properties and modeling air flow. CFD software requires information about the size, content and layout of the data center.

The fundamental basis of almost all CFD problems are the Navier–Stokes equations, which define many single-phase (gas or liquid, but not both) fluid flows. These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related.

CFD models have been used to study indoor air quality (IAQ) problems, pollutant distributions, and performance of HVAC systems [5] investigated contaminant removal effectiveness of the air distribution systems for a compartment by using CFD modeling. They showed that directional airflow systems could reduce people's exposure to contaminants.

The increasing developments of CFD in recent years have opened the possibilities of low-cost yet effective method for improving HVAC system in design phase, with less experiment required. As CFD has so many advantages, it is already generally used in industry such as aerospace, automotive, biomedicine, chemical processing, heat ventilation air condition, hydraulics, power generation, sports and marine etc.

1.4 Application of CFD for Air Conditioning

A packaged air conditioning unit comprises several components which are compressor, condenser coil, evaporator coil, expansion valve, etc. that may be assembled in several ways. The assembly affects the airflow patterns, which in turn, may affect the performance of each component and the overall performance of the unit. The non-uniform distribution of airflow over the evaporator and/or condenser may cause severe reduction in heat transfer. Therefore, it is important to predict the airflow characteristics within the unit in order to improve its performance. The CFD technique can reproduce the swirling motions and the positions of eddies, quantitatively confirming their dimensions. CFD models can be checked using the measurements (i.e. flow distribution and total flow rates) of existing systems in practical application before the models are used to predict the performance after retrofitting. The accuracy and reliability of simulation results appears to be adequate for the practical applications. Certainly, serious validation of CFD models should be based on more fundamental measurements besides these overall parameters. This might not be practical, as cost and time are of major concern, and might not be necessary in practical applications like the study case, as requirements regarding computational accuracy are not very high.

For this study uses one of the main commercial code of CFD which is Fluent. It is used in the air conditioned of air-flow distribution in passenger train. FLUENT is a cell cantered finite volume, segregated/coupled, implicit/explicit, density based solution technique. In cell cantered schemes the flow variables are stored at the centers of the mesh elements.

Air outlets are usually ceiling-mounted linear slot air diffusers. The main supply duct must be insulated from the ceiling cavity to prevent thermal gain/loss and condensation.

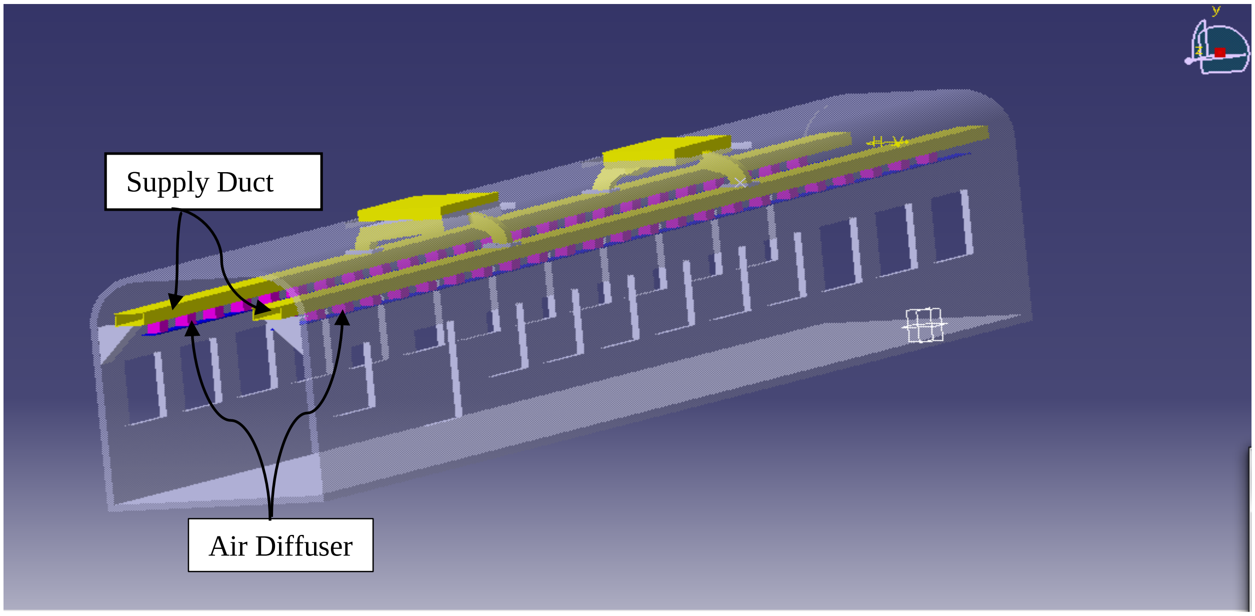


Figure 1.1 Supply Duct and Air Supply Diffuser

1.5 Objectives of the Research

1.5.1 General Objective

The main objective of this study is to analyze the air flow distribution inside the passenger train compartment as well as to show the terminate velocity (draft) below 0.7 m/s around the passenger's necks by using computational fluid dynamics (CFD) or Fluent software.

1.5.2 Specific Objective

The specific objectives of this research are as follows:

- Develop to 3D modeling of full length train body and also the seat bodies with their passengers using CATIA software and also of the air diffusers exhaust griller and train wall using CFD geometry workbench.
- The air diffuser and exhaust griller will be done as follows:
  - ✓ Determine the total numbers and select the type
  - ✓ Calculate the area and flow rate
- Shows the temperature profile in the diffusers, grillers and train wall.

1.6 Statement of the Problem

Air distributions in the passenger train plays a role in the perception of thermal comfort. Slight temperature and air flow imbalance can easily cause respiratory malfunction in humans and also insufficient air motion promotes stuffiness and air stratification. When air motion is too rapid, unpleasant drafts are felt by the passenger train occupants. The neck, upper back, and ankles are most sensitive to drafts, particularly when the entering cool air is more below normal room temperature.

The exact limits to acceptable air motion in the occupied zone are a function of the overall confined space conditions of temperature, humidity, and mean radiant temperature, along with the temperature and humidity conditions of the moving air stream. The need to improve the air flow distribution with the passenger trains is to be an important for their health and it should be given careful attention to air distribution.

1.7 Significance of the Research

Comfort in a rail vehicle is a major importance for enhancing the attractiveness of public transport and make the rail transportation mode more enjoyable. In this study is focused an element of comfort which is air flow distribution, so that the better air distribution in the passenger's compartment generate good feeling by avoiding the fogging phenomenon. By doing this, Ethiopian Railway Corporation can play a significant role in making public transport a valuable alternative long distance journey as compare as the other transport sectors.

1.8 Methodology

The following systematic steps were guiding me in the process of completing this thesis so as to meet each of the objectives.

i. ***Data collection***

- The research data's have been carried out from different books, journals, and previous research works.
- Rail vehicle detailed dimension and vehicle capacity from Ethiopian Railway Cooperation.

ii. ***Modelling Geometry***

The geometry was simplified for the CFD calculations in order to save computational resources.

- Isometric view of the passenger car cabin with eight passengers and also full passengers in the 3D is modelled by using CATIA software. The air inlet, air exhaust and the control volume of a train were done by Ansys Fluent workbench geometry.

iii. Numerical Analysis

- The mathematical model, implemented for the optimization of the air distribution system inside the passenger compartment of a train, was built using CFD. To further reduce the required cell number for the simulation in one case, only one twelve of the compartment was simulated and a sectional symmetry was assumed.
- Mesh structure of the computational domain is very important for getting predicted results in good accuracy and reducing computing time. The volume of the human models and seats were excluded from the meshing process since they were treated as solid bodies.
- The CFD-software used was Fluent, with a standard k-ε turbulence model. Using the boundary conditions which are air-velocity and temperature, occurring in the specific measuring program for the approval in the climatic chamber for railway vehicles. For the unknown boundary conditions suitable assumptions were drawn: Walls and the floor were taken as homogeneous.
- Parametric studies on air supply diffuser and exhaust grillers have been studied for different velocities to assess airflow distribution and thermal comfort of passenger train.

1.9 Limitations and Scopes of the Study

1.9.1 Limitations of the Study

This research is limited due to the following reasons:

- The application of CFD is limited by the preciseness or knowledge of the boundary conditions respectively and by the available hardware performance.
- Fluent is the lack of air diffuser models within the program.
- Only using the route of Addis Ababa to Djibouti,
- There is a direct travel which means no passenger in and out throughout the rout, the door don’t open every stations**.**

1.9.2 Scopes of the Study

The scopes of this study will be covered as follows.

- Analyzing the air flow distribution on the diffusers within the different angles which are at zero, 15 and 25 degrees with respect to the celling and making to get rid of draft in the passengers' compartment.
- Evaluating the temperature distributions only for in the air diffuser, grills and interior wall of the train coach.
- The CFD method follows the use of commercial software ANSYS Fluent 14.5 to solve the problem.
- Using the mixed or ceiling unit ventilation type.

1.10 Outline of the Report

This thesis work is grouped into six chapters. In the first Chapter represents the Overview of Computational Fluid Dynamics, Application of CFD for Air Conditioning, Air Distribution and Ventilation in Passenger Train, objective, the Statement of the Problem, Significance, limitation and scopes of the thesis.

The second chapter 2 is devoted on the extensive literature survey experimental and numerical study of air flow and temperature distribution. In the theoretical survey, more emphasis is given on computational fluid dynamics (CFD) analyses and thermal comfort of it.

The third chapter deals with air distribution in train coach, ventilation selection and type's diffusers strategies.

The fourth chapter a passenger train was modeled; air inlet and outlet were investigated. The model equation includes the equation of continuity, momentum and energy. The CFD computations simulations were done on Ansys fluent software package.

The fifth chapter deals with result and discussion part of the thesis.

Finally on the last chapter conclusion and recommendation together with future work have been addressed in the study.

CHAPTER TWO

LITERATURE REVIEW

While many computational fluid dynamics studies have been conducted to analyse air flow and temperature distribution inside the motor vehicles and rooms area, there were little studies found that investigated the effect air conditioned distributions in passenger train compartment. The literature survey is arranged according to similarity to the work done in this thesis.

In this literature review emphasis is directed on:

- Thermal comfort for passenger train from A.A to Dire Dawa and from Dire Dawa to Djibouti.
- Introduction of Computational Fluid Dynamics.
- Numerical and Experimental study of air flow and temperature distribution.

2.1 Thermal Comfort for Passenger Train from A.A to Dire Dawa and from Dire Dawa to Djibouti

This thesis has been done based on the designed air conditioning system from Addis Ababa to Djibouti. For this literature, it has been taken information from geographical and vehicle data in previous work.

2.1.1. General Comfort Considerations

American Society of Heating, Refrigerating, and Air-Conditioning Engineers (ASHRAE) Standard 55, Thermal Environmental Conditions for Human Occupancy, may be referenced to determine the railcar interior thermal environmental factors that will provide environmental conditions that will be acceptable to a majority of the passengers. [6]

The following factors, typically encountered during rail travel, should also be considered:

- Changes in regional microclimates caused by local geography along the railcar’s route.
- Air infiltration, drafts, and rapid temperature changes associated with door openings.
- Thermal Stratification (Vertical Air Temperature Variation).
- Velocity of air at passenger level (seated and standing passengers).
- Radiation effects of windows and structure.

- Solar loading variation due to dynamically changing solar orientation.
- Clothing insulation.
- Temperature recovery capability and pre-conditioning for passenger occupancy.
- Contribution of HVAC system to overall vehicle noise
- Length of journey.

2.1.2 The Comfort Parameters

Thermal comfort is achieved when passengers perceive the air temperature, humidity, air movement, and heat radiation of their surroundings as ideal and would not prefer warmer or colder air or a different humidity level.

Thermal comfort is influenced by:

- Personal factors (degree of activity, clothing, journey time),
- Spatial factors (radiant temperature, temperature of enclosing surfaces),
- Ventilation factors (air temperature, air speed, relative humidity).

These factors have complex effects on the heat balance of passengers. Thus all contributing factors must be considered in order to achieve conditions which will be perceived as comfortable by a majority of passengers. [7]

Other factors influencing thermal comfort are air quality (dust content; microorganism content; gases and vapours; smells; ion content; electrical and electrostatic fields), noise, lighting, colour scheme, etc. While these factors do not have a direct effect on ambient temperature, they may influence the subjective perception of thermal comfort.

The comfort criteria parameters and design features stated herein are based on practices that have been utilized by the rail transit industry and have resulted in acceptable levels of railcar comfort.

i. *Air Motion*

Controlling air motion within an acceptable range in a passenger railcar is an essential element of passenger comfort. Compared to a residential or commercial building HVAC system, the high occupant density and restricted space of a typical rail passenger vehicle make air distribution very challenging. Higher air velocities are normally desirable during high temperature and humidity operation when maximum cooling is required whereas high velocities constitute uncomfortable drafts when in heating or low level cooling operation.

ii. Temperature Variation

Depending on the type of service conditions that influence the temperature balance in the passenger compartment can change rapidly. Because it is highly impractical to accurately simulate these dynamic conditions in a test environment, evaluation of these criteria are best made under stabilized conditions with all doors closed for the duration of the evaluation measurements.

iii. Temperature and Humidity Design Range

Actual interior temperatures may be expected to exceed the design range when a combination of extreme environmental conditions, passenger overload, or operation occurs beyond the rating conditions. The limits to which interior temperature may exceed the design range are defined in some cases in the equipment section and may have a specific test or analysis requirement for conformance. In any case, these events are considered temporary in nature and may result in a higher percentage of discomfort for some people. The maximum recommended relative humidity (RH) during rated design cooling mode operation is 60%.

iv. Thermal Recovery Rate

The need for a properly designed passenger rail car HVAC system is to maintain stable interior environmental comfort throughout its specified range of external ambient conditions and changing passenger loads is of primary importance. However, consideration must also be given to the ability of heating and cooling equipment to respond quickly to sudden changes in the thermal load which is a common occurrence in a rail passenger vehicle.

v. Spaces Separated from the Main Passenger Compartment

Some vehicles, mostly long distance inter-city type and some used in commuter service, have transient areas that are not intended for prolonged occupancy such as vestibules and toilet rooms and as such have less demanding thermal comfort requirements than the passenger occupied areas. Some are indirectly controlled, relying solely on branch conditioned air ducts from the passenger compartment, while other applications have thermostatically controlled local heaters. Few, if any, have local air conditioning devices or controls for unoccupied areas. The required comfort criteria in these areas can vary widely depending on the level of service and passenger expectations.

vi. Supply Air Temperature

The majority of rail passenger vehicle HVAC systems distribute conditioned air from ceiling mounted grilles or diffusers. Floor and sidewall distribution systems are also used in some applications. Care must be exercised in the layout, design, and adjustment of the air distribution devices to assure uniform air distribution and mixing while avoiding uncomfortable drafts.

2.1.3. Ventilation

a) General consideration for Ventilation

Outside air is typically introduced into a rail passenger vehicle's HVAC system, return air stream by means of a separate ventilation fan or is drawn into the return air stream by the HVAC system's main supply air blower. The outside air passes through separate outside air filters or combined outside/re-circulated air filters prior to entering the HVAC system cooling, dehumidification, and heating apparatus. In addition, depending on the type of service, number of station stops, and car configuration, some unprocessed outside air may also enter the passenger area directly when doors are opened. To mitigate the introduction of unprocessed outside air due to door openings, drafts, and air infiltration due to door seal leakage when the vehicle is in motion, a positive pressurization of 12.46Pa to 37.37Pa is normally recommended (measured at static conditions). Pressurization is achieved naturally by maintaining a well-sealed car body structure and environmental seals (outside doors, windows,), such that sufficient static pressure is created by the supply of outside air and its restricted natural exhaust leakage through small gaps and resilient seals. On particularly well sealed vehicles it may be necessary to add exhaust grilles (active or static as the application demands). Comprehensive research on the air quality on rail passenger vehicles of different types, passenger density, and service schedules is not currently available; therefore, the ventilation rates recommended based on a combination of accepted current best practice for passenger railcar HVAC design and requirements from similar environments contained in ASHRAE Standard 62.1, Ventilation for Acceptable Indoor Air Quality. The suggested ventilation rate recommendations provided here in (Table 1) are expressed in litter per second (l/s) per person at design passenger load. Passenger density on most transit vehicles varies significantly throughout the day from a few passengers during off-peak time to full capacity, including standees, at peak operating periods. The design passenger load, defined in the vehicle technical specification, is the basis for determining the ventilation and cooling capacity sizing of the HVAC equipment. Exceptional passenger loads, above the design capacity, may also be defined in the Technical Specification, but these infrequent short term conditions are generally not considered in the ventilation criteria. Guidance for outside air quantities on other enclosed passenger compartments such as berthing compartments can be found in ASHRAE Standard 62.1.

Table 2.1 Recommended Ventilation Rates lit/s/ Person at Design Conditions

Rail Transit Vehicles	Urban	Commuter	Inter-City
Minimum Recommended ventilation Rate(l/s)	3.54	4.72	5.664

b) Variable Ventilation Control

Recent increased emphasis on reducing energy consumption plus the improvement in design and reliability of variable speed motor designs and variable damper controls have now led to these features being specified in some new vehicle contracts. Some control schemes being considered are indirect means of matching ventilation airflow to passenger load using load sensors or electronic passenger counting systems. Other more direct means such as CO_2 concentration sensors are also under consideration. More research on the effectiveness of variable airflow control schemes is needed before they are widely accepted by the industry.

The use of automatic powered dampers has, however, been adopted on many newer designs based on the ability to expeditiously shut off outside air from the vehicles by the operator if smoke or other hazardous exterior conditions are encountered. Another benefit of controlled powered dampers is to temporarily throttle-off outside air during car start-up to more quickly bring the car temperature to the proper interior temperature after an extended layover period where the HVAC system is shut down.

c) Air Exhaust

The oversupply of air in most rail passenger vehicles resulting from the introduction of outside ventilation air is normally exhausted naturally by the resulting positive pressurization and ex-filtration through the small gaps around structural members and outside door and window seals. Intercity coaches and similar vehicles that have few exterior doors often require static exhaust grilles to relieve excessive pressurization (Excessive pressurization can unnecessarily require higher fan power ratings and energy consumption to maintain the required outside airflow).

In addition, in localized areas where concentrated heat is generated (equipment lockers, food galleys, etc.), or objectionable odours may exist (toilets, food preparation areas, etc.), local forced exhaust fans are recommended to direct this undesirable air away and directly to the outside without adversely affecting the surrounding passenger environment. When forced exhaust is required, care must be taken to ensure that sufficient outside air is introduced to maintain positive pressurization in the passenger section. This may require higher ventilation rates than the minimum recommendation provided in Table 2.1.

Typical exhaust rates for passenger rail car toilet compartments range from 23.6 l/s to 94.4l/s depending on the size of the room. The exhaust of electrical lockers and food preparation apparatus varies significantly depending on the size and design of the equipment. Specific guidance for exhaust design can be found in the ASHRAE Fundamentals handbook.

2.1.4 Geographical and Vehicle Information

In this study, Aisha is the hottest climate area through the route Addis Ababa- Djibouti from Ethiopian Meteorology Service Agency (like location, elevation, latitude, altitude, daily Mean range, and hourly dry bulb and wet bulb) and vehicle dimensions from Ethiopian Railways Corporation Rolling Stocks Specifications.

The geographical information of the selected area called Aisha is given as:

- Elevation: 721m, Latitude: 10045" N, Longitude: 42034"E

And from weather data the external environmental condition is given as shown below:

- DBT = Dry Bulb Temperature = 40°C;
- CWB = Coincident Wet Bulb Temperature = 26.4°C;
- MDR = Mean Daily Range (K) = 16.8 Geographical information

In thermal comfort literature, it has been designed the passenger internal compartment within the recommended thermal comfort zone , the interior cooling temperature at 23.5°C DB and relative humidity of 50%.

