DESIGN AND ANALYSIS OF SMOKE AND FIRE IN ENCLOSED SPACES USING CFD

A thesis submitted in partial fulfillment of the requirements for the degree of

Master of Technology in CAD/CAM

By

MAHESH PATIL

(14MCD1046)

SCHOOL OF MECHANICAL AND BUILDING SCIENCES

VIT UNIVERSITY
(Estd. u/s 3 of UGC Act 1956)
www.vit.ac.in
Vellore • Chennai
CHENNAI CAMPUS
Vandalur - Kelambakkam Road, Chennai - 600127

April, 2016

DECLARATION

I hereby declare that this written submission represents my project description and have included facts and ideas from various sources in my own words. I have adequately cited and referenced the original sources. I also declare that I have adhered to all principles of academic honesty and integrity. I have not falsified any idea, data, fact, source in my submission. I understand that any violation of the above will be a cause for disciplinary action.

Place: Chennai **MAHESH G PATIL**
Date:

ABSTRACT

As per the record in last few years, there were millions of cases of vehicle fire resulting in fatal casualties. Prevailing investigations showed that many people died as results of suffocation due to smoke compared to burn injuries. Therefore nowadays, many car parks are designed with ventilation system for circulation of smoke in case of fire. The use of CFD package has also helped to design efficient ventilation system in underground car parking garage with reduced need of experimental testing. Some of the widely used CFD packages for smoke analysis include software like FLUENT and Pyrosim FDS. This study focusses on design and analysis of ventilation system in underground car parking garage using CFD packages. Comparative study between these two simulation software were conducted on the basis of their mathematical models, simulation field, and simulation theory. The results showed that Pyrosim FDS is appropriate software in field of building fire simulation because of its flexibility in model generation, assignment of boundary conditions, accuracy of results, and reduced computational time and resources as compared to FLUENT. In order to optimize the design of ventilation system in car parking garage, Taguchi design of experiment approach was used coupled with CFD runs. The parameters selected were angle of jet fans, number of jet fans and type of exhaust with two levels each. The results provided the best optimum combination of parameter and their levels. The study also laid down guidelines to design an efficient ventilation system in underground car parking garage using CFD.

ACKNOWLEDGEMENT

I would like to express my deep sense of gratitude and respect to my project guide **Dr. Shymakumar M.B.** (Associate Professor, SMBS, VIT University, Chennai) for his excellent guidance, suggestions and constructive criticism. Working under his supervision greatly contributed in improving quality of my research work and in developing my engineering and management skills

I am greatly indebted to **Dr. Kiran Bhagate (**Managing Director), Mr. Sumedh Suryawanshi (Senior Engineering Analyst) and the staff of IDAC India Pvt. Ltd., Pune for their guidance, patience, inspiration and constant encouragement during the period of this work.

I would also like to thank **Dr. Davidson Jebaseelan** (Program Chair - M. Tech CAD/CAM), **Dr. K. Janardhan Reddy** (Dean - SMBS, VIT University, Chennai) and faculty members of VIT University, Chennai for their support throughout the course.

Finally I would like to thank my family members and all my friends who inspired, motivated and supported me throughout the course of work and every hand that rendered help directly or indirectly

Place: Chennai **MAHESH G PATIL**
Date:

CONTENTS

LIST OF FIGURES

LIST OF TABLES

Chapter 1

INTRODUCTION

In commercial car parking space, when fire breaks out it is difficult to handle with normal method. To overcome this situation we need fire safety design.

Figure 1 Car fire in parking

Ventilation in underground car parking is always recommended to limit concentrations of carbon monoxide and other vehicle emissions and to remove smoke and heat in the event of a fire. It is also important for architects, building managers and owners to confirm that the installed system will clear smoke as required and meet fire regulations even before the system is installed. Since car park ventilation systems generally serve dual purpose, i.e. to provide ventilation for vehicle emission and clear smoke in the event of fire. In absence of ventilation, underground car park presents several internal air quality problem, most serious problem is emission carbon monoxide (CO). When CO level exceeds its threshold value, it directly combines with human haemoglobin which lowers the oxygen carrying capacity of blood. Level of CO in car parking garage vary depending on following factors

- Number of cars running in garage
- Length of travel and repairing time of car in garage
- Emission rate of vehicle
- Hence it is essential to maintain level of CO below its threshold value in underground car park.CO level should not be more than 35 ppm for all parking garage (Moncef Krarti, 2001).

The principal use of CFD analysis in car parking is required for jet fan distributor. They must prove that its equipment meets civil building coding and they need to take design approval for distribution of air is sufficient to effective ventilation of car park under normal conditions as well as in emergency situations. Basically there are two types of coding used in fire ventilation.

- ASHARE (American Society of Heating, Refrigerating & Air-Conditioning Engineers)
- NBC (National Building Code of India)

ASHRAE was firstly released in 1894 to presents a source of technical standards. This organization also promotes a code for HVAC professionals. The operation of automobiles serves two concerns. The most serious concern is the emission of carbon monoxide. The second affecting concern is the presence of oil and gasoline fume. Principal requirement of ventilation is to dilute carbon monoxide to acceptable levels will also allow the other contaminants satisfactorily. In order to save energy, fan systems must be controlled by carbon monoxide sensors with multiple fans. In multilevel parking garages usually independent fan systems installed, each under individual control.

NBC Code was first founded in 1970 by Planning Commission of India and then revised in 1983. Thereafter three amendments were issued, two in 1987 and last is in 1997. NBC is a national instrument providing guidelines for regulating the building construction activities across the country. It is provided as a Model Code for adoption by all agencies which involved in building construction works. These agencies consist Public Works Departments, government construction departments, private construction agencies. The Code mainly describes administrative regulations, general building requirements, fire safety requirements; structural design and construction. According to NBC coding of India, Ventilation rate will confirm that the Carbon monoxide (CO) level is maintained within its limiting value as 29mg/m3 with peak levels not to exceed 137mg/m3. In the event of fire, there should be a clear view at height of 1.7 metres along the distance of 30m at 45 minutes after the fire get started.

TYPES OF VENTILATION

1) NATURAL VENTILATION

If the car park is above ground level then generally it is better to provide natural ventilation, because compared to other ventilation (mechanical or impulse) it is simple, cheaper to purchase and especially there are no running costs. Such car parks are having a higher level of fire resistance.

Open sided car park is one where the ventilation openings in the walls which is equal to at least 5% of the floor space. So if car park floor area is 1,000m2 then 50m2 will be ventilation area needed. In addition to this there is further improvement is needed like ventilation area should split, so that it will be quarter of it is down each of two at opposite sides and reasonably equally spaced so it can create cross flow through the car park. The remaining 50% of the ventilation can be put in location wherever you can fit for room.

2) MECHANICAL VENTILATION

Ventilation requirement occur at densely occupied spaces. In recent years due to continuous improvement in ventilation systems, Impulse ventilation is more advantageous than ducted ventilation system. Mechanical ventilation is considered as conventional ventilation system, it has provision of fresh air fan and exhaust air fans with ducting arrangement as per requirement. These fans can be operated based on timing control irrespective on smoke level or CO concentration, In addition to this it is difficult to install and require more cost as well as skills for installation.

Figure 2 Mechanical ventilation system

3) IMPULSE VENTILATION

Impulse ventilation is modern method of ventilation. This system contains a set of fresh air fans and exhaust fans no need of bulky ducts for ventilation. Through a well inlet vent fresh air is pumped in car parking area and the exhaust fan sucks these air and throw it out. It is also used for energy conservation of fan, it has provision of sensors to detect level of CO. When CO level reaches to its threshold value, then fan will start operating. Ductless ventilation improves quality and efficiency of ventilation in car park. CFD analysis, is used to decide the number and location of Jet fans for effective and optimum performance.