According to the highest temperature region, Aisha, the cooling load analysis has been calculated as shown the Table 2.2 [18]

Table 2.2 Cooling Load Analysis Result

Cooling Loads	**Sensible [kw]**	**Latent [kw]**	**Total [kw]**
Conduction (including glazing)	10.31	0	10.31
Passengers & Operator	8.5	5.44	13.94
Outside Air (ventilation)	16.01	23.81	39.82
Passenger-Door Cycling Air	0.825	1.227	2.052
Solar Radiation (glazing)	6.002	0	6.002
Solar Radiation (opaque surfaces)	2.4	0	2.4
Evaporator Fans / Motors (assumed)	2	0	2
Miscellaneous Internal and infiltration Loads(assumed)	2.74	0	2.74
Interior Lighting Heat Gains	0.96	0	0.96
Evaporator Cooling Load	49.745	30.477	80.22
Vehicle Cooling Load (Net Cooling Load) (Gross Cooling Load minus evaporator unit motor and unit internal sensible load)	47.745	30.477	78.22

2.2 Introduction of Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is a branch of fluid dynamics providing a cost-effective means of simulating real flows by the numerical solution of the governing equations. [8] The governing equations for Newtonian fluid dynamics, namely the Navier-Stokes equations, have been known for over 150 years. However, the development of reduced forms of these equations is still an active area of research, in particular, the turbulence closure problem of the Reynolds-averaged Navier-Stokes equations. For non-Newtonian fluid Dynamics, chemically reacting flows and two phase flows, the theoretical development is at less advanced stage.

Computational techniques replace the governing partial differential equations with systems of algebraic equations that are much easier to solve using computers. The steady improvement in computing power, since the 1950's, thus has led to the emergency of CFD. This branch of fluid dynamics complements experimental and theoretical fluid dynamics by providing alternative potential cheaper means of testing fluid flow systems. It also can allow for the testing of conditions which are not possible or extremely difficult to measure experimentally and are not amenable to analytic solutions.

The increasing developments of computational fluids dynamics in recent years have opened the possibilities of low-cost yet effective method for improving HVAC system in design phase, with less experiment required.

2.2.1 Concept of Computational Fluid Dynamics

Computational Fluid Dynamics is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solves, numerical parameters, and grid generations, etc.). The process is as the figure 2.1.

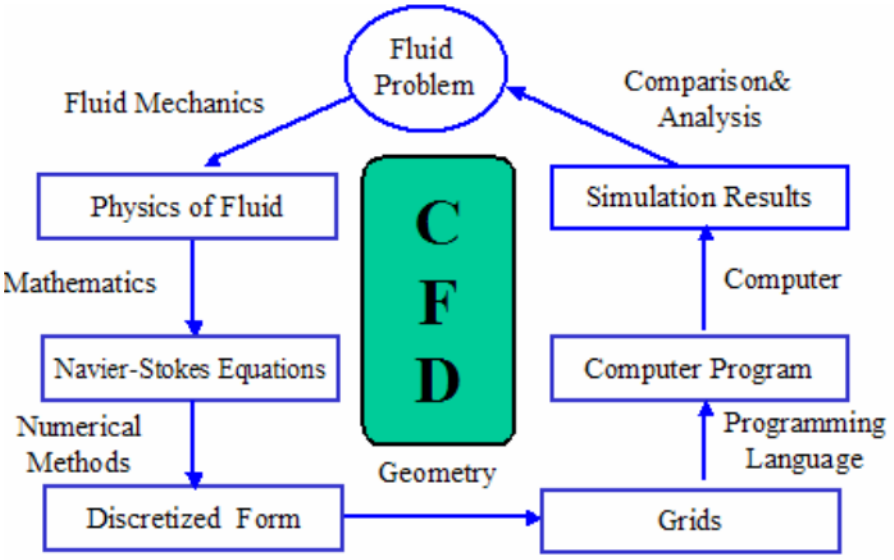


Figure 2.1 Process of Computational Fluid Dynamics [9]

Firstly, we have a fluid problem. To solve this problem, we should know the physical properties of fluid by using Fluid Mechanics. Then we can use mathematical equations to describe these physical properties. This is Navier-Stokes Equation and it is the governing equation of CFD. As the Navier-Stokes Equation is analytical, human can understand it and solve them on a piece of paper. But if we want to solve this equation by computer, we have to translate it to the discretized form. The translators are numerical discretization methods, such as Finite Difference, Finite Element, Finite Volume methods. Consequently, we also need to divide our whole problem domain into many small parts because our discretization is based on them. Then, we can write programs to solve them.

2.2.2 Importance of Computational Fluid Dynamics

There are three methods in the study of fluid: theory analysis, experimental and simulation (CFD) a new method, CFD has advantages compared to experiments.

Table 2.3 Comparison of Simulation and Experiment

	Simulation (CFD)	**Experiment**
Cost	Cheap	Expensive
Time	Short	Long
Scale	Any	Small/Middle
Information	All	Measured point
Repeatable	Yes	Some
Safety	Yes	Some dangerous

CFD gives an insight into flow patterns that are difficult, expensive or impossible to study using traditional (experimental) techniques. [10]

Table 2.4 Comparison of Quantitative Description of Flow Using Simulation and Experiment

Experiments	**Simulations**
Quantitative description of flow phenomena using measurements ▪ for one quantity at a time ▪ at a limited number of points and time instants ▪ for a laboratory-scale model ▪ for a limited range of problems Error sources: measurement errors, flow disturbances by the probes.	Quantitative prediction of flow phenomena using CFD software ▪ with high resolution in space and time ▪ for the actual flow domain ▪ for virtually any problem and realistic operating conditions Error sources: modeling, discretization, iteration, implementation.

The results of a CFD simulation are never 100% reliable because the input data may involve too much Equipment and personnel are difficult to transport CFD software is portable, easy to use and modify

- guessing or imprecision
- the mathematical model of the problem at hand may be inadequate
- the accuracy of the results is limited by the available computing power

The reliability of CFD simulations is greater

- for laminar/slow flows than for turbulent/fast ones
- for single-phase flows than for multi-phase flows
- for chemically inert systems than for reactive flows

CFD uses a computer to solve the mathematical equations for the problem at hand. The main components of a CFD design cycle are as follows:

- the human being (analyst) who states the problem to be solved
- scientific knowledge (models, methods) expressed mathematically
- the computer code (software) which embodies this knowledge and
- provides detailed instructions (algorithms) for

- the computer hardware which performs the actual calculations
- the human being who inspects and interprets the simulation results

CFD is a highly interdisciplinary research area which lies at the interface of physics, applied mathematics, and computer science.

2.2.3 Conservation of Finite Volume Method

If we use finite difference and finite element approach to discretized Navier-Stokes equation, we have to manually control the conservation of mass, momentum and energy. But with finite volume method, we can easily find out that, if the Navier-Stokes equation is satisfied in every control volume, it will automatically be satisfied for the whole domain. In another words, if the conservation is satisfied in every control volume, it will be automatically satisfied in whole domain. That is the reason why finite volume is preferred in computational fluid dynamics.

2.2.4 Grids

The entire face is divided into innumerable small finite number of elements. This process is called meshing and the grid generated is called a mesh. Meshing gives us a scope to study the behaviour of various parameters (such as temperature, velocity etc.) at each of these elements.

There are three types of grids: structured grids, unstructured grids and block structured grids. The simplest one is structured grid (fig. 2.2(a)). This type of grids, all nodes have the same number of elements around it. We can describe and store them easily. But this type of grid is only for the simple domain. This type grid is implemented to this study.

If we have a complex domain, we can use unstructured grid. For example, fig 2.2(b) is an air foil. The structure of air foil is very complex. The flow near the object is very important and complex; we need very fine grid at this region. Far away from the air foil, the flow is comparably simple, so we can use coarse grid. Generally, unstructured grid is suitable for all geometries. It is very popular in CFD. The disadvantage is that because the data structure is irregular, it is more difficult to describe and store them.

Block structure grid (fig.2.2 (c)) is a compromising of structured and unstructured grid. The idea is, firstly, divide the domain into several blocks, and then use different structured grids in different blocks.

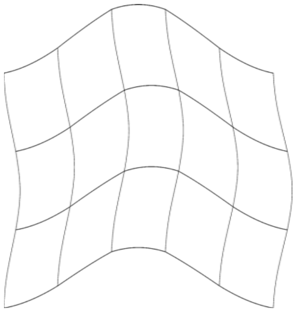


Fig. 2.2(a) Structured

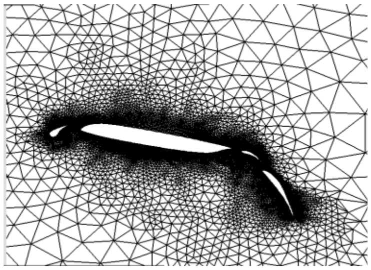


Fig. 2.2(b) Unstructured

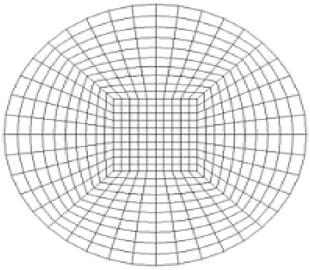


Fig. 2.2(c) Block Structured

2.2.5 Turbulence models

The following turbulence models are applicable to indoor airflow analysis: [15]

a) k-epsilon

b) k-omega

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

Just like the k-ε models, the k- ω model computes the Reynolds stresses by solving two transport equations: one for the turbulent kinetic energy and one for the specific dissipation rate, ω, which can be seen as the ratio between ε and k. The turbulent viscosity is then calculated as a function of k and ω. The major difference in performance between the k-ε models and the k- ω models is found in the fact that the k-ε models are primarily valid for turbulent core flow (i.e., flow in regions relatively far from walls) whereas the k- ω models are created to be applicable through the boundary layer close to the wall. There exists one variant of the standard k-ω model in FLUENT: the SST (Shear-Stress Transport) k-ω model. This model combines the features of the standard k-ε and the standard k-ω model by using the former for the flow somewhat far away from walls and the latter when modelling the flow close to a wall. It uses slightly greater computational resources than k-epsilon but gives a solution of better accuracy as it also includes the boundary layer formation effects in the solution. Reynolds stress model uses seven equations and require 50 % to 60 % greater computational resources than k-omega. It gives a solution of the highest accuracy among the solvers mentioned above. The k-epsilon solver was chosen for the analysis over k-omega as it captures the effects of swirl flows effectively.

FLUENT provides three k-epsilon models:

a) Standard
b) RNG (Renormalizing Group Theory)
c) Realizable

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

2.2.6 Finite Volume Method

CFD software’s normally employing the Finite Volume Method to solve the flow governing equations. In this method, the solution domain is subdivided into discrete control volumes (cells, elements) through a computational grid, where the variable of interest is located at the centroid of the control volume. Then, formal integration of the governing equations of fluid flow over all the control volumes (cells) of the solution domain is carried out. Finite difference type approximations are introduced to replace each integrated equation by a set of linearized algebraic equations written, therefore, in terms of discrete nodal values of the dependent variables. These algebraic equations that represent the balance of fluxes of various flow variables across the finite control-volume faces are then solved by an iterative method to calculate the flow field. This method guarantees global conservation, as well as boundedness and transportiveness, of the fluid properties for the entire domain.

2.2.7 Steps Involved in the Simulation Process

The outline of the simulation process is summarized as follows:

a) Pre-processing

- Modeling the geometry and the flow domain
- Establishing the boundary and initial conditions
- Mesh generation

b) Solving

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

c) Post-processing

- Post-processing the simulation to get results graphs, plots, contour plots etc.

a) Pre-processing

For modelling the geometry, it is a general purpose pre-processor for CFD analysis which provides meshing capabilities wherein the model can be meshed and subsequently imported into FLUENT and solved. Predefined grid topology templates are used to minimize grid setup time and optimize the mesh for the given application.

i. Modelling procedure

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

ii. Mesh generation

The entire face is divided into innumerable small finite number of elements. This process is called meshing and the grid generated is called a mesh. Meshing gives us a scope to study the behaviour of various parameters (such as pressure, velocity etc.) at each of these elements. The finer the mesh (more elements) the better is the scope for analysis since it gives us more number of points to study the behaviour of parameters.

iii. Establishing the boundary conditions

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

b) Solving

Solving is an important phase in CFD analysis. The mesh file is read into FLUENT and a routine grid check is performed to detect the presence of any skewed cells. Skewness is the difference between the shape of the cell and the shape of an equilateral cell of an equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. These skewed cells can be weeded out either by use of smooth/swap grid option in FLUENT. Then, we may proceed to setting up the problem.

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

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

2.3 Experimental and Numerical Study Of Air Flow and Temperature Distribution Different Journals

The following studies were useful in setting up proper CFD models for the purposes of this research. The literature offers many studies that analyze the climate inside motor vehicles and there are several measurement systems to analyze the air and temperature distribution, and to validate the internal temperature and humidity conditions. Some researchers used to evaluate the thermal comfort in vehicles a model for calculation. This method, tested in environments characterized by uniform and constant temperature conditions, is not valid for vehicles within which the parameters that affect the comfort of the users which are radiant temperature, air velocity, etc. are not uniform. Most of the selected journals used similar numerical equations and assumptions even if the vehicles have different types.

Roberto de Lieto Vollaro [11] studied on to analyze indoor climate CFD analysis for urban mobility buses. He suggested that the thermal environment in urban buses also varied greatly: passengers were exposed to local heating and/or cooling due to vertical temperature gradients, radiant asymmetry and local unexpected airflow and the interaction of the cabin thermal environment, created by the HVAC system, the outdoor conditions as well as the occupants was rather complex. He noted that at the moment no standards exist for assessment and classification of the thermal environment quality in vehicles. To obtain some evidences, in order to enhance the indoor climate for a city bus and to improve occupants' comfort monitoring local temperature and air distribution around passengers. He used that the numerical model was implemented with Computational Fluid Dynamic software (CFD, Fluent Inc.): it permits the evaluation of the thermal and fluid-dynamic performances of the air conditioning system and diffusers' distribution. He got that the experimental results were in good agreement with the CFD evidences. This shows that the model developed could give reliable results to optimize and locally modify the air diffusers distribution inside cabin spaces. His evidences could help to improve the air conditioning distribution as a function of the obstructions' typical for a city bus vehicle and to reduce the draught risk related to the bus stop door apertures. One of the most important reasons of local temperature differences and unexpected air velocity gradients was due to the multi-door system apertures at each bus stop.

In a study by Tim Berlitz and Gerd Matschke [12], the authors used interior air flow simulation in railway rolling stock. They introduced that the thermal comfort of passengers in modern trains' thermodynamic parameters like pressure, temperature and relative humidity are important, but also technical features like location, velocity and direction of the air supplied to the compartment must not be neglected. They examined that the interior air flow in a regional train first class compartment was simulated numerically and they used numerical model by full-scale measurements, one gets a powerful and sound tool for parameter variations and the design and construction of air conditioning facilities. By their result that showed a good agreement to the full scale measurements, performed in the climatic chamber.

Seohiro Kikuchi and Kazuhide Ito [14] performed CFD analysis of indoor environmental quality in commuter train. In this paper focused that the indoor environmental quality for the commuter train space was analyzed for assuming the load factor of seat capacity to be a parameter of analysis and especially flow fields, temperature and air quality distributions in vehicle space were analyzed by using CFD technique. They chose the displacement ventilation system was applied in their study.

In their result study, the environments in a commuter train were simulated by CFD, and the controllability of the environments with the introduction of a displacement ventilation system was examined.

In a study by Hao Yang, Yidong Wang, Taibi He [13], the authors used The Analysis on the Effect of Passenger car Air Conditioning and Distribution with Different Inlet Parameters. They suggested that the development of vehicle industry and the improvement of life quality, many factories and scholars are more focused on heating facility and air quality of the Passenger car. Their study was for the Passenger car indoor air distribution, which was analyzing effect pattern of inlet parameters for temperature field and velocity field and has providing a theoretical basis for improvements of indoor thermal environment and heating comfort

Rameshkumar.A, Jayabal.S, and Thirumal.P [3] put together a CFD Analysis of air flow and temperature distribution in an Air conditioned car. He recommended that the human comfort conditions were affected by various Indoor Air Quality parameters in an air conditioned space. They prediction that the methods for an air conditioned system evaluates the indoor environmental conditions of a specific location and selects the most appropriate actions so as to reach the set points and contribute to the indoor environmental quality by minimizing energy costs. Their focuses area was on the numerical study of the temperature field and air flow inside a passenger's cabin with different human load using computational fluid dynamics (CFD) method. Their main goal was to investigate the distribution of temperature and air flow with various human loads inside the passenger compartment in the steady-state conditions. They showed the temperature and air distributions in a passenger's car cabin using CFD analysis and compared with experimental values. They get that predict values were very close to experimental values.

Jalal M. Jalil and Haider Qassim Alwan [16] the authors used entitle CFD Simulation for a Road Vehicle Cabin. In their study focused on a numerical study of a two-dimensional, turbulent, recirculating flow within a passenger car cabin is presented. Their study was based on the solution of the elliptic partial differential equations representing conservation of mass, momentum, temperature, turbulence energy and its dissipation rate in finite volume form. Algebraic expressions for the turbulent viscosity and diffusion coefficients are calculated using the two-equation model (k−ε). They considered different parameters to illustrate their influences on the flow filed and temperature distribution inside car cabin. These parameters include number and location of the air conditioning systems inlets inside car cabin, different air temperatures at the inlets, different air velocities at the inlets, and different solar intensity during day-time for a certain day of the year, different diffuse solar radiation (variation in the kind of car glass).

Generally, they got the results that indicate some of negative effects such as development of zones of low air circulation. Also it is found that the number of inlets inside car cabin play an important role in determining car air conditioning system efficiency. They conclude that the air temperature and velocity at inlets play an important role in determining cabin climate.

CHAPTER THREE

TRAIN COACH AIR DISTRUBUTING

The rail passenger vehicle HVAC system should be designed to provide a comfortable and pleasant interior thermal environment to the passengers during all expected external ambient environmental conditions. For that purpose, appropriately sized and controlled heating, cooling, and air supply components are integrated into the passenger car design. Based on the cooling load analysis on the previous thesis work and the specification the HVAC equipment configuration which may consist of single or multiple unitized HVAC units or split system components mounted beneath, within, or on top of the vehicle but in thesis we use roof mounted HVAC units. In addition to the HVAC units and depending on the application, other HVAC components like floor heaters (strip, radiant, or forced-air type), duct heaters, ducting, dampers, diffusers, exhaust fans, booster fans, etc., may need to be integrated into the passenger car design.

3.1 Vehicle Cooling

To the extent possible, cooling system and air distribution design for the passenger area compartment should minimize drafts and the temperature gradient between air at the diffuser outlet and the return air, while providing enough cooling capacity to compensate for all different cooling loads.

3.2 Vehicle Ventilation and Air Distribution

The HVAC system air ventilation, mixing, treatment, distribution, and exhaust are described in the following sections.

3.2.1 Outside (Fresh) Air

Outside air is required to provide car pressurization and to provide ventilation within the car for all operating conditions.

Outside air should be thoroughly mixed with the return air and then be treated by the HVAC unit before being delivered to the passenger area. Outside air is normally drawn into the system along with the re-circulated air by the HVAC main blower/fan. However, when necessary, a separate booster fan may be required to compensate for the pressure losses in the outside air ducting.

Outside air grille face velocity should be uniform as much as possible and kept to the lowest practical value possible to prevent snow and water ingress as well as pressure drop.

Automatic outside air dampers may be employed for one or any combination of the following purposes:

- To reduce pull-down or pull-up time,
- To close the outside air intakes in the event of an exterior or tunnel smoke condition,
- To reduce the equipment thermal load in extreme temporary conditions, thus preventing it from shut-down, by modulating the outside air volume,
- To adjust the required outside airflow rate based on the number of passengers and/or operation mode, for energy saving,
- To minimize the effects of pressure waves as the result of entering tunnels or passing trains at high speeds.

Dampers that have an ability to control the air volume will require additional inspection and maintenance, as necessary, of the actuating devices and related controls.

3.2.2 Return (Re-Circulated) Air

Return airflow is determined in combination with outside air requirements to achieve the evaporator coil airflow necessary to achieve the design cooling capacity. Return air is usually drawn by the HVAC unit blower through a ceiling mounted grille located below the HVAC unit mixing plenum. Another option is to draw air from smaller grilles or slots distributed in the passenger area. This design has the advantage of reducing the noise and air velocity caused by a single return air opening below the HVAC unit but requires return ducting to be installed in the space-critical ceiling area. To minimize noise and pressure loss, it is recommended that the average air face velocity at the return air grille not exceed 2.04m/s.

Typical return air grille recommended features are the following:

- Rattle free construction,
- Secondary retention hangers to prevent head injury from accidental opening, and,
- Light weight/easily removable design.

3.2.3 Supply Air

The amount of supply air is dictated by the required system capacity (sensible and latent) and the acceptable supply air temperature. The supply (evaporator) airflow rate for this application is calculated using evaporator-coil performance information. From previous research work, a supply airflow rate of a coach has been taken 2001l/s, each of the two HVAC units on this vehicle provides a design supply airflow rate of 1000.5 l/s.

3.3 Exhaust Air

Air exhausted from the passenger car through passive (static air exhaust) or active (forced air ventilation) exhaust systems or a combination of both in sufficient quantity to facilitate the entrance of the outside air without compromising the recommended car interior pressurization criterion. Accordingly, the total exhaust air through the exhaust system should be somewhat lower but in our case air exhausted from the passenger car through passive exhaust system.

3.4 Air Supply Diffusers and Exhaust Grills

3.4.1 Air Supply Diffusers

Air diffusers should be arranged to provide uniform distribution of air throughout the car. Provisions for closing off sections of the air diffusers near return air inlets to prevent supply air from being drawn directly into the return outlet are sometimes required. The diffusers should be designed to provide uniform airflow distribution and mixing of the conditioned air with surrounding air throughout the car. To minimize uncomfortable drafts, air diffuser should be designed to direct outlet air along the ceiling and the sides of the car.

a) Air supply Diffuser Selection

Choice of the linear grille diffuser or involves the assignment of the claim, performance, and data analysis. Diffusers and grilles should be selected and calculated according to the following criteria:

- The type and style
- Function
- Air volume requirement
- Throw requirement
- Pressure requirement
- Sound requirement

The volume of air per diffuser or lattice is what is needed for cooling, heating, ventilation or requirements of the area served by the unit. The amount of airflow required when related to quit, sound, pressure or design limitations, determines the proper diffuser grille or size.

Supply diffusers are selected based on throw length and noise level depending on application using flow rate.

A linear diffuser type LDW/LDB 12/8/2 is selected.

The Function a linear diffuser as follows:-

- The clean air supply is split into a large number of individual jets. Additionally, a part of the supply air is diffused directly along the ceiling through a slot in the diffuser border profile. With this air curtain, dust particles from tobacco smoke to textile abrasions are prevented from depositing on the ceiling which considerably reduces the need for frequent repainting or repair
- High cooling capacity, high air exchange and thorough rising with fresh air, noiseless and without draft phenomena even with low ceiling heights fast reduction of temperature and velocity difference b/n supplied air and room air within the shortest distance from the diffuser.

Linear supply Diffuser features are:-

- High induction effect for rapid reduction of speed and temperature differences, thus high comfort due to low air speeds in the occupied zone.
- Additional ceiling air curtain to reduce pollution on the ceiling in the diffuser area caused by high induction.
- Possibility to adjust diffusers subsequently to optimize the indoor air flow.
- Extremely low-noise with comparatively high flow rate.
- Flexible border options to match many ceiling systems.

Manufacture’s catalogue provide information on the throw, drop, noise level and pressure drop for a specified air field air flow rate and a variety of sizes and types.

Based on the following parameters we can select the suitable linear diffuser:

- The volume flow rate

- supply air temperature
- Room air temperature

The selected linear diffuser Resulting data:

Model: Linear diffuser type LDB 12/8/2

- ❖ pressure drop(pa) =14,
- ❖ sound power level(dB) =28,
- ❖ Extension of jet at which the maximum speed of ambient air was measure (a(Cmax) in m) =2m,
- ❖ Maximum speed of ambient air with uniform distributed thermal load (cm/s)=23,
- ❖ Height of measuring point(h) =1.7m,
- ❖ Height of Room/ cabin(H) =2.8m

The diffusers shall be designed to deliver equalized airflow throughout the car and meet the temperature variation requirements specified.

b) Determination of number of air diffusers per coach

The total number of occupants (P) are 119 i.e. 118 are passengers and one operators. Based on the seating arrangement, one air diffusers serves for two passengers, therefore, the total number of diffusers in the coach are 60. For this study simulation, the total numbers of diffusers are four.

c) Determinations of diffuser flow rate

According to the previous thesis work [17], the total supply flow rate ($\dot{Q}_T$) was taken 2001 l/s.

- To determine flow rate per person, $\dot{Q}_P$

$$\dot{Q}_P = \dot{Q}_T/P$$

$$=2001/119 = 16.8 \text{ l/s/p} = 0.10168 \text{ m}^3\text{/s/p}$$

- To determine flow rate of air diffuser, $\dot{Q}_d$

$$\dot{Q}_d = 2*\dot{Q}_p$$

$$= 2*16.8 = 33.6 \text{ l/s} = 0.0336 \text{ m}^3\text{/s}$$

d) Determination of area of diffuser, A_d

$$\dot{Q}_d = V_d*A_d$$

The inlet speed of the diffuser value based on the consideration of the draft effect on the passenger, V_d =1.2m/s

$A_d = 0.0336/1.2 = 0.028m^2$

Based on the linear diffuser standards,

Length, L_d= up to 2m =0.28m

Width, W_d = (0.031-0.24) = 0.1m

3.4.2 Air Return / Exhaust Grills in Passenger Trains

a) Selection of grillers

Grillers for the package of return air from the passenger compartment to the plenum chamber through the hole. In this case there will be ten pieces of return grill on the ceiling, Transfer grill type-AGS with fixed inverted vee blades with visible fixing screw, boarder counter punched is selected.

b) Determination of total number grills per coach

For this study considered, the four supply air diffusers are exhausted by one air return grillers in passenger compartment. Therefore the total number of exhaust grillers are 15 in a coach.

c) Determination of the flow rate , $\dot{Q}_g$

The volume flow rate diffuser should be equal to the volume flow rate of the return grills. For this study, the volume flow rate of air return grill will be

$$\dot{Q}_g = 4*\dot{Q}_d$$

$$= 4*33.6 = 134.4 \text{ l/s} = 0.1344 \text{ m}^3/\text{s}$$

d) Determination of the speed, V_g

$V_g = \dot{Q}_g/A_g$

Let assume the dimension of grill, $L_g = 0.3$ and $W_g = 0.2$ i.e. $A_g = 0.06 \text{ m}^2$

Therefore,

$V_g = 0.1344/0.06 = 2.2$m/s

3.5 Air Diffusion in Passenger Compartment

The ventilation is the process to provide fresh air to a closed place in order to refresh, to remove and/or to replace the existing atmosphere. The ventilation is used commonly to remove polluting like gases, dust occurring as particles and provide a healthful and healthy atmosphere of work. It can be obtained by natural methods (opening windows or doors) or mechanical methods (fans or blowers). [19]

It is sometimes much more important to ask how the air is distributed into the space than to know the total ventilation flow rate [20]

Just below, different strategies of air diffusion in rooms are described according to ASHRAE [21]. All the strategies have advantages and disadvantages and the final election will depend on each case of application.

3.5.1 Air diffusion strategies

i. Mixing (Ceiling Unit) Ventilation

In mixing ventilation the fresh air, is provided with high momentum to induce the recirculation of all the air and to obtain a sufficient mixture of the polluting agents with the fresh air. The objective is to dilute the contamination level until an acceptable level is obtained all over the room. The air is typically provided to the space through the ceiling with a high momentum to create a good flow of mixture so that gradients of concentration or temperature are not generated in the ideal case. The air jets are the main factors that affect to the movement of the air in the room.

The mixing ventilation has advantages and disadvantages. The ideal mixture of ventilation comparatively uses high air flow of provision, which turns it an inefficient solution as far as power aspects. The high initial momentum from the air diffusers would be sufficient to mix the air of the room. This means that the diffuser has a high loss of pressure, high level of noise and pressure in fans. Therefore each one of the systems consumes more electricity and the electrical energy used in the system becomes function of the total pressure losses of the system. The main advantage is the easy calculation of the system.

The mixing or ceiling unit type of ventilation has been selected for this study.

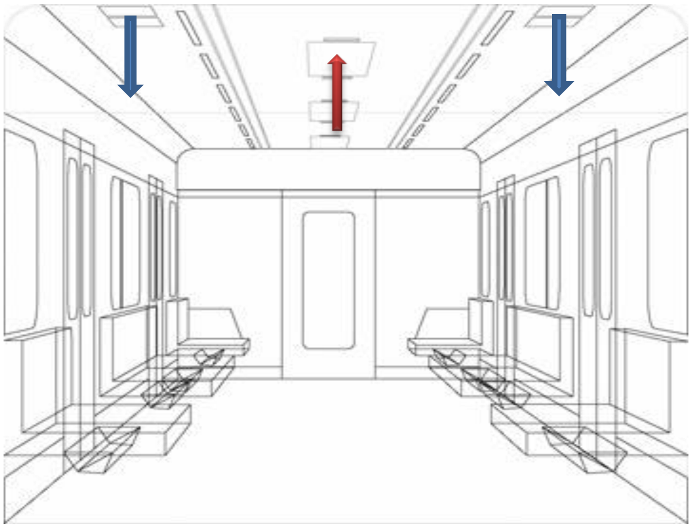


Figure 3.1 Mixing /Ceiling Unit/ Ventilation type

ii. Ventilation by displacement

In the ventilation by displacement the fresh air is provided at floor level with a low momentum and low velocities in the diffusers. Sometimes this configuration is called ventilation by stratification because the field of the flow is almost entirely created by the density differences. The air is moved from the occupied zone by the humans to the superior zone of the room, where the air is extracted by means of extraction diffusers.

The ventilation by displacement is difficult to calculate numerically due to the thermal turbulences of heat sources that generate instabilities in the process of simulation in CFD. The ventilation by displacement can only be used in rather cold conditions. The air flow is characterized by a stable thermal stratification with a vertical linear distribution of temperatures in the room created by the heat sources. The most important advantage of the ventilation by displacement is the use of small air flows compared with the complete mixing ventilation. The ventilation by displacement is significantly influenced by heat sources in the room. The provided air with a low velocity of diffusion and lower temperature than room's air can produce some thermal discomfort if the temperature differences are too great throughout the vertical axis of heights.

Figure 3.2 Displacement Ventilation type

iii. Ventilation by piston

This case is only used in clean rooms. A low turbulence and low air flow velocity is provided through the whole section of the room, moving towards ahead the volume of air towards an extractor that occupies all the whole section. This method is better to remove all the polluting throughout all the entire room. Nevertheless this strategy is inefficient because a great air volume and much energy are used.

Figure 3.3 Piston Ventilation type

iv. Zoning Ventilation

The fresh air is provided with a high force from the ceiling level to the occupied zone. This configuration uses diffusers that would be characterized by a high velocity and decay temperature. The objective of this type of ventilation is to control the air conditions inside the selected zone in the room by means of the provision of air and to allow the stratification of heat and pollutants in other zones of the room. The parameters of the flow in a vertical or horizontal zone can be controlled. In many cases it is desired that the accumulation of heat and polluting agents is made in the superior zone. This type of ventilation is a good solution between the mixing ventilation and displacement ventilation. The efficiency to remove the polluting agents, the extra heat and the relative humidity of the controlled zone is very dependent of the method of air distribution and the internal configuration of the room. In addition, the efficiency of the ventilation with this configuration can be high with a suitable design.

The occupied zone is characterized by a temperature and pollutants constant level. The air flow in the room is partially controlled by provision of air and partially controlled by the buoyancy effect. Normally the zoning strategy is applied to rooms with much height, when the air is impelled at room in the occupied zone and the extraction is in the ceiling of the room.

CHAPTER FOUR

NUMERICAL PROCEDURE

4.1 Physical Model

A three-dimensional model of a train in the shape of a rectangular prism was developed by using CATIA version 5 and Ansys design modeller .The physical dimensions were set to be 3.1m wide by 24m long by 2.8m tall, with seat body arrangement (fig 4.1). The train model in this analysis is used 1/12 of the total length of the passenger coach. The passenger train of the air diffusers, the air return grills and both side ducts is located in the roof, the supply air duct outlet along the duct length direction uniformly distributed in the upper part of the seat rows, return air mouth arranged in the middle of the passenger train.

The model is composed by 60 air supply distributor and 15 exhaust, all of them arrangements were modelled for an overhead air distribution system which are placed in the ceiling of the train. The inlet and outlet are modelled like rectangle. They blow air-conditioned air towards in the passenger compartment. Isometric view of the passenger train with eight human loads is shown in figure 4.2 The construction of all curve surfaces of the human body was assume as flat surfaces.

In this study, 8% of the vehicle is used to analyse the air distribution in the passenger compartment because there is the symmetry in the geometrical shape and boundary conditions.

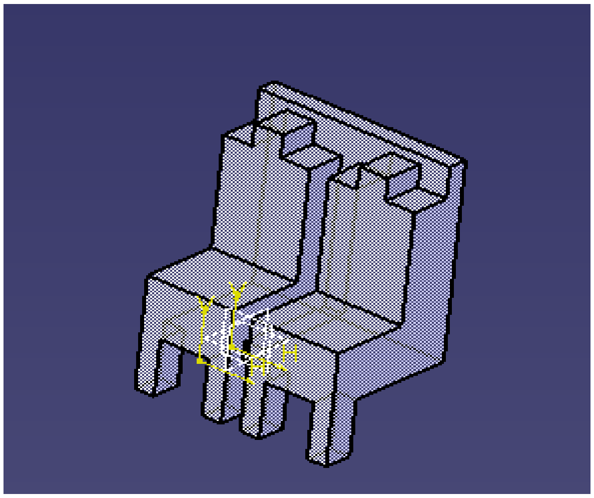


Figure 4.1 The Seat body diagram

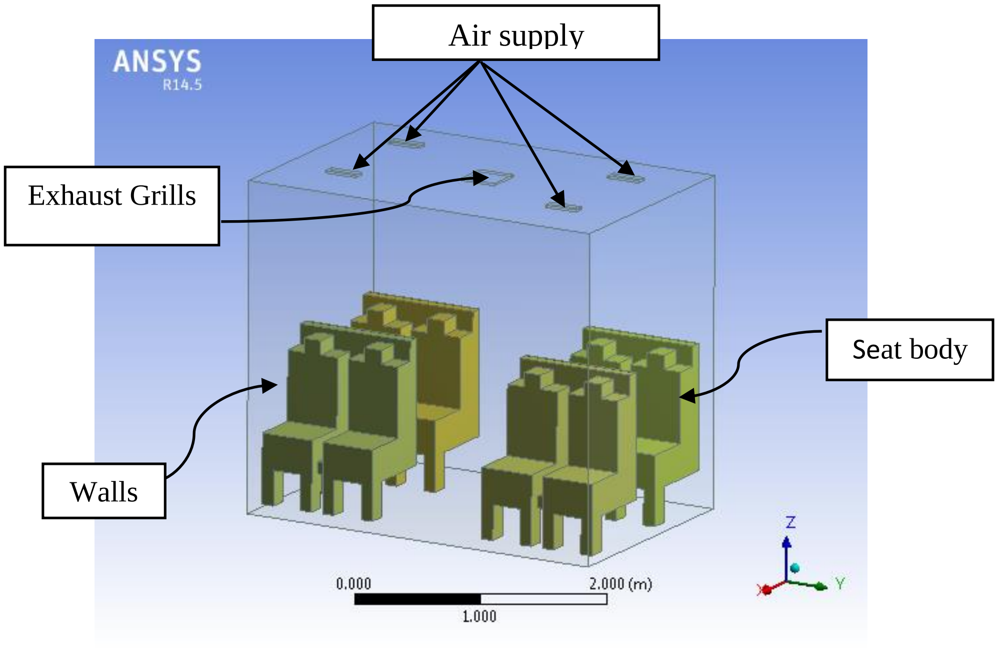


Figure 4.2 The CFD geometrical model of passenger train.

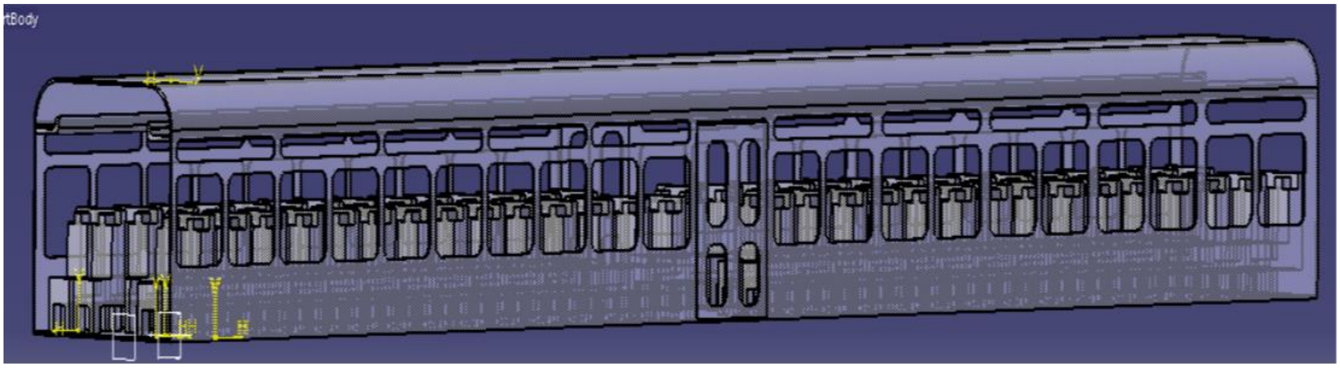


Figure 4.3 The overall geometrical model of passenger coach.

4.2 The Governing Equations

The computer simulation solves numerically the integral equations which describe the mass conservation, momentum or Navier Stokes and energy. The solution of this set of equations provides the distribution of temperature, velocities and pressure fields. This technique is known as computational fluid dynamics CFD. In CFD, the equations are discretized to solve numerically the flow field.

Most simulations are done using steady, incompressible and three-dimensional conditions. Both viscid and inviscid formulations are mentioned. Here a control volume based technique is used to convert the governing equations to algebraic equations that can be solved numerically. This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control volume basis. The following sections describe the governing flow equations.

4.2.1 Mass Conservation Equation

The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{V}\right) = S_{\mathrm{m}} \text{ --- } (4.1)$$

Equation (4.1) is the general form of the mass conservation equation, and is valid for both incompressible compressible flows. The source S_{m} is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources.

Where ρ is the air density and $\vec{V}$is the velocity vector. The first term on the left expresses the time rate of change of density while the second term describes the net mass flow through the control volume.