Impulse ventilation system(IVS), based on jet fans location under the enclosed car park ceiling, induces contaminated fresh air or smoke towards pre-designed extract points, where a conventional extract system can exhaust them. In general use, IVS removes smoke containing carbon monoxide, produced by Vehicles fire. IVS was established from tunnel longitudinal ventilation system. Its function partially likes positive pressure ventilation (PPV) studied by Kerber at NIST. The use of IVS increases due to flexible installation and its lower cost of building excavation, comparing with ducted ventilation system. Only limited research has been done on Impulse Ventilation System for smoke control in underground car park. Simultaneously, some IVS design companies have been working on many projects.

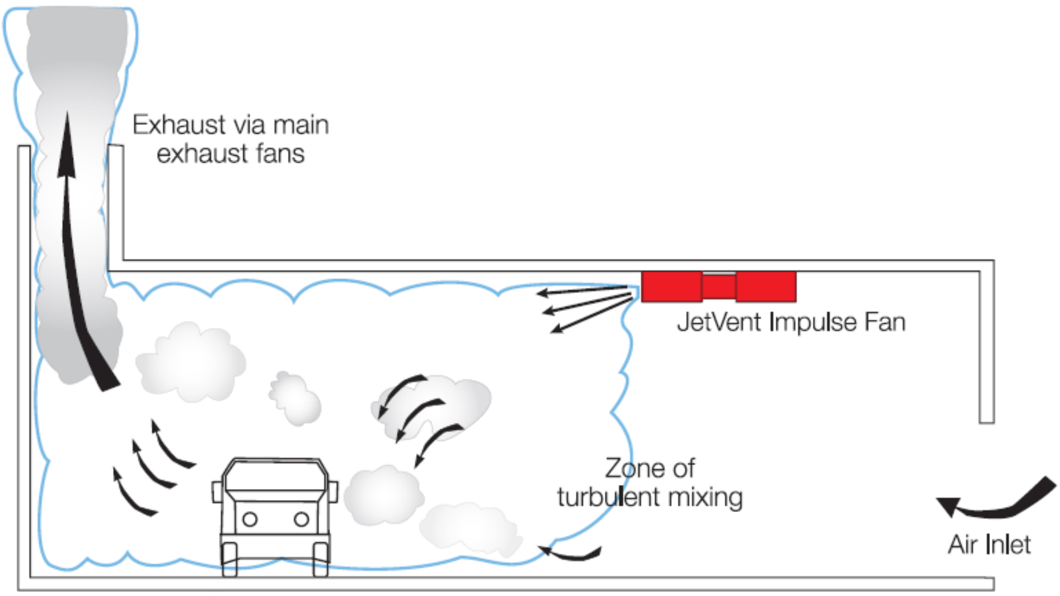


Figure 3 Impluse ventilation system

During event of Fire, the volume of air to be handled by jet fan is in the order of 1.25 lakh CFM. In such situation ductless system serves better compared to ducted system. Duct ventilation are associated with pressure drop, hence larger size Fresh air/exhaust fans are required. Location of jet fan decided using CFD Analysis in such a manner that it will help for given size of fresh air/exhaust fans to suck/exhaust fresh air respectively, so that smaller size (HP).Fresh air/exhaust fans required In the situation of low traffic in Basement, depending on level of CO concentration, only a set of Jet fans may be switched ON to dilute CO concentration, while in case of ducted system higher capacity size (HP) Fresh air/exhaust fans need to kept ON most of the time

Advantages of Impulse ventilation

- They protect environment since toxic fumes not released to environment
- Fully installed system ready to operate at available lower cost than bulky duct system
- An expensive duct with its external blower system that is always difficult to maintain is not required.
- They are portable and can relocate easily
- They are easy to re position to meet requirements of customer
- Improved air quality and efficient in Operation ,Optimum use of Car Park area
- Flexibility in installation, quicker installation
- Effective smoke management

Typically IVS consist of four main components, i.e., jet fans, extract /supply fans, sensor system and control system

I. JET FAN

The principles of jet fan ventilation in parking lots is an important for smoke control in enclosed spaces, after natural ventilation and ducted mechanical ventilation the most commonly used method in ventilation is known as jet fan ventilation and is based on change of speed criteria. The jet fan ventilation method can be optimized through continuous testing and install into the parking lot security systems. All fresh air is drawn by fans and ducts in conventional mechanical ventilation systems. This is worked for both ventilation system. Their basic principle is to provide fresh air in throughout car parking space. The airflow rate has to be keep as low as possible to prevent pressure drop. Meanwhile this means the channels must be relatively wide, and therefore required area should be larger. But in case of jet fan ventilation a different approach is used. In this system, small amount of contaminated air is sucked into a fan and then it is thrown away at a higher speed. When the contaminated air hits in front of the jet fan, it pushes that air forward and pulls the surrounding fresh air at the same time. In this way, the movement of all the surrounding air will be done without using channels.

Figure 4 Jet Fan

Whole parking lot treated as an air duct. The main principle behind the jet fan ventilation system is the same as it is used in rockets where a small amount of contaminated air are thrust at larger speed. There are some advantages of this ventilation system, such as uniform air, distribution, and flexible installation, effective ventilation in the parking lot, Space-saving, energy and cost savings. The capability of jet fans is expressed by using thrust. The thrust is the force created by jet fans, is resented in Newton [N] and it's the multiplication of variable in speed and mass flow rate. In practical use, it is always suggested that the distance between the nearest beams need to be 0.5 meters at the entrance and 2 meters at the outlet for the efficient yield in jet fan ventilation. Height of the beams need not be more than 0.4 meters. Else, the beam height have to be compensated either by hanging of fans from the ceiling or by enlarging fan distance between the closest beam. All amount of energy has to be transported in the form of speed from the discharge air to surrounding. Fan always remains at their place when air is delivered forward. As a result of this drag force will generate, the amount of moving air is always higher than the amount of air passing through the fan. The amount of moving air is always same in different sections of the fan. Variation in speeds will be achieved in particular sections due to the size of the system. Not just the size of fan, jet fan number, size of parking and its design plan but also depend on the purpose of the system. It means ventilation used for the CO evacuation or smoke control.

In case of a fire, jet fans will not operate immediately with smoke evacuation speed, and will remain off for a certain period of time until the fresh air and exhaust fans start working at their maximum power. This will give time for people in the parking lot to escape. Jet fans start to work only after contaminant level reaches to its threshold value, and push the air towards exhaust. This gives two benefits. First, the smoke acquire only restricted space, and this allows us to detect fire's location and exit. Second, it confirm that the temperature is low closer to the fire location, so that firefighters can come nearer to that location.

II. EXTRACT/ SUPPY FAN

It are often ducted or supplied with associate inlet bell for open (or unducted) inlet installations. The power to fine-tune the system performance through blade angle adjustment insures the user of an extremely economical, economical, versatile, quiet, and long-running fan.