4.2.2 Momentum Conservation Equations

Conservation of momentum in an inertial reference frame is described by:

$$\frac{\partial}{\partial t}(\rho\, \bar{u}) + \nabla \,.\, (\rho \bar{u}\bar{u}) = -\nabla p + \nabla \cdot (\, \bar{\tau}\,) - \rho \vec{g} - \bar{\mathrm{F}} \text{------------------------- } (4.2)$$

Where p is the static pressure, $\bar{\tau}$ is the stress tensor, and $\rho\vec{g}$ and $\bar{\mathrm{F}}$ are the gravitational body force and external body forces respectively. $\bar{\mathrm{F}}$ can also contain other model-depended source terms. For steady, incompressible and viscid fluid flows with the presence of a jet and without gravitational forces the equation can be written as:

$$\rho\bar{u}\ .\nabla\,(\bar{u}) = -\nabla p + \nabla \cdot (\, \bar{\tau}\,) - \rho\ \bar{\mathrm{F}} \text{---------------------------------------}(4.3)$$

4.2.3 Energy Equations

ANSYS FLUENT solves the general form of the energy equation can be expressed as:

$$\frac{\partial}{\partial t}(\rho E) + \nabla . (\bar{u} (\rho E + p)) = \nabla . (k_{eff} \nabla T - \sum h_j J_j + (\overline{\tau} . \bar{u})) + S_h \text{------ (4.4)}$$

Where k_{eff} is the effective conductivity and J_j is the diffusion flux of species j. The first three terms on the right-hand side of equation 4.4 represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively. S_h includes the heat of chemical reaction, and any other volumetric heat sources that have been defined. For inviscid flows the first and third terms on the right-hand side cancel and the equation reduces to:

$$\frac{\partial}{\partial t}(\rho E) + \nabla . (\bar{u} (\rho E + p)) = \nabla . (\sum h_j J_j) + S_h \text{---------------------------- (4.5)}$$

Because only incompressible flows are considered, the continuity and momentum equation provide a closed system of equations that are fully solvable by numerical algorithms. The necessity of solving the energy equation will in this case disappear.

4.3 Mesh Generation

The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. The finer the mesh or more elements the better is the scope for analysis since it gives us more number of points to study the behaviour of parameters. The attributes associated with mesh quality are node point distribution, smoothness, and skewness.

The mesh file is read into FLUENT and a routine grid check is performed to detect the presence of any skewed cells. Skewness is the difference between the shape of the cell and the shape of an equilateral cell of an equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. Mesh structure of the computational domain is very important for getting predicted results in good accuracy and reducing computing time.

In this study, the volume of the human models and seats were excluded from the meshing process since they were treated as solid bodies. 3-D tetrahedral mesh was used in the present computations. This mesh structure contains triangular elements on the surfaces, tetrahedral elements in the volume region. The surface mesh had a maximum skewness of 0.799 and the volume mesh had 0.92. A picture of the mesh is shown in Figure 4.4 below.

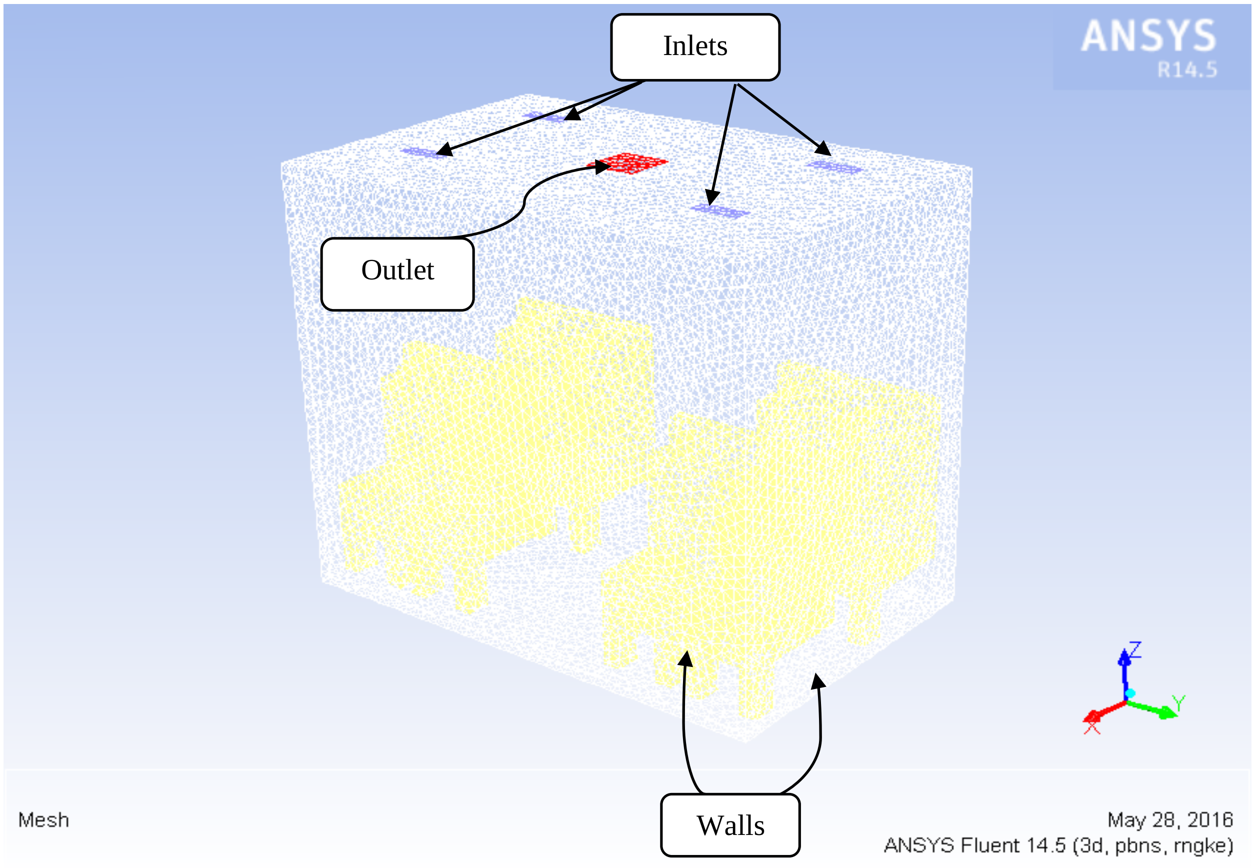


Figure 4.4 Mesh used in CFD simulations

4.4 Boundary Conditions

Boundary conditions define the airflow and the thermal variables in the surroundings of the physical model. For the resolution of the differential equations which have been described in previous sections it is necessary to specify the boundary conditions for the dependent variables in all the border of the calculation extension. The CFD modelling is a very powerful method to solve indoor airflows, but it is very sensitive to its given numerical methods and boundary conditions. The boundary conditions employed in this model can be grouped into three categories:

4.4.1 Inlets Boundary Condition

The air flow comes out through the linear diffusers with a discharge of approximately 34 lit/s through each diffuser unit. The air velocity of inlet vents were specified as 1.2 m/s and the temperature of entrance of the fluid has been considered of 23.5C° (296.5K).

4.4.2 Outlets Boundary Condition

Outlet is set to the pressure - outlet, Zero pressure and zero gradient for all other flow parameters were used as boundary conditions for the exhaust. The air leaving the room was assumed to be at room temperature i.e. 30 C° (303K).

4.4.3 Wall Boundary Condition

According to the physical model of the passenger train, there are internal, external walls. All the types have been described as follows.

Internal walls: The same temperature in both sides of the wall has been supposed, because there is no big temperature difference, so there is not heat flow through the wall. It can be said that the internal walls are considered as an adiabatic surfaces. A no slip boundary condition was assigned for the train wall surfaces, where both velocity components were set to zero at that boundary i.e. $\upsilon_x = \upsilon_r = 0$.

External surfaces: The external surfaces are the external walls and windows. In order to simplify the calculations no heat flow through the walls has been considered. It assumed that the difference of temperatures between the inside and the outside are not too large.

All the surfaces in the indoor space are considered as walls. Close to the wall region laminar viscosity becomes more significant than turbulent viscosity as a result of the damping effect of the wall. Therefore, the turbulence model does not apply to regions close to a solid boundary because turbulence model neglects the laminar viscosity. Fine mesh would be needed near the wall. To avoid this remedy a low Reynolds number model "Wall function" was used.

4.5 Fluid Conditions

A fluid zone is a group of cells for which all active equations are solved. The only required input for a fluid zone is the type of fluid material. The properties of the fluid will be described in the next section Physical properties.

4.6 Physical Properties

An important step in the setup of the model is to define the materials and their physical properties. In this section different properties of the materials will be described briefly to understand the importance of these parameters in the calculations.

4.6.1 Physical properties for solid materials

For solid materials, only density, thermal conductivity, and heat capacity are defined. In the calculations is used the pressure based solver reason why the density and heat capacity are not required. Furthermore it is not necessary to specify the thermal conductivity because as it is written in the boundary conditions sections the heat transmission by conduction is not considered.

In the property list of Fluent the default material with its characteristics has been considered. These values are used only for post processing enthalpy, not in the calculations.

4.6.2 Physical properties for fluid materials

The air of the ventilation system will be the modelled fluid. It is necessary to know the operating conditions of the flow to define correctly the physical properties of the flow.

Therefore it is necessary to define the physical models [24] which will govern the model to analyze. Establishing the input parameters, the fluid inside the train is air with the default properties as defined in the FLUENT library of materials.

4.7 Turbulence models

In order to obtain correct calculation the right choice of the turbulence model is essential. There are a lot of different models for simulating air flow in enclosed environments and it is chiefly their calculating approaches that make them differ from each other. The crudest approach is the DNS, as it does not use any approximations but computes flow any by solving Navier-Stokes equations on all length scales. It solves the behavior of eddies from the smallest dissipative scales to the integral scales at which eddies contain most of the kinetic energy and require a really fine mesh for calculations. Another disadvantage of the DNS is that the need for very short time steps entails a high number of mesh cells and consequently very long calculation times. [25]

The LES approach is based upon Kolmogorov’s theory of self-similarity that suggests that large eddies depend on geometry while the ones on smaller scales are universal. LES separates eddies into small and large ones and calculations both length scales differently. The behavior of large eddies is simulated directly while small eddies calculations use turbulent transport approximation.

The RANS approach requires less computational power than either the LES or the DNS and is therefore the one mostly used on personal computers today. It calculates statistically averaged Navier-Stoke equations to simulate flow with different turbulence model, which can quickly predict air distribution on the basis of mean air parameters.

The choice of the turbulence model mostly depends on the required accuracy of the results and on the affordable computational time. Appointed RANS simulation models were used because of limited computational resources and remarks stated below. They should provide accurate results with reasonable time duration of simulations. The models used were the standard *k*-ε model and RNG *k*- ε model.

The remarks stated below were made in a study evaluating different turbulence models in enclosed environments

- The standard *k*- ε model provides good results for global flow and temperature effects but has not require in predicting high buoyancy effects or large temperature gradients. It does not require many of computational resources.
- The RNG *k*- ε model provides slightly better results in enclosed environments that the standard *k*- ε model and does not need much more computing time.

i. Standard *k*- ε model

The standard *k*- ε model was developed by Launder and Spalding and is today one of the most used models in simulating flows and heat transfer because of its reasonable accuracy for a wide range of turbulent flows. Many of simulations have been made and the standard *k*- ε model has been found to be very good in a wide of situations.

ii. RNG *k*- ε model

The re-normalization group (RNG) *k*- ε model includes a few refinements on the standard model and is therefore reliable for a wide range of flows. It is derived from Navier-Stokes equations using renormalization group methods. The model is thus capable of producing a more accurate prediction of flow in low Reynolds area. There RNG turbulence model also provided good results when used for simulating indoor flows.

iii. SST k-ω model

The k- ω models are the most recent among two-equation eddy-viscosity turbulence models. The symbol ω represents the ratio of ω over k. there are two k- ω models available. The shear stress transport (SST) model was developed by Menter and uses k- ω equations near wall boundaries and a transformed k- ω model far from walls, transitions being controlled by blending functions.

4.7.1 Turbulence modelling with the k-ε model

Air movements in a compartment are usually turbulent and need be modelled in CFD. The fluctuations can be on small scale and high frequency, so that, introducing them in the equations at the moment an extra term called Reynolds stress appears. The introduction of this new term will cause that the equations are not closed and the creation of flows models is required to be able to solve these equations. There are a large variety of methods to solve it, from 0-equations methods until much more complex transport equations of Reynolds stress.

Fluent allow several models to simulate turbulent flows. The election of the turbulence model depends on the considerations that are made, of the physical conditions of the fluid, the capacity of computational calculation and the time available to make the simulation.

Standard k-**ε** has been chosen between the available options. This is a simple model of two equations, where the solution of the transport equations takes independently to determine the turbulent speed and the lengths of scale. The Standard k-**ε** model is robust, economic and presents good approximation for a large rank of turbulent flow, in addition it is a model very used in simulations of heat transference. For these reasons this model has been used in the simulations.

When the flow infers in the domain of the control volume, it is necessary to specify two variables: the intensity of turbulence (I) and the turbulence length scale (μt/μ). From these data and of iterative way the software is able to calculate the values of the turbulent kinetic energy (k) and the ratio of turbulent dissipation (**ε**). The ratio of turbulent dissipation must be of 10^{-4} order to get a good simulation.

4.8 Definition of the Model in Fluent

FLUENT is a state-of-the-art computer program for modelling fluid flow and heat transfer in complex geometries. [26]. FLUENT provides complete mesh flexibility, solving flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular and quadrilateral meshes, 3D tetrahedral, hexahedral and pyramid/wedge meshes and mixed (hybrid) meshes. FLUENT also allows refining or coarsening grids based on the flow solution. Before use can be made of the FLUENT solver grids have to be built with different meshing programs that are supported by FLUENT. [22]

In this study, all flow simulations are completed using ANSYS FLUENT Version 14.5. Fluent allows applying the CFD to solve a large variety of compressible and incompressible flows. The large variety of physical models in Fluent allows predicting, with great exactitude, laminar and turbulent flows, heat transmission, chemical reactions, multiphase flows and other involved phenomena. There are multiple options of solution combined with Multigrid methods to improve the convergence. In Fluent there are multiple ways to solve the dynamic flow problems. Different simulations have been made to know what method was the most appropriate to the model. In the next sections are described the physical models and calculation methods used to solve the airflow distribution and temperature in the passenger train. [23]

4.8.1 Summary of the Numerical Scheme Used

Fluent allows using two numerical methods, the pressure-based solver and the density-based solver. Pressure-based solver is more adapted for low-speed incompressible flows. Therefore this numerical scheme available in Fluent has been used.

In the pressure-based approach the velocity field is obtained from the momentum equations and the pressure field is determined by solving a pressure or pressure correction equation which is obtained by manipulating continuity and momentum equations.

The control-volume-based technique is applied to solve the integral transport equations.
This technique consists of

- Division of the domain into discrete control volumes using a computational mesh.
- Integration of the transport equations on the individual control volumes to construct algebraic equations for the discrete dependent variables such as velocities, pressure, temperature, and conserved scalars.
- Linearization of the discretized equations and solution of the resultant linear equation system to yield updated values of the dependent variables.

The transport equations are solved sequentially. Due to the nonlinearity of the equations, it must be carried out several iterations until a convergent solution is reached. Each iteration consists of the steps that figure 4.5 illustrate and that are described next.

1. The properties of the fluid are updated with the values of the present solution. In the first iteration the values are taken from the initial conditions.
2. The equations are solved at the moment using the present values of pressure and mass flows in the faces of the cells to update the velocity field.
3. Because the velocities obtained in the previous step can locally not satisfy the continuity equation, an equation of Poisson type is used for the correction of the pressure. This equation is solved until obtaining corrections to the velocity field, pressure field and mass flows in the faces that fulfill the continuity equation locally.
4. The equations are solved for scalars, turbulence and energy using the updated values of the variables of the previous step.
5. The convergence of the equations set is checked and the steps are repeated until the convergence criterion is satisfied

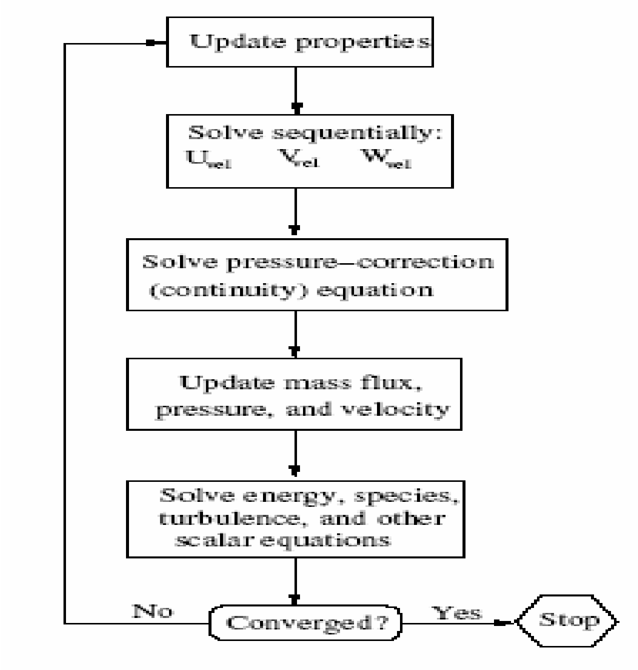


Figure 4.5 Process to solve the equations

The segregated algorithm is memory efficient, since the discretized equations need only be stored in the memory one at a time. Nevertheless, the convergence of the solution is relatively slow since the equations are solved of a decoupled way.

4.8.2 Spatial Discretization

Fluent uses the technique based on the control volume to turn the general transport equation in an algebraic equation that can be solved numerically. This technique of the volume control consists of integrating the transport equation on each control volume, obtaining a discrete equation that express the conservation law on a control-volume basis.

4.8.3 Linearization of Integral Equations

The set of integral equations nonlinear are linearized to produce a system of equations for the dependent variables in each cell. Then the linear system is solved to obtain an updated solution of the flow.

The discretized transport equation contains the unknown scalars variables in the center of the cell as well as the unknown values of the neighbour's cells. This equation is nonlinear with respect to these variables.

Similar equations can be written for each cell in the grid. An implicit form with respect to the dependent variable is obtained as resulting from the linearization. For a determinate variable, the unknown value in each cell is calculated using a relation that includes all the known values and the unknown values of the adjacent cells. Therefore, each unknown variable appears in more than one equation of the system. These equations must be solved simultaneously to determine the unknown values. The result is a system of linear equations with one equation for each cell of the domain.

4.8.4 Resolution of the Linear System of Equations

When the linear system of equations described in the previous section is obtained, this is solved using the implicit method of Gauss-Siedel for linear equations, in conjunction with an algebraic multigrid method. This process of solver is sequential for each one of the considered variables of the flow.

i. Discretization Method

Fluent allows the use of several upwind schemes. The application of one or other method is depend of the model that will be simulated (geometric characteristics, type and structure of the mesh, flow properties, boundary conditions...)

First Order Upwind and Second Order Upwind discretization schemes have been used in the simulations. The First Order Upwind is good when the flow is aligned with the mesh.

This method increases the numerical errors of discretization if the flow is not aligned with the mesh. It has a greater rapidity of convergence but the obtained results are not so good. The results are better in the Second Order Upwind, but this can difficult the convergence.

ii. Pressure-Velocity Coupling

Pressure-velocity coupling is necessary to solve the flow in the face of the cell. Fluent provides different pressure-velocity coupling algorithms. Each one of which works better for the different types of flows.

In this case the SIMPLE algorithm (Semi-Implicit Method for Pressure-Linked Equations) has been used. This method uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field.

4.9 Simulation with Fluent

When the physical models available in Fluent and the numerical methods to solve these have been analyzed, it is time to start with the simulations.

Since it has been said throughout the previous sections, the size of the model influences remarkably in the number of necessary iterations until reaching the wished solution. Furthermore, the transport equations of continuity, momentum, energy and turbulence, must be solved in a sequential way in each iteration, therefore the process of iterative calculation requires time.

In the first section it will be explained the systems used to make the simulations and the consumption of resources and time that has been necessary. Later it will be explained the simulation process followed, from the introduction of models and boundary conditions until the attainment of the final solution and verification of the convergence.

4.10 Simulation Process

To perform the simulation process, we must lay out some rules that affect the physics of the problem. The problem is then set-up based on these under-lying assumptions. The present study is based on the following assumptions:

a) Incompressible fluid flow and finite-volume approach
b) Steady state –summer air conditioning simulation of the room environment
c) Negligible buoyancy effects
d) Negligible air leakage throughout the conditioned space
e) Negligible solar radiation and internal heat sources
f) Semi-Implicit Method for Pressure Linked Equations (SIMPLE pressure- velocity coupling)
g) Negligible radiation and species transport
h) Room boundaries were modelled as adiabatic walls
i) Two solvers are available in FLUENT namely:
    1. Pressure based solver
    2. Density based solver

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

Operating Conditions of the Defines menu, the default value of the operating pressure (101325 Pa) has been used. The option of gravity has been activated in a value of –9.81 m/s in the Z axis. The gravity is necessary so that the effects of buoyancy due to density air differences are considered.

Next, the boundary conditions have been defined according to it was explained in section 4.4. All the inlets, outlets and walls have been defined. Also the materials used in the simulation have been defined.

The next step has been to define the solver of Fluent. The Pressure-based solver has been used since it has been justified in section 4.8.4(ii). Implicit formulation has been used and the equations are solved in a segregated way, that is to say, one behind the other. The calculation has been made for the steady case and with an absolute formulation for the velocity.

Sometimes to begin the iterations considering only the mass and momentum conservation equations is recommendable, and to select the options of energy and turbulent models when a stable solution has been reached. In this case the problem is that the movement of the flow is very dependent of the natural convection, reason why better results were reached if the energy equation and turbulence model were considered from the first iterations.

The turbulent model used has been the RNG k- ε, the option swirl dominated flow, full buoyancy effects and the standard wall function is used. This model is explained in section 4.7.

An important step to reach the good results is to define correctly the *Solution Controls* in the menu *Solve*. To couple the pressure and velocity and ensure continuity is satisfied, the SIMPLE algorithm was employed to obtain a numerical solution of the momentum equations. The SIMPLE algorithm in Fluent uses a procedure similar to the one outlined. Using this method, the face value of velocity is not averaged linearly. Momentum-weighted averaging, which uses weighting factors, is performed. *Standard discretization* has been selected for the discretization of the pressure, and the *First Order Upwind Scheme* has been selected for the discretization of the momentum, viscosity and energy equations. This type of discretization facilitates the convergence of the solution but the results are not very good. Therefore, the *second order* momentum, turbulent kinetic energy, turbulent dissipation, and energy discretization were utilized to discretize the governing equations.

The last step is to initialize the solution. The initialization consist on provide Fluent with an initial guess for the solution flow field. The complexity of the model has taken to initialize the solution for all zones.

4.11 Setting the Solution Controls and Obtaining a Converged Solution

A summary of the options set in Fluent is included in tables 4-1 through 4-2. The rest of the options were all set to default. With these settings and boundary conditions, the solution was initialized and iterated till convergence. Iteration consists of the following steps:

a) Fluid properties are updated, based on the current solution. And if the calculation has just begun, the fluid properties will be updated based on the initialized solution
b) Three momentum equations are solved in turn using current value of the pressure and face mass fluxes, in order to update the velocity field
c) The velocity obtained in first step may not satisfy the continuity equation locally. A Poisson-type equation for the pressure correction is derived from continuity equation and the linearized momentum equation. This pressure correction equation is then solved to obtain the necessary corrections to the pressure and velocity fields and the face mass fluxes such that continuity is satisfied
d) When interface coupling is to be included, the source terms in the appropriate continuous phase equations may be updated with a discrete phase trajectory calculation
e) A check for convergence of the equation set is made

Above steps re-occur until convergence criterion is achieved.

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

Table 4.1 Solver settings

Feature	**Status**
Space	3D
Formulation	Implicit
Time	Steady
Energy equation	Enable
Viscous	Standard k-ε model
Near-wall treatment	Standard wall functions
Viscous heating	disabled

Table 4.2 Discretization scheme

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

CHAPTER FIVE

RESULT AND DISSCUSION

In this section, the computational results for the simplified passenger compartment are discussed and the results of the simulation are going to be analyzed to get an idea of the air flows and temperatures distribution obtained with this model. The RNG k-ε turbulence model took for this study because of the formation swirl motion in the vehicle compartment. This paper adopts the air supply velocity 1.2m/s, the supply air temperature is 296.5K and within different angles air diffuser arrangement which are at zero, 15 and 25 degrees. The simulation results of air flow and temperature distributions for the eight passengers are shown figures below.

5.1 Velocity field of diffuser at angle of zero degree.

General observation of the flow behavior inside the compartment has been shown below which is characterized by the flow discharged from the air diffusers at angle of 0° with respect to celling in which air diffusers are mounted on the celling. As we can see from the figures 5.1 through 5.6, the smallest velocity of zero m/s is displayed blue, the highest velocity of 2.086 m/s is colored red.

In the analysis of results section the results of the simulation are going to be analyzed to get an idea of the air flows in the passenger train model. The velocities obtained in the model are directly affected by the decision taken on how to model the inlets.

Figure 5.1 illustrates the velocity distribution in the passenger's train model. It shows the velocity field magnitude for inlet speed of 1.2m/s, air exhaust speed with a range from 0m/s to 2.086m/s.

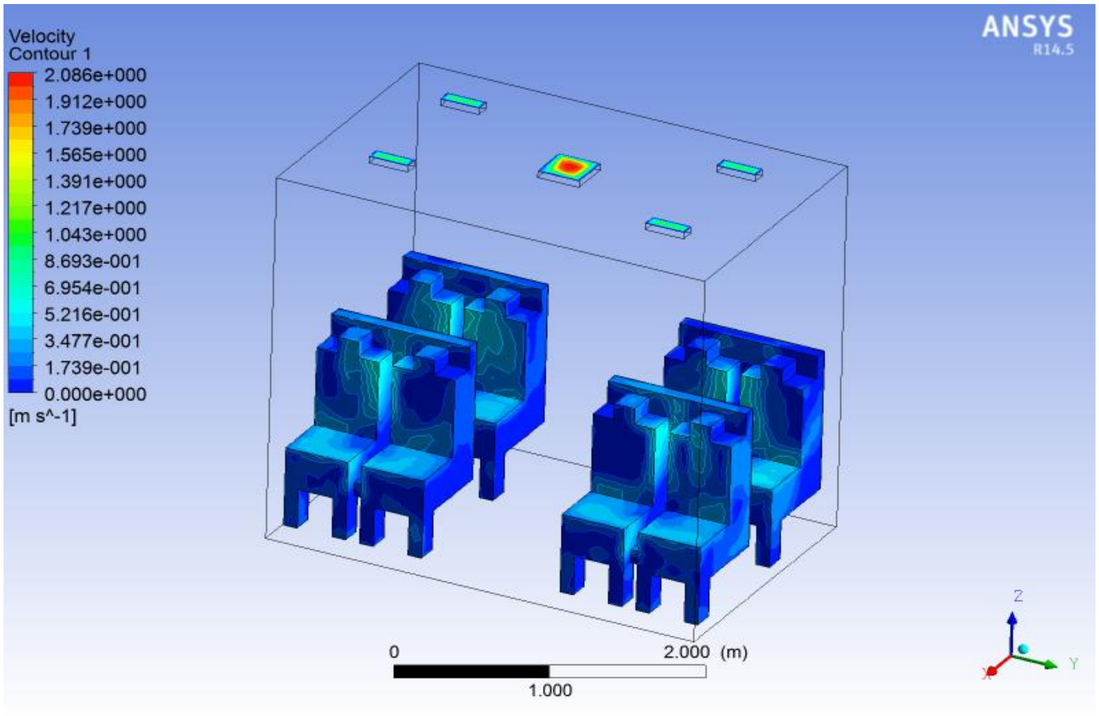


Figure 5.1 Velocity contours of air diffusers exhaust and seat body at zero degree.

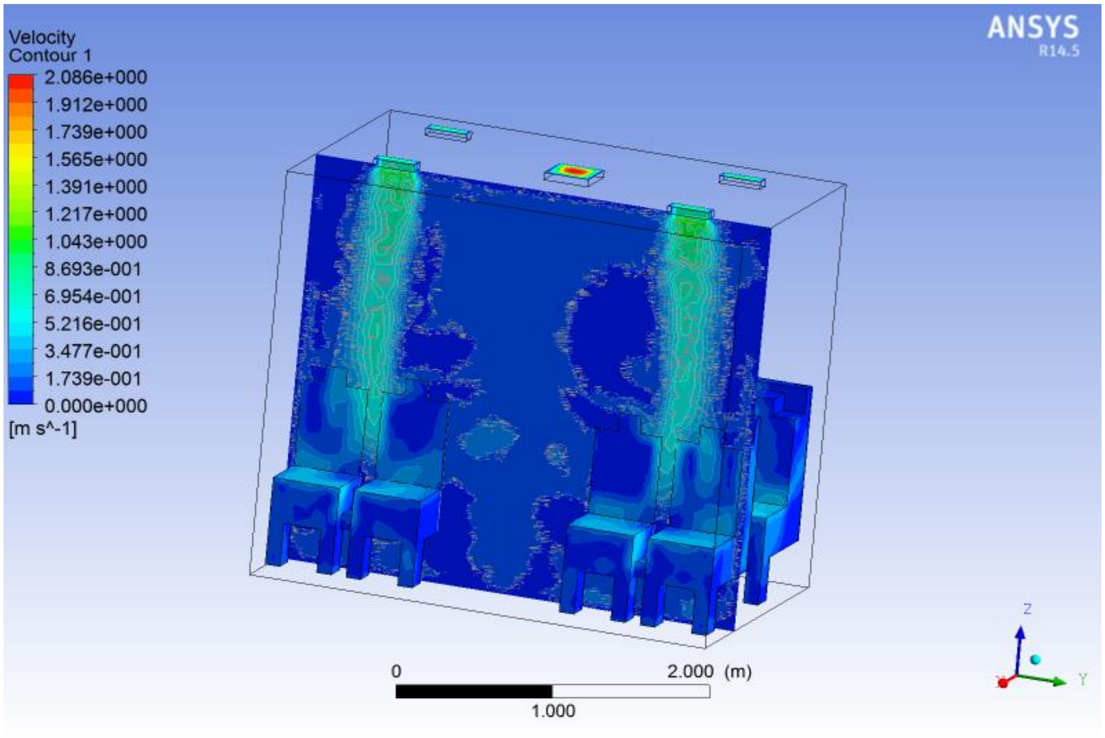


Figure 5.2 (a) velocity contours in Z-Y plane inlet speed 1.2m/s at zero degree.

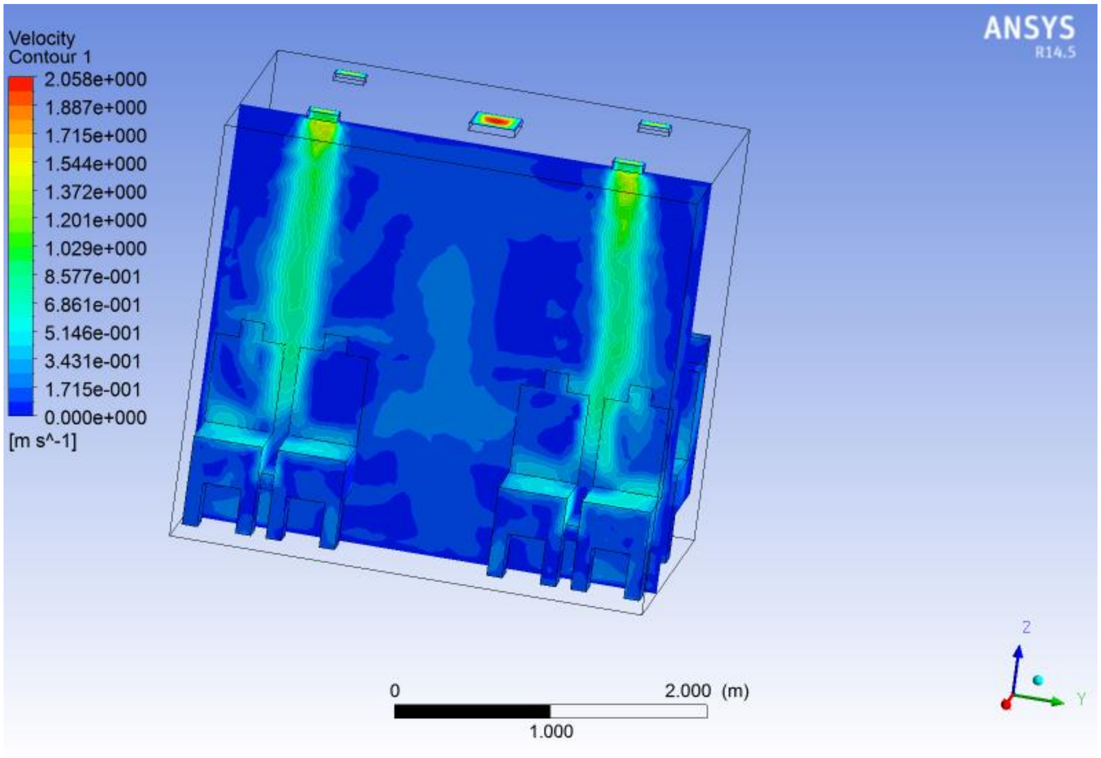


Figure 5.2 (b) velocity contours in Z-Y plane inlet speed 1.5m/s at zero degree.

i. Effect of Inlet Velocity

Different inlet velocities have been used to study the impact of inlet velocity flow fields. Fig.5.2 (a), and (b) indicates the velocity distribution at vertical plane z-y at the centre seat body of the compartment. The velocity distribution is nearly uniform entire the model train except below the passenger legs, but comfort zone (below 0.7 m/s) varies for two cases. Based on the draft cases, the air speed of on the passenger’s neck should be lower than 0.7 m/s. when we look the comfort zone in the passenger, the value speeds were in fig. 5.2(a), (b) below 0.6954m/s and 0.8577m/s respectively. Therefore, the inlet speed should be 1.2 m/s in the diffuser outlet to create a better comfort.

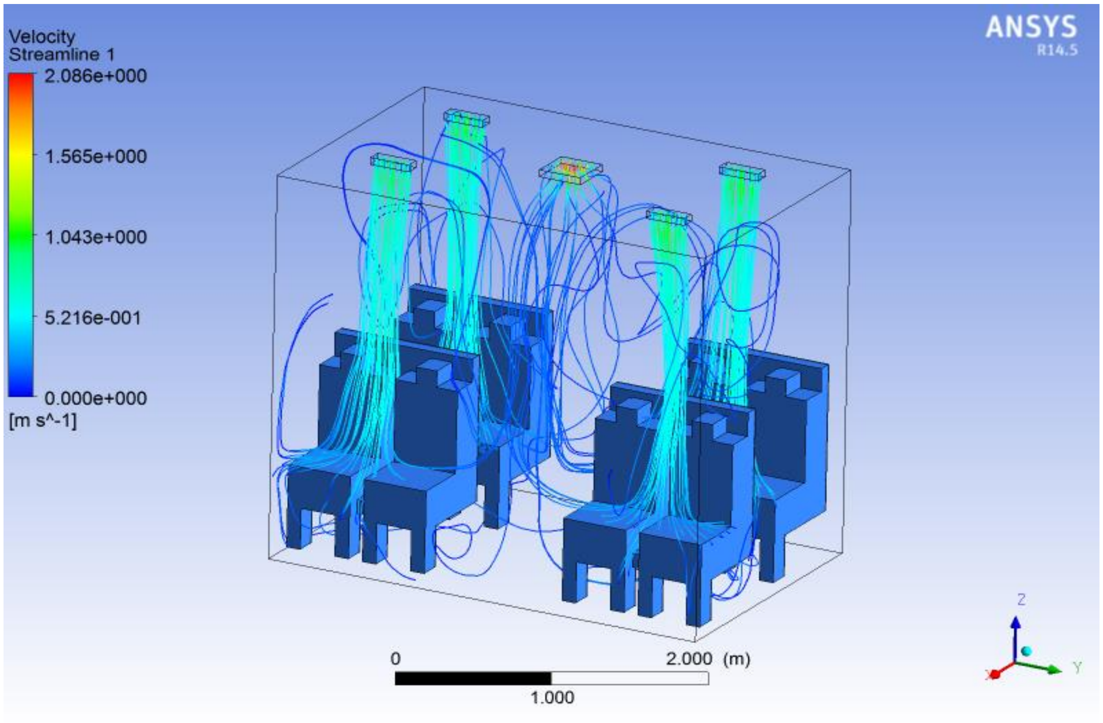


Figure 5.3 (a) velocity streamlines without the effect of human load at zero degree.

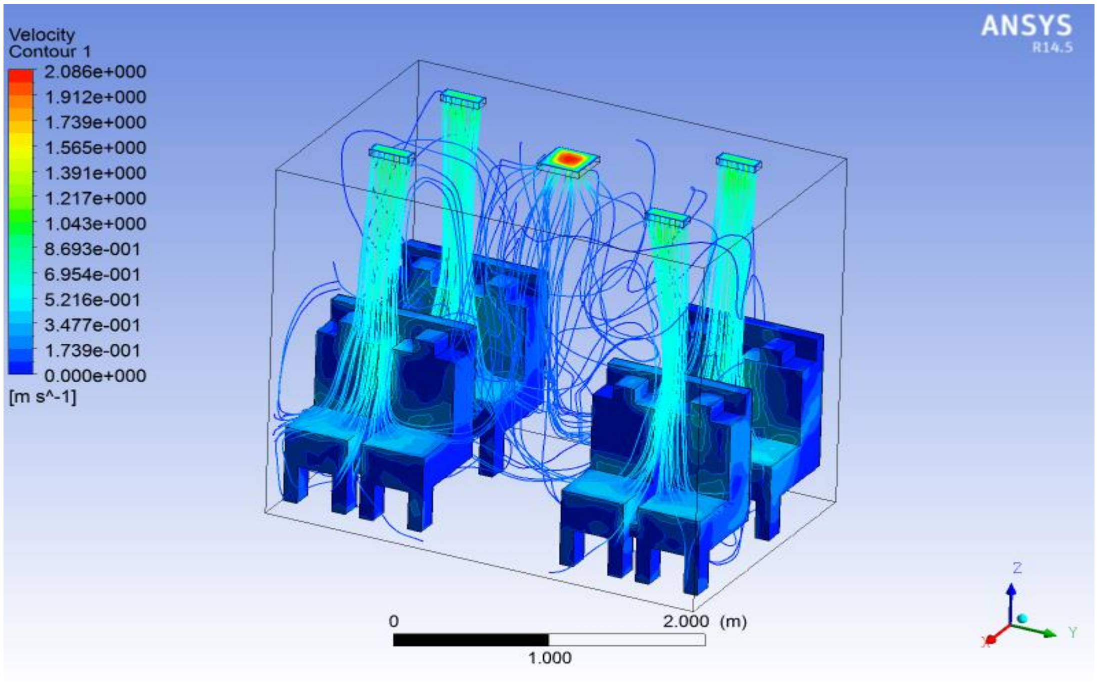


Figure 5.3 (b) Velocity streamlines at zero degree.

Figure 5.3 (a) and (b) illustrate velocity streamlines in passenger train model with the variation maximum number of points 25, 150 respectively. It also shows the effects the air flow in the passengers. As we can see the seated body, the magnitude of the velocity varied from 0 m/s to 0.5216 m/s.