Figure 5 Exhaust Fan

The direct drive exhaust fan is that the good alternative for steps ventilation. The need for steps pressurization has increased because of a lot of tight public building codes. Throughout associate steps, the requirement to make a positive pressure inside the steps enclosure ensures a secure exit means. For this application, the axial uses a maintenance free, direct drive motor, and therefore the blade adjustability permits fine-tuning of the system to the optimum purpose of rating

The proprietary axial blade style provides the client with the flexibility to change the blade angle so as to vary the performance while not loosening or removing any hardware. The resilient O-ring acts sort of a spring (producing friction between hub and blades therefore the blade doesn't flip throughout operation), to preload the blade holding disc and hub arrangement, whereas providing a cantering force. It insures a correct work of blade and hub. The O-ring additionally minimizes vibration between the blades and hub, reducing the potential for material fatigue. The angle of the axial blades may be adjusted while not tools. With the fan wheel in motion, the friction between the blade holding disc and therefore the hub itself is increased because of force that holds the blade firm and prevents any unsought modification in blade angle.

Computational Fluid Dynamics is a field of science involving fluid flow, heat and mass transfer and associated phenomena like chemical reactions by means of computer-based simulation. It is a simulation software used for fluid study. The set of partial differential equations is to be solved on given geometrical domain. CFD technique has many application in industrial area. This project uses two CFD software, i.e. FLUENT and FDS for design of smoke and fire movement in car parking.

The need of CFD in car park is primarily important to confirm that there is enough air movement in all areas of car park so that we can predict number of air exchanges required. CFD provides cost effective and accurate alternative for scaled model testing with variations on the simulations. It is also used to confirm that number of jet fans required and their location for installation. The CFD software like Fluent and FDS have been developed to analyse fire and smoke in car parks and to study the behaviour of air, CO, smoke, temperature etc. in that environment. CFD is much easier and cheaper option to carry out initial iteration to optimise the system before installation.

FLUENT is most powerful CFD software tool available, empowering CAE engineers. ANSYS Fluent software consist of broad physical modelling needed to model flow, turbulence, heat and mass transfer, and industrial applications ranging from air flow over an aircraft to combustion in an engine, from various bubble columns to different oil platforms, from blood flow in human body to semiconductor ventilation, and from enclosed room design to drainage water treatment plants. Some Special models serves the software the ability to model combustion cylinder, acoustics, turbomachinery, and multiphase modelling systems have broadened its capability.

FLUENT software adopts FVM (finite volume method), which distribute calculated area into a number of control volume. Each control volume has a node as an important factor. Finding the discrete equation by maintaining the volume control equation for the integral. During the process of finding, the function and its first derivative of the interface always need to be assumed. There are three numerical algorithms of Finite volume method, while all other commercial CFD software can provide only one of them. FLUENT has powerful unstructured grid generation capacity.

Fire dynamic simulator (FDS) was firstly released by NIST (National institute of standards & technology) in 2000. FDS is Computation fluid dynamic model driven for design of smoke handling systems and sprinkler activation studies. It is also used for residential and industrial fire reconstructions. During its complete development, FDS has been targeted at solving practical fire in fire protection engineering, while simultaneously serves a tool to study fundamental fire dynamics. FDS solver setting always used to solve numerically formed Navier-Stokes equations for low-speed (Mach number < 0.3), thermodynamically-driven flow with an application on smoke heat transfer and mass transport from fires. Smoke view is a separate visualization that is used to see the results of an FDS simulation.

To study the comparative research in the FLUENT and FDS, lot of emphasis is given on simulation of underground car parking garage. Many research papers are based on validation using either FDS or FLUENT. It is interesting to study comparative results of software in order to get interpretation of emission ventilation and fire ventilation system design in indoor parking lot with jet fans.

The use of CFD analysis before installation is to decide location of jet fan in such a manner that it will help for given size of fresh air/exhaust fans to supply/exhaust fresh air respectively.

Design of Experiment (DOE) is powerful statistical technique introduced to study the effect of multiple variables simultaneously. DOE is highly be effective for:

- Optimizing product and process design

- Studying the influence of individual factor on performance and determining which factor has more influence, and which one has less.
- Finding which factor should have higher tolerance and which tolerance should be relaxed.

In industry designed experiments can be used to symmetrically investigate the process or product variable that influence the product quality after you identify process conditions and product component that influence product quality, you can direct improvement effort to enhance product manufacturability, durability, quality and field performance .Because resources are limited, it is important to get most the most information from each experiment you performed. Well-designed experiment can perform significantly more information and often require fewer runs than haphazard or unplanned experiments. In addition well designed experiment will ensure that you can evaluate the effects that you have identified as important (Philips, 2008). Design of experiment can effectively implemented for optimisation of ventilation system. For proper ventilation management it is essential to understand the contribution and the effect of different components in the system and take necessary step to optimize the ventilation system of car park.

1.1 PROBLEM STATEMENT

- Comparative research on FLUENT vs FDS numerical simulation software for car parking garage
- Design optimization of ventilation system using Taguchi design of experiment.

1.2 OBJECTIVE

- To examine and understand smoke and fire control capacity of IVS (Impulse ventilation system)
- To compare numerical simulation of FLUENT vs FDS for smoke spread in car parking garage
- To predict the most appropriate simulation software for modelling smoke and fire in car park.
- Use of DOE for design optimization of car parking garage ventilation
- To discover most affecting parameter on design ventilation system

The first research on IVS implementation for underground car park was done by viegas (2001). The enclosed car parks should have means of ventilation, it may be natural or mechanical, that serves the vehicle emission released by the engines passing through and to control the smoke released by vehicle fire. In years of 2000, a new mechanical ventilation system released, which is based on the use of axial ventilators (jet fans) mounted under the car park ceiling. Jet fans create necessary momentum to promote the internal ventilation. Like this way the inlets and exhausts may be points of study in some of the underground car park. Due to the general construction of car parking, the flow will be rather complex, so that ventilation affecting parameters like the jet fans position, their orientation and volume flow rate must be carefully chosen. While design recommendations for this kind of projects were missing, the use of CFD software, as alternative tool in the evaluation of ventilation design. In this literature CFD simulations were analysed, using FDS, to estimate the performance of an available ventilation system, installed in car park. They concluded that jet fans have an important role in smoke control and strong distribution of smoke reduces temperature and contaminants of smoke.

S. lu, Y.H. Wang, has examined that smoke control capacity of Impulse Ventilation System in car park and made comparison with ductwork system. The effect of smoke control simulated using CFD software such as FDS. In this literature, 10 simulation cases was taken under study for smoke control capacity of IVS. These simulation cases consist of number of jet fan, jet fan velocity and extract rate, as these are important parameters in IVS design. Its results show that the smoke control ability of impulse ventilation system is depends on jet fan numbers and increasing extract rate. They concluded that too high speed jet fan may cause severe smoke recirculation problem and an IVS is able to control smoke movement and bring it to extract points under two typical varying fire locations

Wang Binbin has explained on comparative research on FLUENT and FDS's numerical simulation of smoke spread in subway platform fire, by comparison between simulation software field and simulation technology theory as well as the contrast of the physical model of fire smoke spread they compared the numerical results of the two simulation software respectively. Fire spread and smoke movement in a large underground car park using FDS. They simulated the effect of ventilation on the fire spread and smoke movement and results gave heat release rate, oxygen and soot concentrations as well as temperature X.G. Zhang

Chapter 2

METHODOLOGY

The aim of this study was to formulate a ventilation design of fire smoke in car parking garage. Comparative research was carried out by simulating car fire using two different simulation software FLUENT and FDS. In stage 2, FDS was used to carry out design optimization study of car park. Taguchi Design of Experiment approach was used to find best combination of number of fan, angle of fan and extraction rate

For this project the methodology followed was:

- Literature survey
- Comparative Research on FLUENT and FDS's Numerical Simulation for car parking garage.
- Identification of critical factors for design optimization
- Implementation of Taguchi method
- Modelling and analysis of tests using CFD package
- Analysis of data using Minitab software
- Results and discussion
- Documentation and preparation of final report

CFD simulation of car park used to show ventilation design and numerical simulation using two simulation software FLUENT and FDS .This project is divided in to two stages. In first stage of this study, comparative study of FLUENT and FDS. Comparison made between their mathematical model, simulation field, simulation theory and lastly their numerical results for current simulation case of smoke control in single car parking garage. In second stage of study includes use of field simulation software i.e. FDS for the design optimisation of ventilation in car park using Taguchi design of experiment approach. Taguchi method is used for optimization of ventilation design in order to decide size of exhaust, number of jet fans required, location of jet fans, and orientation of jet fans.