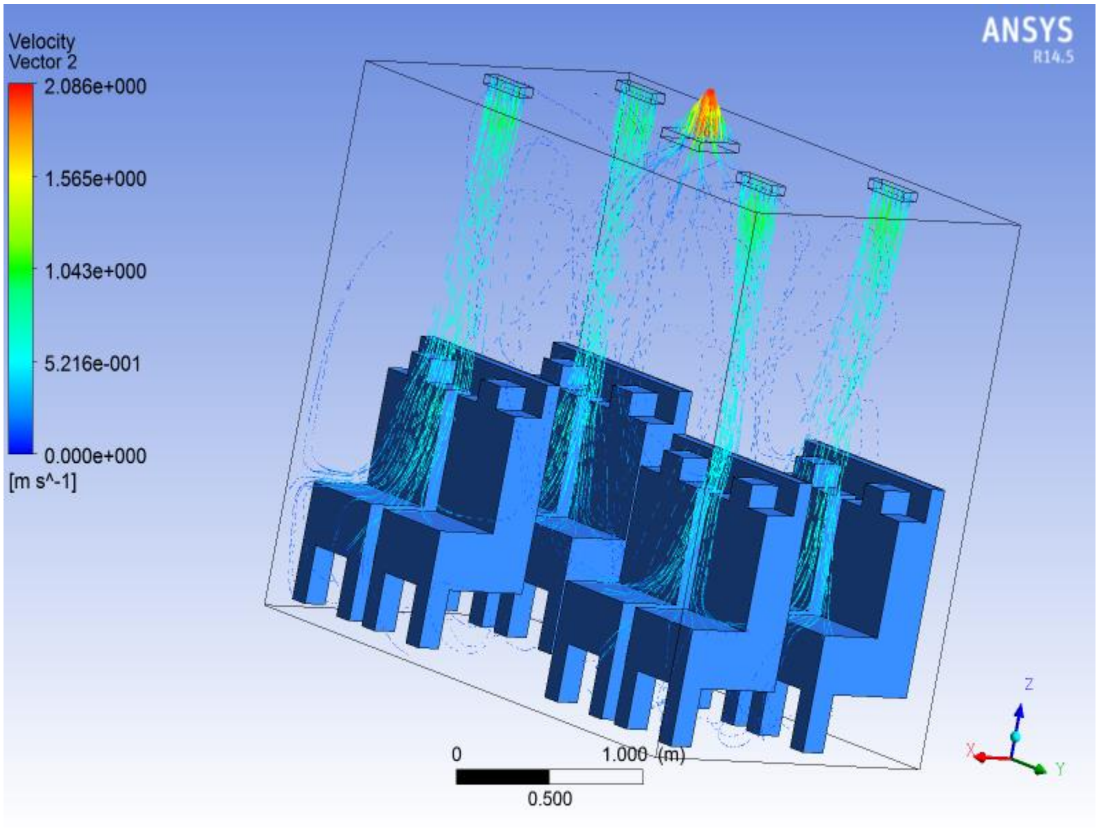


Figure 5.4 (a) Velocity vector at zero degree.

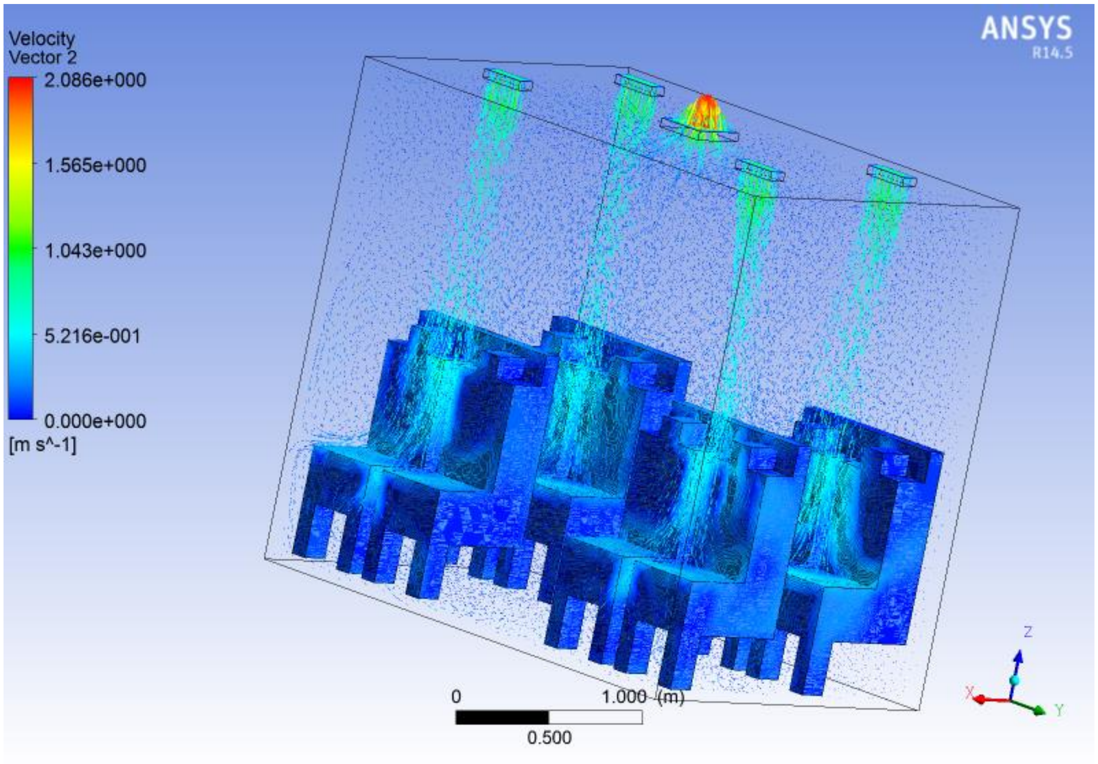


Figure 5.4 (b) Velocity vectors with the effect of seat body at zero degree.

Figure 5.4 (a) and (b) shows the velocity vectors with different maximum number of points in the passengers' train model. The velocity field results in the compartment are the same as the velocity streamlines and also velocity volume rendering.

ii. Description of Airflow Emerged from the inlet

The air flow emerged from the opening enters the room in one direction and suddenly expands to fill the whole passenger train model as shown in Fig. 5.5. The rectangle of the airflow vanishes when its front reaches to the person's shoulders' therefore; presence of human increases the turbulence and disturbance in the flow distribution.

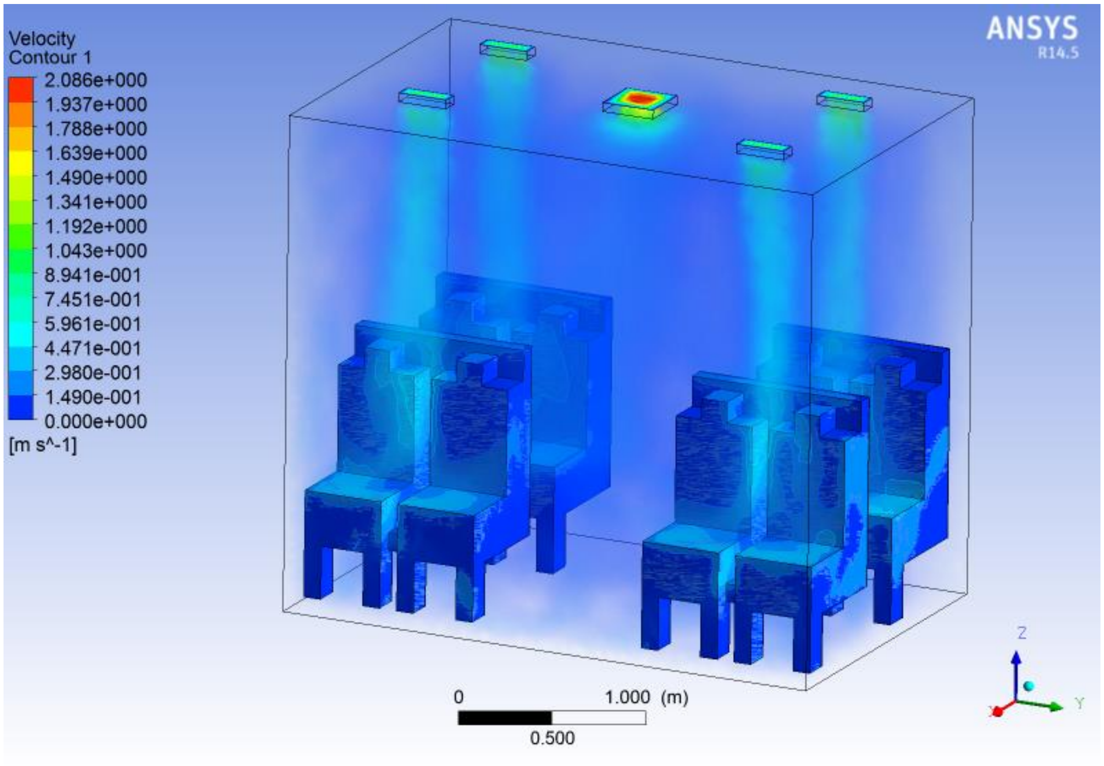


Figure 5.5 velocity volume rendering at zero degree.

A lack of flow has been observed near the corners and the lower part of the compartment. Consequently, a critical dead zone occurs, most continuously at the corners and at the bottom most part of the passenger seats.

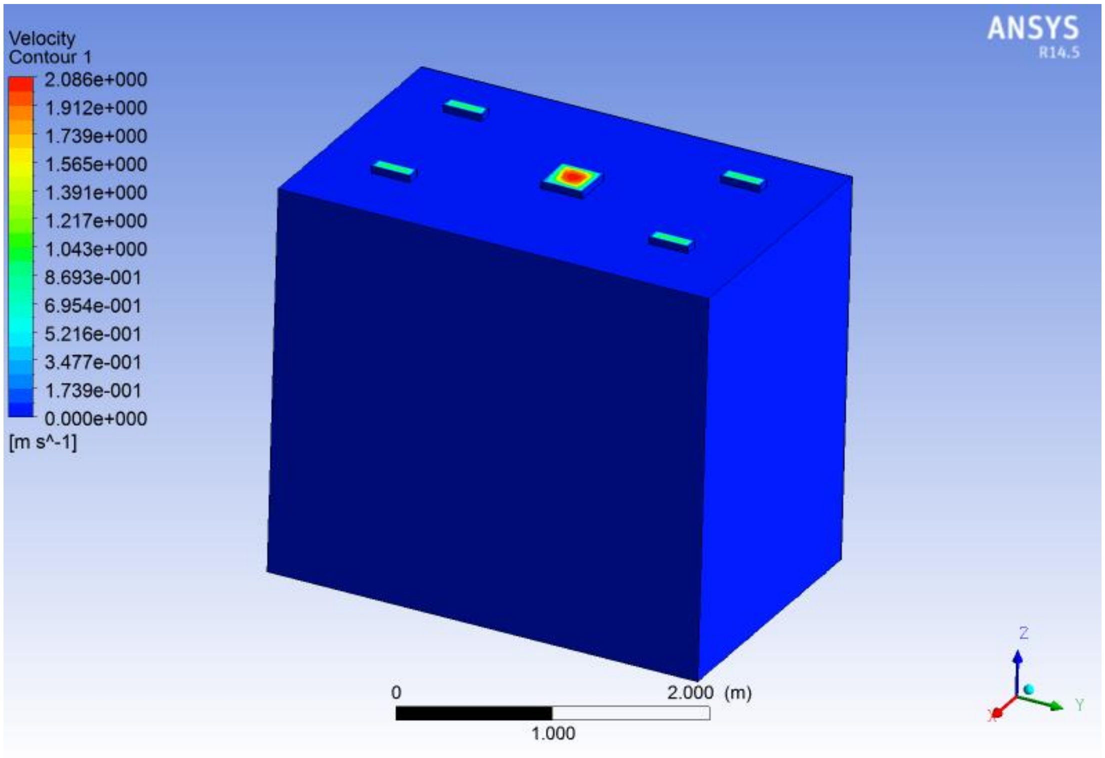


Figure 5.6 velocity contours on the wall at zero degree.

As we can see the figure 5.6 shows that the inlet diffuser angles do not affect using the different angles on train wall velocity. The train wall velocity contour has been observed zero m/s because the wall boundary condition close to the wall region laminar viscosity becomes more significant than turbulent viscosity as a result of the damping effect of the wall. Therefore, the turbulence model equations' do not apply to regions close to a solid boundary because turbulence model neglects the laminar viscosity.

5.2 Temperature field of diffuser at angle of zero degree.

The temperature distrubution in the vehicle are not much signifcant difference between the passengers. In the figure 5.7 shows that the interior wall temperature distrubutions are 300 K and 298.9 K respectively. And also in the figure 5.8 the maximum and minimum temperature in the passenger seated body are 297.5K and 297.2K respectively.

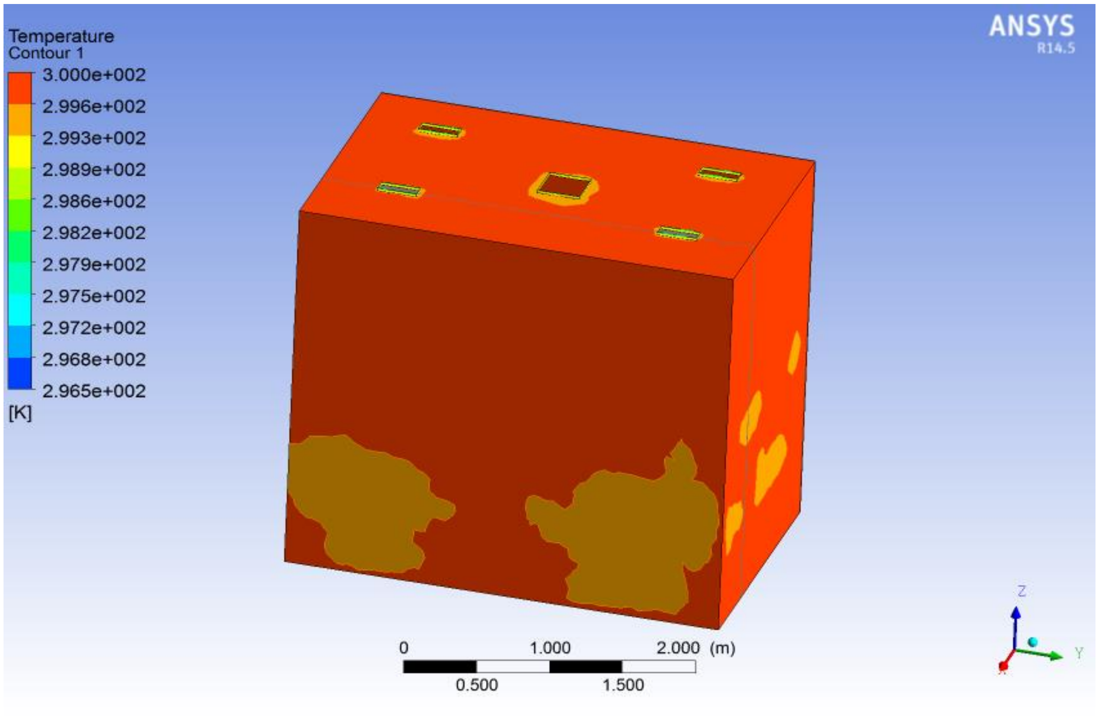


Figure 5.7 Temperature contour of wall at zero degree.

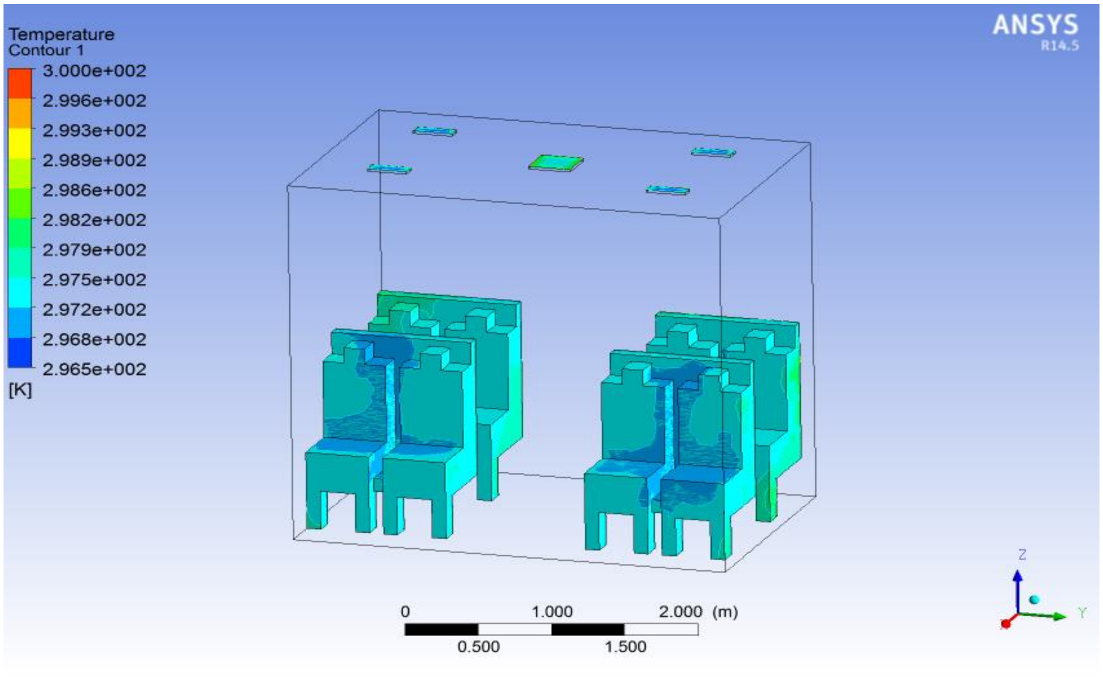


Figure 5.8 Temperature contours of diffusers exhaust and seat body at zero degree.

5.3 Velocity field of diffuser at angle of 15 degree.

The flow behavior inside the compartment has been shown below which is characterized by the flow discharged from the air diffusers at angle of 15° with respect to celling. As we can see from the figures 5.9 through 5.12, the smallest velocity of 0 m/s is displayed blue, the highest velocity of 2.80 m/s is colored red.

Figure 5.9 illustrates the velocity distribution in the passenger train model. It shows the velocity field magnitude for inlet speed of 1.2m/s, air exhaust speed with a range from 0m/s to 2.80m/s.

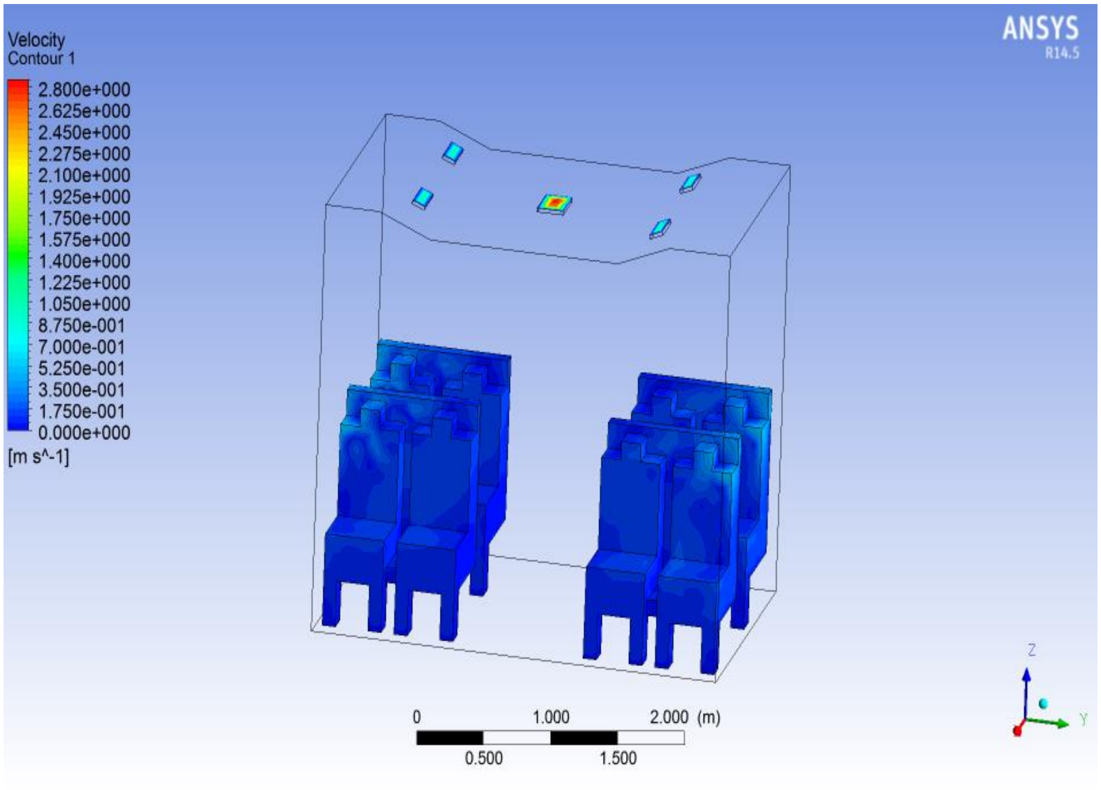


Figure 5.9 Velocity contours of air diffusers exhaust and seat body at 15 degree.

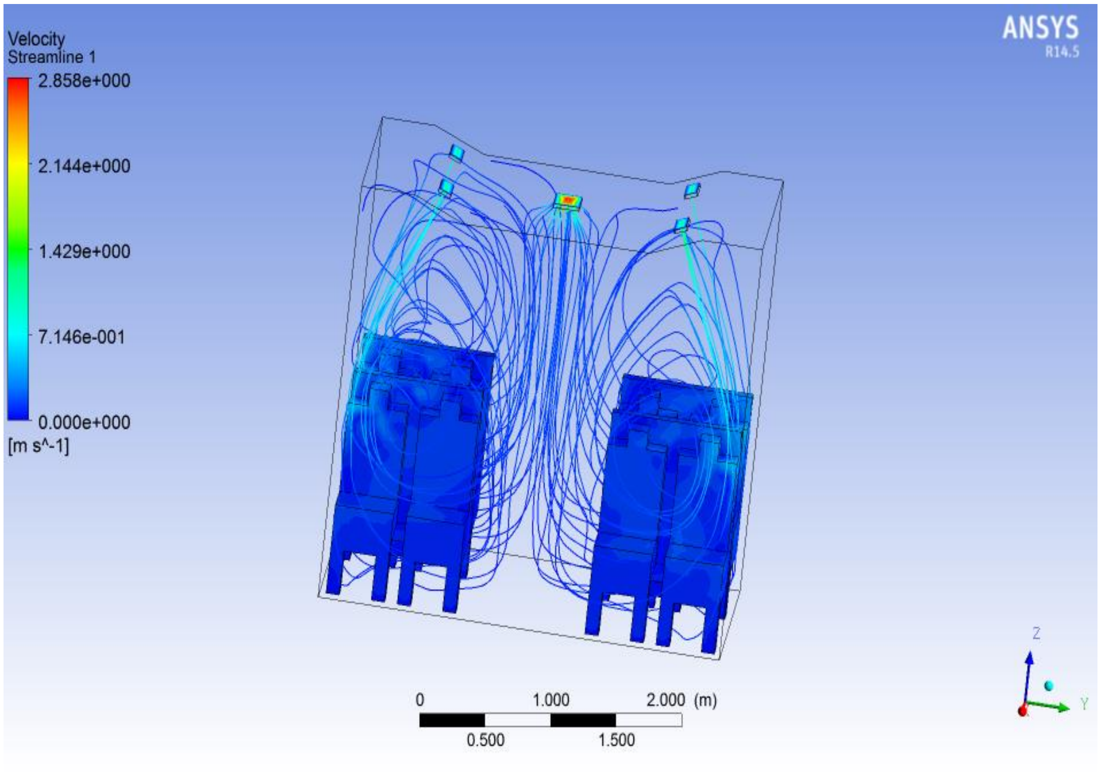


Figure 5.10 velocity streamlines with the effect of human load at 15 degree.

Figure 5.10 illustrate velocity streamlines in passenger train model with the variation maximum number of points 25. It also shows the effects the air flow in the passengers. As we can see the seated body, the magnitude of the velocity varied from 0 m/s to 0.7146 m/s.

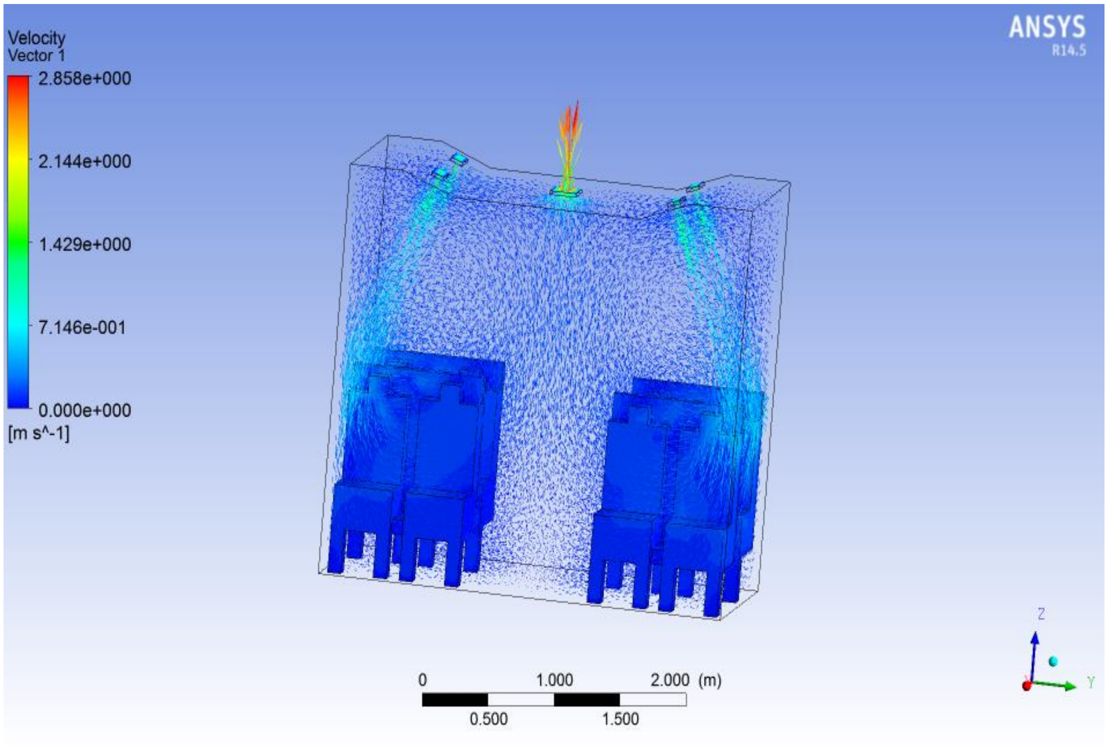


Figure 5.11 Velocity vectors with the effect of seat body at 15 degree.

In the figure above shows the velocity vectors with different maximum number of points in the passengers’ train model.

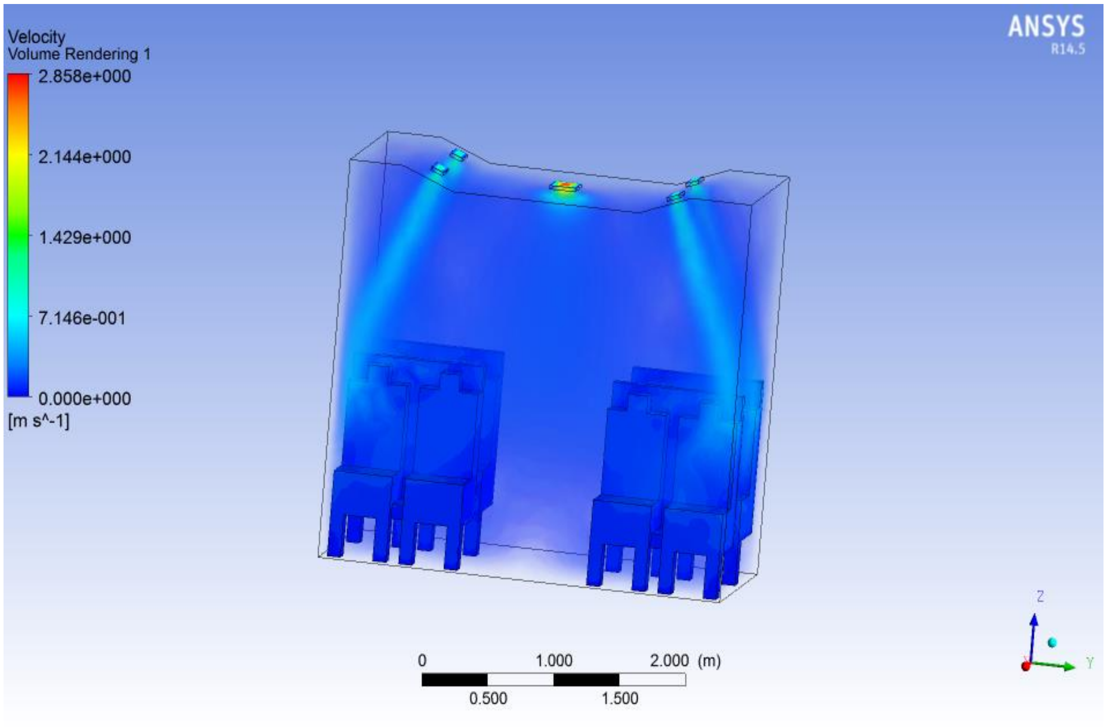


Figure 5.12 velocity volume rendering at 15 degree.

5.4 Velocity field of diffuser at angle of 25 degree.

The inside train compartment has been shown below which is characterized by the flow discharged from the air diffusers at angle of 25° with respect to celling. As we can see from the figures 5.13 through 5.16, the smallest velocity of 0 m/s is displayed blue, the highest velocity of 2.747 m/s is colored red.

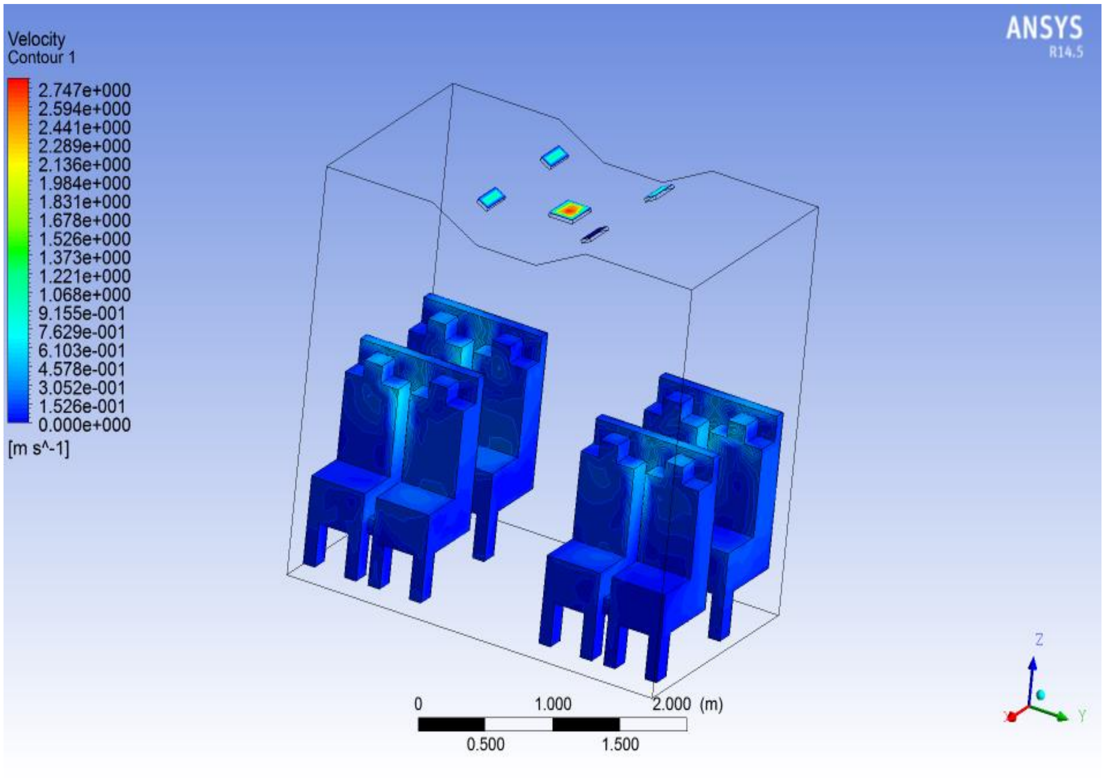


Figure 5.13 Velocity contours of air diffusers exhaust and seat body at 25 degree.

In the above figure illustrates the velocity distribution in the passenger train model. It shows the velocity field magnitude for inlet speed of 1.2m/s, air exhaust speed with a range from 0m/s to 2.747m/s.

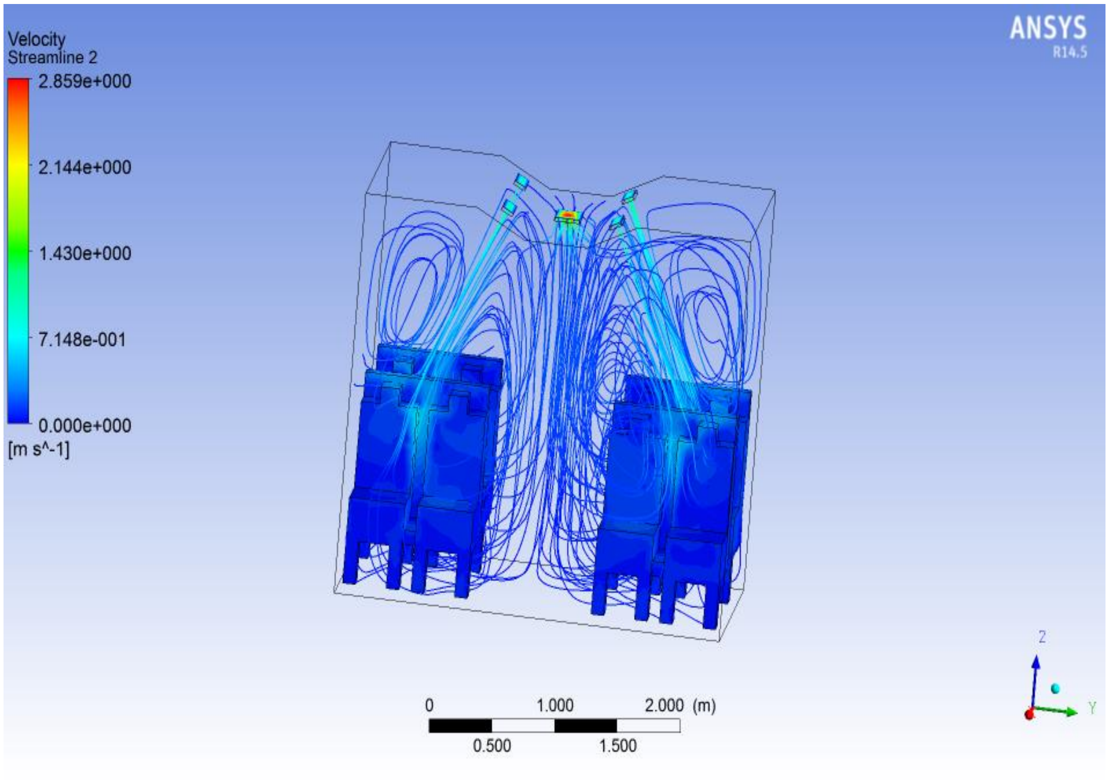


Figure 5.14 velocity streamlines with the effect of human load at 25 degree.

Figure 5.14 illustrate velocity streamlines in passenger train model with the variation maximum number of points 25. It also shows the effects the air flow in the passengers. As we can see the seated body, the magnitude of the velocity varied from 0 m/s to 0.7148 m/s.

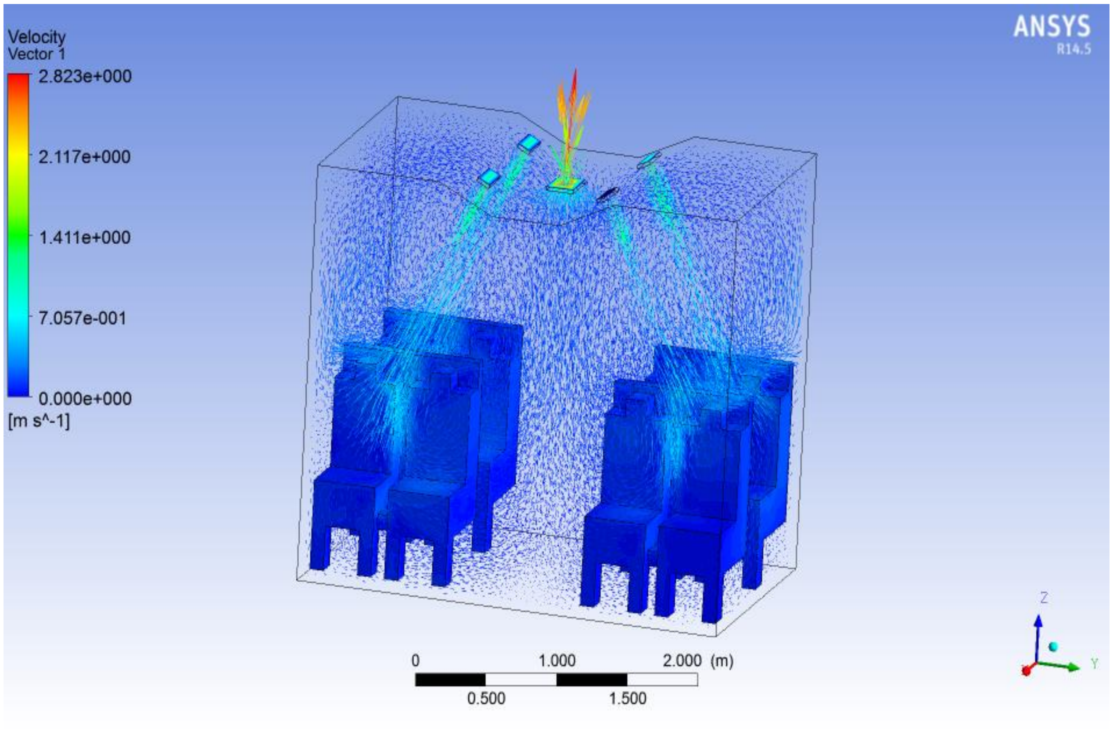


Figure 5.15 Velocity vectors with the effect of seat body at 25 degree.

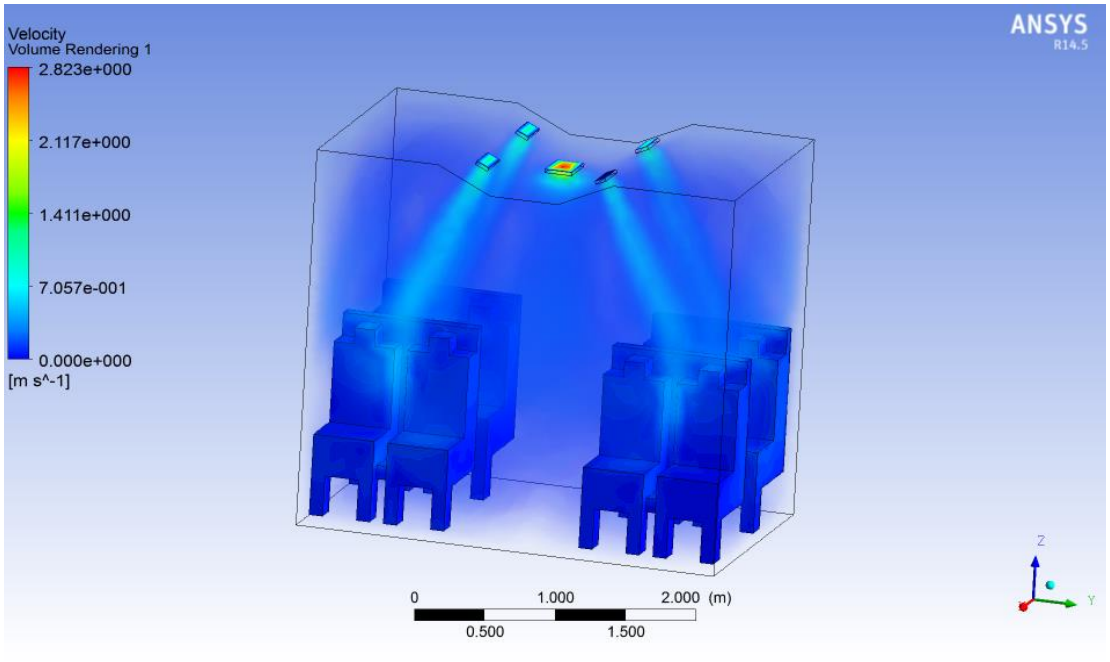


Figure 5.16 velocity volume rendering at 25 degree.