2.1 IMPORTANT COMPONENTS OF CAR PARKING GARAGE

The important car parking garage are:

- Inlet vent
- Structural walls, ceilings and floor.
- Jet fan, jet fan casing.

- Stationary car body
- Outlet vent

2.2 CFD (COMPUTATIONAL FLUID DYNAMICS) APPROACH

CFD or computational fluid dynamics forecasts quantitatively, when fluid are flowing, frequently with the difficulties of concurrent flow of phase change (e.g. melting, freezing, boiling), chemical reaction, and mechanical drives (e.g. Fans, pistons etc.), Stresses and displacement of occupied or neighboring solids. Computational fluid dynamics (CFD) is the branches of fluid mechanics that uses numerical methods and algorithms to resolve and analyses problems that involve fluid flows. Computers are used to get huge calculations needed to simulate the interaction of fluids and gases with the intricate surfaces used in engineering. Even with basic equations and high-speed supercomputers, only approximate solutions can be attained in many cases. On successful research, however, may produce software that advances the correctness and speed of intricate simulation conditions such as turbulent flows. One technique is to discretize the three-dimensional domain into small cells to generate a volume mesh or grid, and then by applying an appropriate algorithm in solving the equations of motion (for inviscid Euler equations and for viscous flow Navier- Stokes equations). Also this type of mesh can be either irregular (for Example consisting of triangles in 2D, or pyramidal solids in 3D) or regular. If we chooses not to continue with a mesh-based technique, a number of substitutes exist, notably Smoothed particle hydrodynamics (SPH), a Lagrangian technique of solving fluid problems, Spectral methods, a technique where the equations are projected onto basis functions like the sphere-shaped harmonics and Chebyshev polynomials, which simulates a matching mesoscopic arrangement on a Cartesian grid, in place of solving the macroscopic system. When all of the applicable length scales can be determined by the grid then we can directly solve the laminar. One technique is to discretize the three-dimensional domain into small cells to generate a volume mesh or grid, and then by applying an appropriate algorithm in solving the equations of motion (for inviscid Euler equations and for viscous flow Navier- Stokes equations). Also this type of mesh can be either irregular (for Example consisting of triangles in 2D, or pyramidal solids in 3D) or regular. If we chooses not to continue with a mesh-based technique, a number of substitutes exist, notably Smoothed particle hydrodynamics (SPH), a Lagrangian technique of solving fluid problems, Spectral methods, a technique where the equations are projected onto basis functions like the sphere-shaped harmonics and Chebyshev polynomials, which simulates a matching mesoscopic arrangement on a Cartesian grid, in place of solving the macroscopic system. When all of the applicable length scales can be determined by the grid then we can directly solve the laminar flows and turbulent flows by Navier- Stokes equations. However, the range of length scales suitable to the problem is greater than even today's immensely parallel computers can model. In these cases, turbulent flow simulations need the introduction of a turbulence model. In many examples, to deal with these scales we need large eddy simulations (LES) and the Reynolds-averaged Navier-Stokes equations (RANS) formulation, with the k-e model or the Reynolds stress model. The Navier-Stokes equations solve other equations. These other equations can comprise those relating species concentration (mass transfer), chemical reactions, heat transfer, etc. for the simulation of more complex cases connecting multi-phase flows (e.g. liquid/gas, solid/gas, liquid/solid), non-Newtonian fluids (Such as blood), or chemically reacting flows (such as combustion) In CFD calculations, there are three main steps.

- Pre-Processing
- Solver Execution
- Post-Processing

Pre-Processing is the step where the modeling goals are determined and computational grid is created. In the second step numerical models and boundary conditions are set to start up the solver. Solver runs until the convergence is reached. When solver is terminated, the results are examined which is the post processing part.

2.2.1 PRE-PROCESSING

In this study, the aim is to investigate the performance of ventilation system by comparing two CFD software and to optimize design of ventilation for its efficient performance. So, an adequate numerical model is to be created. Pre-processing is the most time consuming and least knowledge requiring part.

2.2.1.1 GEOMETRY CREATION

There are two important points here. The first one is the size of the domain, and the second one is the density and quality of the computational grid. Model size is the computational domain where the solution is done. It is important to build it as small as possible to prevent the model to be computationally expensive. On the other hand it should be large enough to resolve all the fluid and energy flow affecting the heat transfer inside the enclosure.

In our problem, domain is selected to be the whole enclosure. To make the models computationally inexpensive such that the computer resources available can solve.

2.2.1.2 MESH GENERATION

The second part of pre-processing is the mesh generation. Mesh is the key component of a high quality solution. It is important to have a good mesh to have an accurate solution. There are some general guidelines to create a good mesh. A good mesh should be fine enough with high quality cells and a good distribution of these cells is essential. Moreover the mesh should not have more cells than the available computer resources can handle. Some important guidelines for meshing are given below

- **Resolution:** It is up to the user to choose the resolution of the mesh. But it should be fine enough to capture the most flow features and the solution at the end should be grid independent.
- **Quality:** There are three quality parameters that need to be checked.

  One is face alignment; it is the parameter calculating skewness of cells. Elements whose skewness is more than 0.85 are severely distorted and it should be avoided to have such distorted cells in the critical regions. The second quality parameter is the aspect ratio. It is defined to be the ratio of the largest side of the cell to the smallest side. Cells that are too slender should not be preferred. The third parameter is the volume quality of cells. Extremely small cells may create difficulty in convergence. But when such cells exist, double precision solver may be used.
- **Smoothness:** A good distribution of cells is very important. Critical parts need to be fine meshes as compared to non-critical parts. An important point in mesh distribution is that the transition from smaller cells to larger ones should be smooth.

2.2.2 SOLVER EQUATION

2.2.2.1 FLOW CONFIGURATION AND BOUNDARY CONDITIONS

The compressibility effects and turbulence inside the car parking garage are the parameters changing the governing equations to be solved. Their roles in this study are explained as well as boundary conditions.

- **Compressibility:** The fluid in the domain is air. The compressibility effects are ignored due to the low speeds. Although air is a compressible fluid, incompressible flow assumption is valid as long as the Mach number is smaller than 0.3 (Panton, 1984) The fans that will be used in this study deliver less air and Mach number is much less than 0.3; therefore incompressible flow assumption will be used in this study.
- **Turbulence:** The flow inside the system is turbulent regardless the Reynolds number. The existence of several different components and several heat sources together with the vortices created by the fans make the flow regime turbulent inside the system
- **Boundary conditions:** The boundaries of the domain should be defined such that they should not effect on simulated smoke movement. Any object which may affect significantly on air flows or fire generated flow and smoke must be represented within the model.

FIRE DYNAMIC SIMULATOR (FDS)

FDS is a command-line software. Its application has been written in Fortran 90 without any graphical user interface, now it is available in compiled versions format for Windows as well as Linux operating systems. The modelling in FDS require an input file for every fire scenario in a text format to represent the geometry of building, computational scope, boundary conditions, grid resolution, materials specification, design fire vent, energy source parameters, mechanical or ductless ventilation systems, as well as the simulation output types. The computational domain in FDS is user defined and generally represents the complete physical bounding box enclosing a region of interest. The computational domain in FDS is made up of set of rectangular meshes, each mesh is having its own 3D rectilinear grid system. The input file contains resolution of the grid, is also user specified, but the dimensions of greed should meet the criteria for the Poisson solver, all voids and solid obstructions are forced to conform to the numerical grids. Building enclosure elements, solid objects, orthogonal solid rectangular blocks or cuboids should be specified in FDS input file. For representing flow obstacles, such as doors and windows are displayed as holes which allowing fluid and particles to flow through it. Each solid obstruction block should be orthogonal to each other, an inclined wall or roof should be modelled in a stair stepping manner by ensuring the grid cells. For an obstruction set of input data, required parameters are the sextuplet of coordinates explaining the lower and upper bound of the rectangular blocks, surface type and material the voids are specified with similar parameters to the obstruction group. Any solid obstruction grid within the enclosed region specified by the void sextuplet are eliminated.

The creation of the FDS input file could be a part of the fireplace modelling method and needs varied degrees of manual input and redaction significantly once considering multiple fire situations. Generally, the foremost time consuming half of the input data creation is that the transfer of building geometry data from paper or CAD drawings to the format needed by FDS. At the foremost laborious level, all the coordinates for the obstruction blocks representing the building geometry are manually determined by measurements and calculations from written drawings. This process is especially inefficient and error prone. There are unit package tools presently offered to help with the creation of the FDS input data, significantly with respect to the transfer of the building geometry and topology information. But these tools need the reconstruction of the building in one type or another, using data derived from written plans or CAD files.

PyroSim is a graphical user interface for the FDS. PyroSim has been developed to construct, browse and edit FDS input files. To help with the development of the 3D building model kind of like the obstruction blocks in FDS, 2D image of the building arrange will be overlaid on the graphics editor screen to permit three-dimensional wall elements to be manually positioned by tracing over the lines. Stair stepping is manually applied to diagonal or curve walls to evolve to the numerical grid system. There are vital edges in sharing a standardized digital illustration of buildings during which common information will be mapped across varied formats and for various input needs of fireplace simulation tools.

Chapter 3

COMPARITIVE RESEARCH ON FLUENT AND FDS

The stage1 simulation summarizes the method for simulation of fire smoke in underground car parking garage. By using two simulation software, i.e. FLUENT and FDS. Comparison made between their mathematical model, simulation software field and simulation technology theory. The CFD unsteady analysis of car parking garage was carried out using FLUENT as well as FDS. The domain consist of single car with engine fire , Jet fan, air inlet and opening for air exhaust.Aim of this investigation was to perform comparative analysis in two simulation software FLUENT and FDS.

3.1 COMPARISION BETWEEN MATHEMATICAL MODELS

As per software simulation field, there are six parameters used to describe flow field, and they are given as velocity component along coordinate directions: u, v, w. parameter for temperature field T, concentration of smoke contaminant C and pressure of flow field P. The flow of fire smoke follows the law of conservation of mass, consist of energy equation, momentum equation, continuity equation.

1) Energy equation

$$\frac{\partial \ell}{\partial t}(\ell c_p T) + \frac{\partial}{\partial x_t}(\ell v_i c_p T) = \frac{\partial}{\partial x_t}(\lambda \frac{\partial T}{\partial x_t}) + q_s - q_r \tag{1}$$

In this equation C_p is gas specific heat capacity at constant pressure, source term q_r, q_s, λ is gas thermal conductivity

2) Continuity equation:

$$\frac{\partial \ell}{\partial t} + \frac{\partial}{\partial x_i}(\partial v_i) = 0 \tag{2}$$

3) Momentum equation:

$$(\ell v_i v_j) - \frac{\partial \ell}{\partial x_i} + \frac{\partial}{\partial x_i}(\ell v_i c_p T) = \left[\mu(\frac{\partial v_i}{\partial v_j}) + (\frac{\partial v_j}{\partial x_i}) \right] - \frac{2}{3}\frac{\partial}{\partial x_i}(\mu(\frac{\partial v_i}{\partial x_i}) + \ell g_j \tag{3}$$

4) Mass equation:

$$\frac{\partial}{\partial t}(\ell c_s) + \frac{\partial}{\partial x_i}(\ell v_i c_s) = \frac{\partial}{\partial x_i}(D\ell \frac{\partial c_s}{\partial x_i}) - W_s \tag{4}$$

In this equation C_s expressed as gas mass fraction, D expressed as gas diffusion coefficient, W_s for rate of gas in combustion

The equation for model used in FLUENT software .

The FLUENT has broad physical models, this project uses standard K-є model two equation turbulence model, which is used in wide range of engineering application, and make 3-D numerical simulation of car park space under fire condition. During the simulation COUPLED algorithm used to solve Reynolds time averaged Naviour stroke equation. Partial differential equation used for field simulation to control smoke density can written as follows:

$$\frac{\partial(\ell\phi)}{\partial t} + div(\ell\mu\phi) = div(\Gamma grad\phi) + s_{\phi} \tag{5}$$

In this equation four time based item spread item, convection item and source item. ϕ is generic variable, Γ is diffusion coefficient.

The equation for model used in FDS

In FDS software, this project uses large eddy simulation (LES) numerical simulation method to solve low speed, heat and mass driven flow navier stroke equation. It is based on smoke and heat calculation in fire condition, mainly focused on turbulent mixing of combustion gases with surrounding air, and its main aim is to make gas vertex of mixing fluid to provide an accurate calculation results for CFD equation. LES technology not only avoid the poor accuracy brought by average of the flow field, but also decreases the enormous computational simulation even for relatively coarse mesh. It divides the building geometry into multiple small grids, solves each conservation equation by numerical method, and shows accurate results of fire simulation, such as fire pressure, temperature, velocity at any location in computational domain and flow of smoke. Basic equation of fire dynamics is the low Mach number flow equation, is shown as follows:

Mass conservation equation:

$$\frac{\sigma\ell}{\sigma t} + \nabla.\ell\mu = 0 \tag{6}$$

Momentum conservation equation

$$\ell\frac{\sigma\mu}{\sigma t} + (\mu.\nabla)\mu + \nabla p - \ell g = f + \nabla.\tau \tag{7}$$

Energy conservation equation:

$$\frac{\partial}{\partial t}(\ell h) + \nabla.\ell hu = \frac{Dp}{D_t} + q + \nabla.K\nabla T + \nabla.\sum_i ph_i D_t \nabla.Y_i - \nabla q_r \tag{8}$$

Gas state equation:

$$P_0(t) = vTR + \sum_i \frac{Y_t}{M_t} \qquad (9)$$

In these equation ρ as density , μ as viscosity , p as pressure ,h as enthalpy of flow field T temperature, Y as mass concentration , D is diffusion coefficient ,W chemical reaction speed ,M is molecular weight , Ʈ is viscous stress tensor ,R universal gas constant, q heat released by fire ,q_r flux equation of thermal radiation. q is heat released by fluid combustion.