Generally, the train wall velocity contour has been observed zero m/s for all velocity field angles within different angle air diffusers arrangement. As we have seen in all results that is a lack of flow has been observed near the corners and the lower part of the compartment and also a critical dead zone occurs, most continuously at the corners and at the bottom most part of the passenger seats.

CHAPTER SIX

CONCLUSION, RECOMMENDATION AND FUTURE WORKS

6.1 Conclusion

The main objective of this paper is to investigate the air flow distribution in the interior of the passenger vehicle as well as to show the terminate speed (draft) below 0.7 m/s around the passenger's necks by using computational fluid dynamics (CFD). The (k- ε) model can be utilized successfully with turbulent flow to predict the air flow and thermal characteristics. As a result, the following conclusions were obtained:-

- In the vehicle compartment of magnitude of velocities varied from 0m/s through 2.086, 2.80 and 2.747m/s based on the air diffuser angle arrangement. For the three cases, the airflow did not cover the whole area on the same magnitude due to the model of passengers.
- Based on the 3-D simulations shows that the passenger necks area speeds on the air diffuser arrangement angle for zero, 15 and 25 degrees were an average 0.6861, 0.7146, 0.7057m/s respectively. So that, the passenger necks area speeds was 0.6861 m/s that is no draft sense on the passengers on the air diffuser arrangement at angle of zero degree. Because the terminate speed (draft) occurs above 0.7 m/s.

Generally, the most of in the vehicle compartment of air velocity is minimum that indicates a better air distribution attain there. As we have seen the results that are used to help trainees better understand on the air conditioning systems and especially the Ansys Fluent software.

6.2 Recommendation

For improving the CFD modelling to simulate better the real phenomena of airflow and heat transfer in real life air conditioned rooms, the following approach can be considered:

- Three-dimensional modelling: the 3-D model train coach in this study has not simulated the overall size. So to get more reliable result, the model takes real dimensions by using a better computer hardware performance.

- The simplification of don’t consider the heat transfer trough the external walls affect the results so a better study with some measurements of external wall temperature could be useful to improve the results.
- Taking into account the equipment’s in the vehicle compartment as obstacles to the air flow as well as heat transfer surfaces where needed. This will give distribution of the parameters of interest closer to the real environment.
- To simulate the train model, it has taken different assumption. This assumptions went into the modelling are questionable. So experimental studies will be necessary to develop a more accurate model.

6.3 Future Works

The focus of the present study was the optimization of air flow distribution inside passenger train model. The vent arrangements, energy consumption, and thermal comfort level were investigated for the worst case scenario. A further study may cover the following subjects:

- ✓ To analyze the thermal comfort level furthermore through adding solar radiation on the windows and adding heat generation from human body.
- ✓ To investigate the interaction of the air flow distribution system with different environmental situations and different human load.
- ✓ To analyze the air conditioned of air distribution with the combinations of seats and standing passenger.

All the future works will be implemented by using computational fluid software and also passenger trains.

REFERENCES

1. Lisa Kay Kelly, “Thermal comfort on train journeys”, Loughborough University Institutional Repository, June 2011
2. Ningbai Ningbai , “Analysis of Air Flow Distribution and Thermal Comfort in Hybrid Electric Vehicles”, B.Eng., Zhejiang University, 2011
3. Rameshkumar.A, Jayabal.S, and Thirumal.P, “CFD Analysis of Air Flow and Temperature Distribution in an Air Conditioned Car”, P.G. Scholar, 2Assistant Professor, Department of Mechanical Engineering.
4. https://en.wikipedia.org/wiki/Computational_fluid_dynamics.
5. Son Hong Ho, “Numerical simulation of thermal comfort and contaminant transport in air conditioned rooms”, University of South Florida, 2004.
6. ASHRAE Guideline 23P, Guideline for the Design and Application of Heating, Ventilation and Air Conditioning Equipment for Rail Passenger Vehicles First Public Review Draft.
7. Gabriel Haller, Thermal Comfort in Rail Vehicles, Climatic Wind Tunnel Vienna Professional article September 2006.
8. Abdlnaser Sayma, “Computational Fluid Dynamics”, 2009.
9. Wangda Zuo, ”Introduction of Computational Fluid Dynamics” FAU Erlangen Nurnberg JASS 05, St. Petersburg
10. Dmitri Kuzmin, “Introduction to Computational Fluid Dynamics”, Institute of Applied Mathematics University of Dortmund, http://www.featflow.de
11. Roberto de Lieto Vollaro, ”Indoor climate analysis for urban mobility buses: a CFD model for the evaluation of thermal comfort”, Dept. Engineering of University of Rome 3, Via della Vasca Navale 79 – 00146 Rome, Italy.
12. Tim Berlitz, Gerd Matschke,”Interior Airflow Simulation in Railway Rolling Stock”, Research and Technology Centre of the Deutsche Bahn AG Division for aerodynamics and air-conditioning.
13. Hao Yang , Yidong Wang,Taibi He, “The Analysis on the Effect of Passenger car Air Conditioning and Distribution with Different Inlet Parameters”, Chengdu Textile College, Chengdu Sichuan 611731, China.

14. Seohiro Kikuchi, and Kazuhide Ito," CFD Analysis of Indoor Environmental Quality in Commuter Train", Wind Engineering Research Centre, Tokyo Polytechnic, University, Atsugi, Japan Kawamotoi-ind.co.,ltd., Yokohama, Japan
15. Abhijyoth Reddy Vempati, "Computational Fluid Dynamics Investigation of Air Velocity And Temperature Distribution in a Room Equipped With Active Chilled Beam Air-Conditioning", University of Florida, 2011
16. Jalal M. Jalil and Haider Qassim Alwan, "CFD Simulation for a Road Vehicle Cabin", Educational Technology Department University of Technology, Baghdad, Iraq.
17. Dawit G. ,"Thermal Comfort For Passenger Train From Addis Ababa to Dire Dawa" AAiT School of Mechanical and Industrial Engineering, 2014
18. Mekonnen T., "Thermal Comfort for Passenger Train from Dire-Dawa to Djibouti", AAiT School of Mechanical and Industrial Engineering, 2015
19. Juan Carlos Ferri, Samuel Marín, "Creation Of A Model For The Study Of The Ventilation Air Diffusion Of The Falun Hospital", a CFD Based Integrated Approach
20. Awbi, H.B. E & FN Spon,"Ventilation of buildings" 1998.
21. ASHRAE, 2001. Ashrae Handbook: Fundamentals (SI Edition), American Society of Heating, Refrigerating and Air conditioning Engineers, Atlanta, USA.
22. W.A. Hageraats, "Fluid Dynamic Simulations of Vortex-Jet Interaction", Faculty of Aerospace Engineering, Delft University of Technology, April 3, 2006
23. FLUENT, "ANSYS Fluent User Guide", release 15, November 2013.
24. Patankar, T. C., "Numerical heat transfer and fluid flow", Mc. Graw-Hill. 1980
25. Zvar Baskovic, U., Lorenz,M. & Butala, V., "Adiabatic Flow simulations in the Air-Conditioned Vehicle passenger Compartment", Germany
26. http://www.fluent.com
27. http://www.ansys.com

APPENDICES

Appendix I

Cooling Design Temperature Profiles

Location: Aisha, Ethiopia

(Dry and Wet Bulb temperatures are expressed in °C)

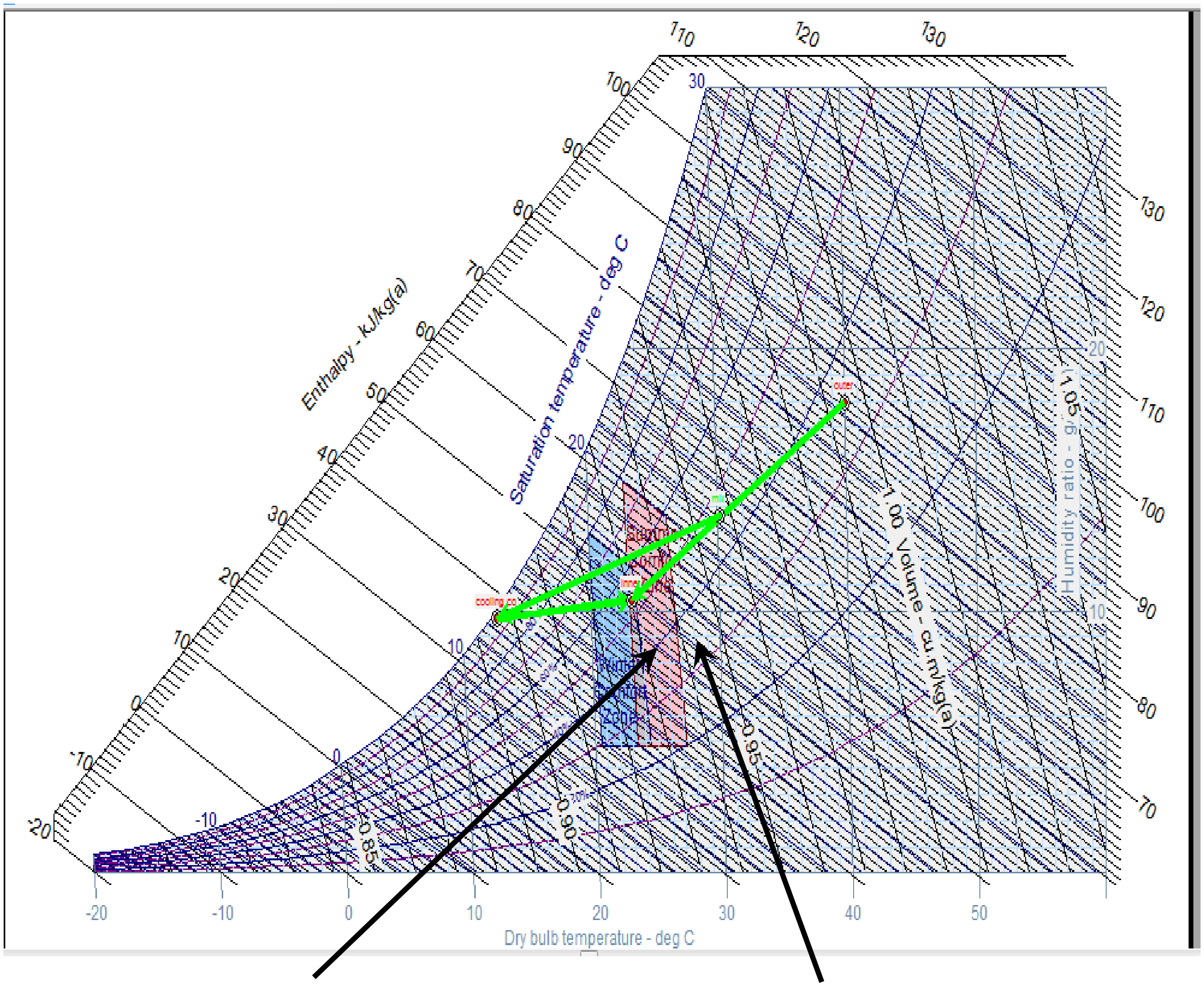
Hr	**January**		**February**		**March**		**April**		**May**		**June**	
	DB	**WB**	**DB**	**WB**	**DB**	**WB**	**DB**	**WB**	**DB**	**WB**	**DB**	**WB**
0000	21.7	19.5	22.8	20.2	24.5	21.4	25.0	22.1	25.6	22.7	27.2	23.4
0100	20.7	19.2	21.8	19.9	23.4	21.2	24.0	21.8	24.6	22.4	26.2	23.2
0200	19.8	19.0	20.9	19.6	22.6	20.9	23.2	21.6	23.7	22.2	25.4	22.9
0300	19.0	18.7	20.1	19.4	21.8	20.7	22.3	21.3	22.9	22.0	24.5	22.7
0400	18.3	18.0	19.4	19.2	21.1	20.5	21.6	21.2	22.2	21.8	23.9	22.6
0500	17.8	17.5	18.9	18.7	20.6	20.3	21.1	20.9	21.7	21.4	23.4	22.4
0600	17.6	17.4	18.8	18.5	20.4	20.1	21.0	20.7	21.5	21.3	23.2	22.4
0700	18.0	17.7	19.1	18.8	20.8	20.4	21.3	21.0	21.9	21.6	23.5	22.5
0800	18.8	18.5	19.9	19.3	21.6	20.6	22.2	21.3	22.7	21.9	24.4	22.7
0900	20.3	19.1	21.4	19.8	23.1	21.1	23.7	21.7	24.2	22.3	25.9	23.1
1000	22.5	19.8	23.6	20.4	25.3	21.7	25.8	22.3	26.4	22.9	28.1	23.6
1100	25.0	20.5	26.1	21.1	27.8	22.3	28.4	23.0	28.9	23.6	30.6	24.2
1200	27.9	21.3	29.0	21.9	30.7	23.1	31.2	23.7	31.8	24.3	33.4	24.9
1300	30.6	22.1	31.7	22.6	33.4	23.8	33.9	24.4	34.5	24.9	36.1	25.5
1400	32.6	22.6	33.7	23.2	35.4	24.3	35.9	24.8	36.5	25.4	38.2	25.9
1500	33.9	22.9	35.1	23.5	36.7	24.6	37.3	25.2	37.8	25.7	39.5	26.2
1600	34.4	23.1	35.6	23.6	37.2	24.7	37.8	25.3	38.3	25.8	40.0	26.3
1700	33.9	22.9	35.1	23.5	36.7	24.6	37.3	25.2	37.8	25.7	39.5	26.2
1800	32.8	22.6	33.9	23.2	35.5	24.3	36.1	24.9	36.7	25.5	38.3	25.9
1900	30.9	22.1	32.0	22.7	33.7	23.9	34.2	24.4	34.8	25.0	36.5	25.5
2000	28.7	21.5	29.8	22.1	31.5	23.3	32.1	23.9	32.6	24.5	34.3	25.0
2100	26.5	20.9	27.7	21.5	29.3	22.7	29.9	23.3	30.4	23.9	32.1	24.5
2200	24.7	20.4	25.8	21.0	27.5	22.3	28.0	22.9	28.6	23.5	30.3	24.1
2300	23.0	19.9	24.1	20.6	25.8	21.8	26.4	22.4	26.9	23.1	28.6	23.7

Cont.….

Hr	**July**		**August**		**September**		**October**		**November**		**December**	
	DB	**WB**	**DB**	**WB**	**DB**	**WB**	**DB**	**WB**	**DB**	**WB**	**DB**	**WB**
0000	27.2	23.2	26.7	23.0	26.1	22.7	25.0	22.1	23.3	21.4	22.2	20.2
0100	26.2	22.9	25.7	22.8	25.1	22.4	24.0	21.8	22.3	21.2	21.2	19.9
0200	25.4	22.7	24.8	22.6	24.3	22.2	23.2	21.6	21.5	20.9	20.4	19.6
0300	24.5	22.5	24.0	22.3	23.4	22.0	22.3	21.3	20.7	20.4	19.5	19.3
0400	23.9	22.4	23.3	22.2	22.8	21.8	21.6	21.2	20.0	19.7	18.9	18.6
0500	23.4	22.2	22.8	22.0	22.3	21.7	21.1	20.9	19.5	19.2	18.4	18.1
0600	23.2	22.2	22.6	22.0	22.1	21.6	21.0	20.7	19.3	19.0	18.2	17.9
0700	23.5	22.3	22.9	22.1	22.4	21.7	21.3	21.0	19.6	19.4	18.5	18.3
0800	24.4	22.5	23.8	22.3	23.3	21.9	22.2	21.3	20.5	20.2	19.4	19.1
0900	25.9	22.9	25.4	22.7	24.8	22.3	23.7	21.7	22.0	21.1	20.9	19.8
1000	28.1	23.4	27.6	23.2	27.0	22.9	25.8	22.3	24.2	21.7	23.1	20.4
1100	30.6	23.9	30.2	23.8	29.5	23.6	28.4	23.0	26.7	22.3	25.6	21.1
1200	33.4	24.6	33.1	24.5	32.3	24.3	31.2	23.7	29.6	23.1	28.4	21.9
1300	36.1	25.2	35.8	25.1	35.0	24.9	33.9	24.4	32.2	23.8	31.1	22.6
1400	38.2	25.6	37.9	25.6	37.0	25.4	35.9	24.8	34.3	24.3	33.2	23.2
1500	39.5	25.9	39.3	25.9	38.4	25.7	37.3	25.2	35.6	24.6	34.5	23.5
1600	40.0	26.0	39.8	26.0	38.9	25.8	37.8	25.3	36.1	24.7	35.0	23.6
1700	39.5	25.9	39.3	25.9	38.4	25.7	37.3	25.2	35.6	24.6	34.5	23.5
1800	38.3	25.6	38.1	25.6	37.2	25.5	36.1	24.9	34.4	24.3	33.3	23.2
1900	36.5	25.3	36.2	25.2	35.4	25.0	34.2	24.4	32.6	23.9	31.5	22.7
2000	34.3	24.8	34.0	24.7	33.2	24.5	32.1	23.9	30.4	23.3	29.3	22.1
2100	32.1	24.3	31.7	24.2	31.0	23.9	29.9	23.3	28.2	22.7	27.1	21.5
2200	30.3	23.9	29.8	23.8	29.1	23.5	28.0	22.9	26.4	22.3	25.3	21.0
2300	28.6	23.5	28.1	23.4	27.5	23.0	26.4	22.4	24.7	21.8	23.6	20.6

Appendix II

Psychometric chart at 721m elevation



Winter comfort zone

summer comfort zone

Appendix III

1 Technical Description of YZ25G Hard-seat Passenger Car

1.1 *General Technical Specification*

Conditions of Use:

- Maximum configuration: 20car per trainset
- Ambient temperature: -400C ~ +400C
- Maximum relative humidity ≤95%
- Platform height, Suitable for platform height with 300mm 、500mm and1250mm, distance between platform edge and railway center is 1750mm.
- Maximum line gradient ≤30‰

1.2 Main technical specification

Railway gauge 1435mm
Maximum operation speed 120km/h
Emergency brake distance on straight line (with 30% overload and with initial speed 120km/h) ≤800m
Minimum negotiable curve

- Single car 100m
- Coupling 145m

Ride index W≤2.5

Noise (120Km/h) ≤68dB（A）

1.3 Main dimension

Carbody length 25500mm
Carbody width Approx 3105mm
Height between rail top to car top (empty car) 4433mm
Vehicle center distance 18000mm
Height from rail top to coupler center (empty car) 880+10mm or 880-5mm
Height from rail top to inter car crossing pedal (empty car) 1333mm
Height from rail top to floor surface (empty car) 1283mm
Carbody center plate to rail top (empty car) 780mm

1.4 *Carbody steel structure*

Body steel structure will use overall carrying beam cylindrical structure, side wall will use flat plate. Thickness of sheets and profiles which ≤ 6mm will use nickel chromium weathering steel, thickness which ≤ 2.5mm will use 05CuPCrNi, thickness which between 3 ~ 6mm will use 09CuPCrNi-B (Q295GNHL), plate thickness more than 6mm will use ordinary carbon steel. Air conditioning seat, toilet and wash room iron floor whichever easily corrosive will use stainless steel.