3.2 COMPARISION IN SOFTWARE SIMULATION FIELD

FLUENT software serves abundant physical models, such as ideal gas model, combustion model, actual gas model, physical parameter, heat transfer model, rotation system model and specific boundary conditions of internal and external fluid flow. In addition to this, FLUENT software consist of eight widely used turbulence models in engineering and each model contains some sub models. There is no software other than FLUENT software can serves such abundant physical models [6]. Because of broad physical models, it is frequently used because of its strong simulation capability increases its application in the field of aerodynamic noise, rotating machinery, and multiphase flow system, combustion engine.

FDS software, founded by fire prevention lab of America, is a fire simulation software based on field simulation. This software is the unique fire simulation software developed by national institute of standards and technology in 2000. It has been examined by many practical examples so that it is used in fire simulation. As compared with other simulation software, FDS is more pertinent. With continuous development, FDS carried out a lot of fire simulation problems in fire control engineering simultaneously it provides a tool to study of fire dynamics.

3.3 COMPARISION OF FIELD SIMULATION TECHNOLOGY

Field simulation is a highly advanced but more complex method, which requires mass calculation. Computational Fluid Dynamics (CFD) is a science, which consist of fluid flow, heat and mass transfer, chemical reactions. FLUENT software based finite volume method, which divides given geometry into a series of control volume. Each control volume has a node as representor. Finding the discrete equation by maintaining the volume control equation for the integration. During the process of finding, the function and first derivative of the interface has to be assumed, and this method of finding is finite volume discrete method. Finite volume method serves three numerical algorithms, while other CFD software can provide only one of them, algorithms are Segregated Solver, Coupled Implicit Solver and Coupled Explicit Solver .Grid generator in FLUENT is GAMBIT, has enormous unstructured grid generation capability.

FDS software is developed to simulate fire environment under most adverse credible design fire. It serves two numerical simulation methods, such as DNS and LES, which can be used in 3d simulation of fire condition. This model adopts finite element method concept and has been tested by large-scale fire experiment.

3.4 COMPARISION OF OTHER ASPECTS

FLUENT has powerful post processing to fulfill requirements of CFD calculation, along with velocity diagram, flow path chart, integrating function to work out force, contour map, contour surface map, torque. Its function includes parameter user cared and calculation of error can be tracked and shown dynamically. Excluding to untreated output data, FDS model serves multiple chart outputs, which used to observe data intuitively. Boundary file, thermocouple, contour surface, and section file are all used for this reason. The graphical display of output data is named as Smoke View, which is used to show output data of FDS simulation. Section file is nothing but colored slice of full control volume. Through this section of slice, users can observe temperature distribution, velocity distribution, and visibility of gas and can observe its variation with the change of time. According to this research section file is used to estimate the temperature distribution at middle of fire location, velocity distribution of air, and visibility.

3.5 COMPARISION IN NUMERICAL SIMULATION

CFD Domain

The starting point for the use of CFD to the simulation of air flow, fire and smoke movement in a car park garage is to create the computational domain for the simulation. The primary considerations are summarized below.

- 3-D Domain.

  Even for simplest geometries, the air flow and smoke flows are three-dimensional.

- Boundaries of the Domain.

  The boundaries of the domain are a function of the CFD simulation. They should enclose the region of interest. The size of the domain will be fully affected by the nature of the car parking garage. The boundaries of the domain for enclosed underground car park, impulse ventilation containing basement car park will be formed by the inlet, outlet, Jet fan, car fire.

- Additional Factors influencing on domain selection.

  - Computational Limits.

    In CFD simulation consider all the possible influencing parameters on air flows, fire generated flow in a car park may higher for the computational resources, like processing power, memory and time available. In order to avoid all such problems, it is necessary to focus on the key parameter affecting the air flow. Meanwhile need to ensure that the influence of the subtracted factors should not compromise the objective of the CFD simulation.

  - Attached Volumes.

    The volume domain influenced by, other volumes, like floors of a multi-storey car park which are not connected to the fire floor. Like in this type of case it will not be possible to model the entire car park. The boundaries of the domain should be defined such that the flows between the attached volumes will be considered to be minimal.

3.5.1 GEOMETRY

The dimensions of the constructed car park are L x W x H = 15m x 10m x 2.5m.The fire is located in the middle of the car park and has a surface A= 2 x 1 m_2, as shown in Figure 6 & Figure 7

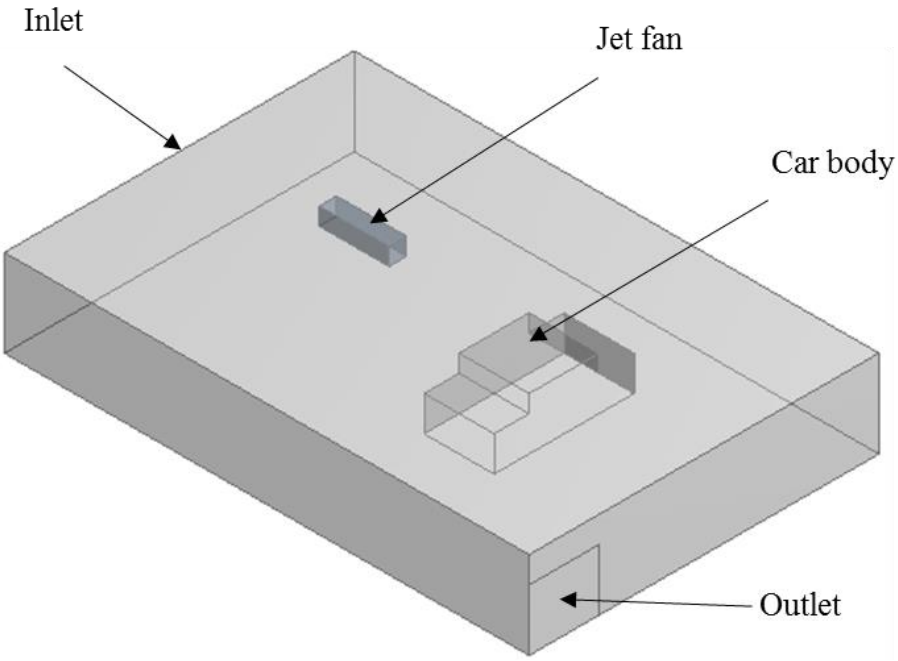


Figure 6 FLUENT geometric model

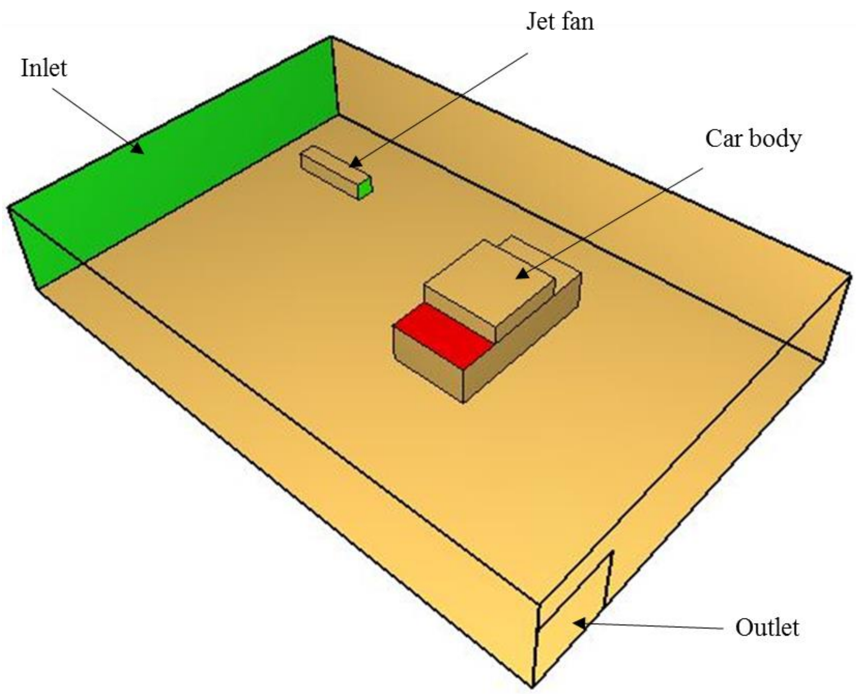


Figure 7 FDS geometric model

3.5.2 MESH GENERATION

Meshing was done separately using ANSYS 14.5 and FDS. Both meshing shows common number of 46474 elements consisting of quad mesh. Meshing model using both software shown in figure 8 & figure 9.

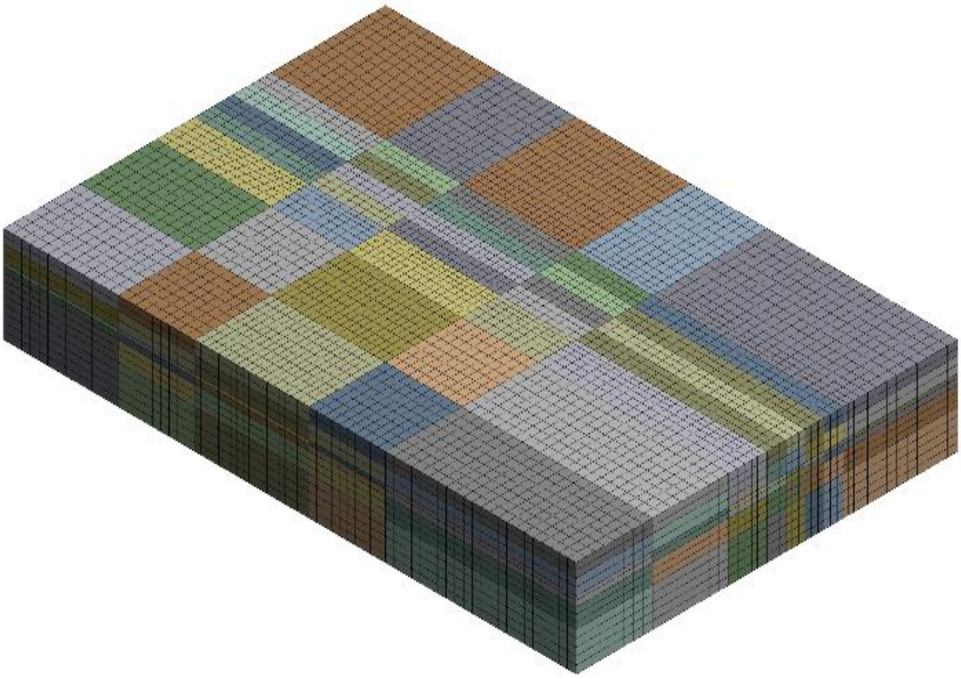


Figure 8 FLUENT mesh model

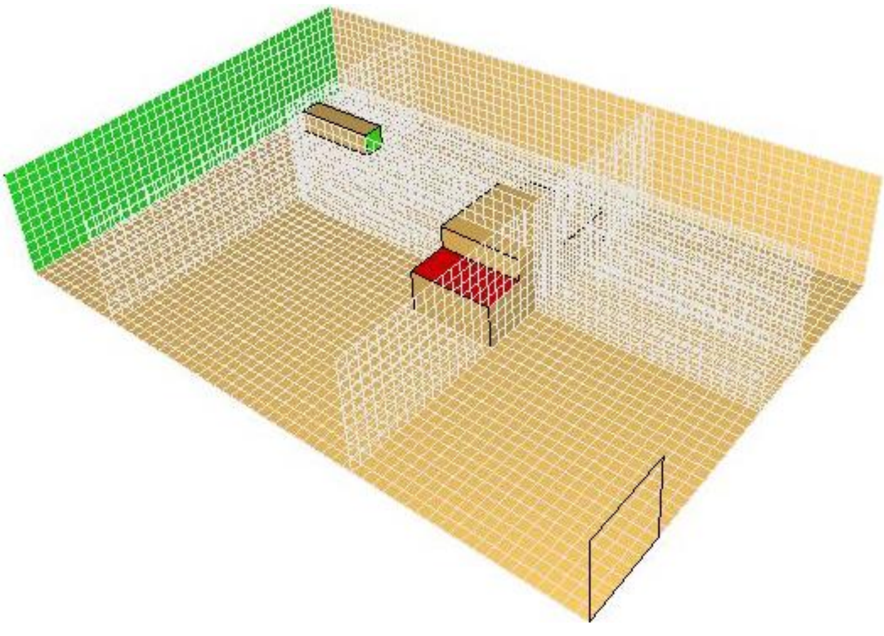


Figure 9 FDS mesh model

3.5.3 FLUID PROPERTIES:

The details of fluid properties is shown in Table 1.

Table 1 Fluid Properties

Fluid	**Density (kg/m^3)**	**Dynamic viscosity (Pa.s)**
Air	1.225	1.7894e^{-05}
CO_2	1.98	1.37e^{-05}
N	1.2506	1.663e^{-05}

3.5.4 BOUNDARY CONDITION

- The fluid medium was multi-species Mixture of N_2, O_2, CO_2 and H_2O
- Unsteady CFD analysis was carried out
- Assuming the fire has reached a stable developed stage
- No radiation considered in analysis
- Fluent uses K-ϵ model, FDS uses LES model
- 0.1 kg/s combustion gases (H_2O and CO_2) at 1200 K are emitted from car body
- 80 N/m^3 momentum source in jet-fan
- Velocity inlet boundary conditions were applied at the inlet. The velocity was specified as 0.8 m/s.
- Pressure outlet boundary conditions were applied at the outlet of the garage

Chapter 4

DESIGN OPTIMIZATION OF VENTILATION SYSTEM IN CAR PARKING GARAGE

4.1 PARAMETERS AND THEIR LEVELS

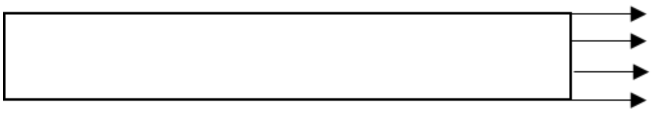
For design optimistion of ventilation system, different factors were considered 3 factors with 2 levels each were finally chosen. The responses analysed were clearance time required to remove smoke. Table 2 depicts the different parameters with their levels.

Table 2 Parameters and their levels

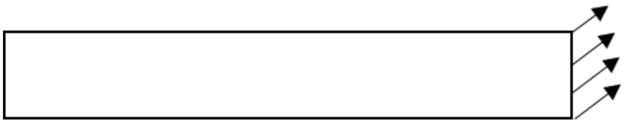
Parameters			**Levels**	
		Units	**1**	**2**
A	Angle of Jet Fan	Degree	0	45
B	Number of Jet Fan	Nos	2	3
C	Type Exhaust Fan (Total capacity= 60 m^3/s)	m^3/s	60 x 1 (Single exhaust)	30 x 2 (double exhaust)

4.1.1 ANGLE OF JET FAN

Generally discharge angle of jet fan is always normal to direction of flow. But in this study of optimization we used discharge of jet fan louvered by an angle of 45° for channelize the smoke flow for effective clearance.



0° Angle of discharge



45° Angle of discharge

4.1.2 NUMBER OF JET FAN

Increment in number of jet fan will help to increase circulation of smoke flow. In this study we used maximum three number of jet fans.

Table 3 Jet Fan Specifications

Jet fan Type	Air flow (m^3)	Air speed (m/s)	Thrust N
CC-JD 301	1.28	17.3	27

4.1.3 TYPE OF EXHAUST

Type of exhaust consist of single and double exhaust, to check the effect of exhaust fan size on the clearance time.

Table 4 Exhaust Fan Specification

Type of Exhaust	Model	Area (ft^2)	Capacity (CFM)
Single Exhaust	TCVX 42D4	9.793	62793
Double Exhaust	TCVX 60D5	19.87	126812

4.2 FIXED DATA

After the parameters and their levels are selected, the other design parameters were fixed. This data were always kept constant and only the parameter levels were varied in the experimental runs.

- Enclosure:

  Width = 14.5m

  Depth = 15 m
- Stationary Car

  Length = 4.5 m

  Width = 1.75 m
- Inlet vent = W x H = 4 m x 2 m
- Size of Jet fan = L x W x H = 1.6 m x 0.4 m x 0.4 m (4m from inlet vent)

4.3 CFD MODELS FOR DESIGN OPTIMIZATION

The dimension of constructed car park in optimization is 15x14.5x2.4 m with open inlet of size 4x2 m, based on scaled model (Yuan Jian-ping, 2011) .The position of fire located at car bonnet and has surface 2 m^2. The fire has peak heat release rate of 4 MW in beginning and stays constant for next 120s.The details of geometry as shown in figure 10 to figure 13.

4.3.1 GEOMETRY

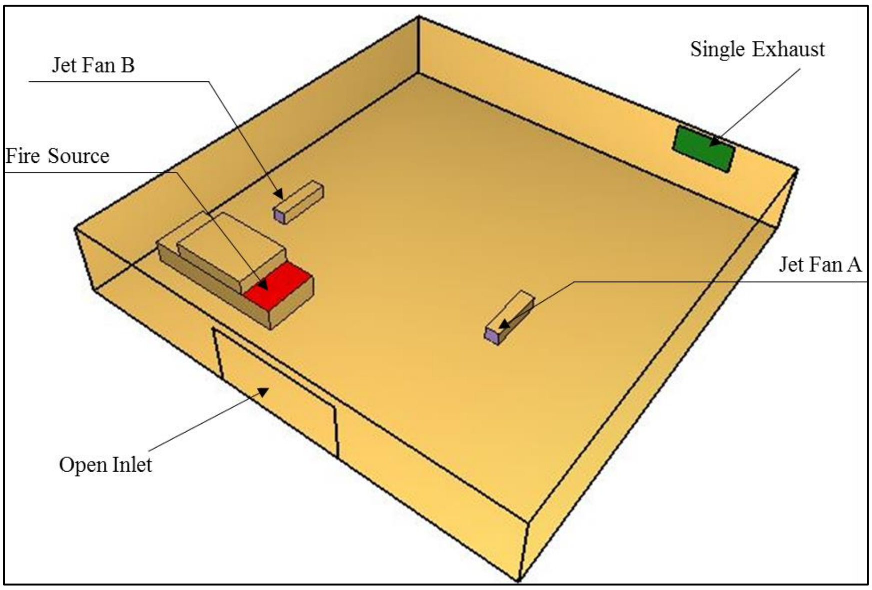


Figure 10 Test Run 1

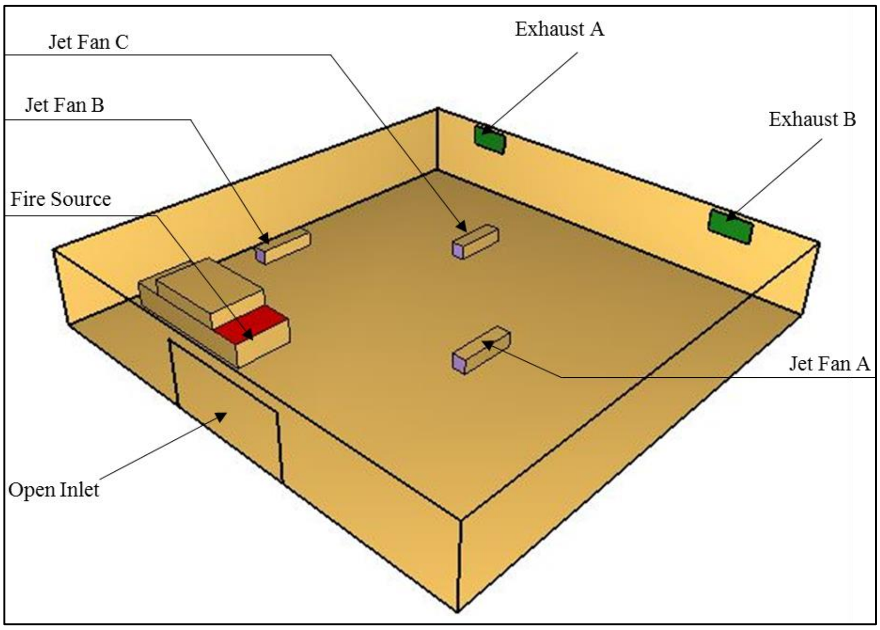


Figure 11 Test Run 2

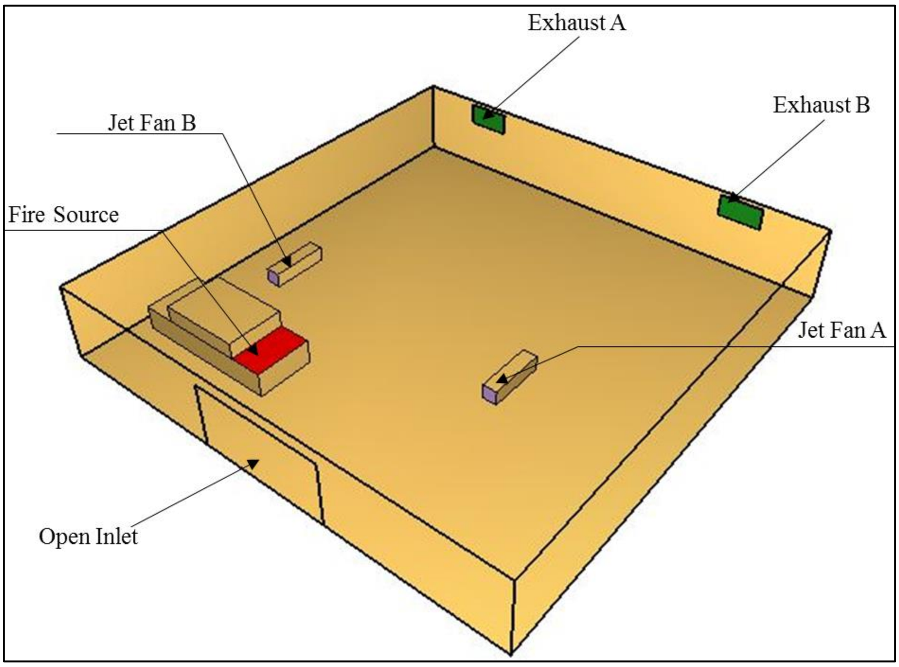


Figure 12 Test Run 3

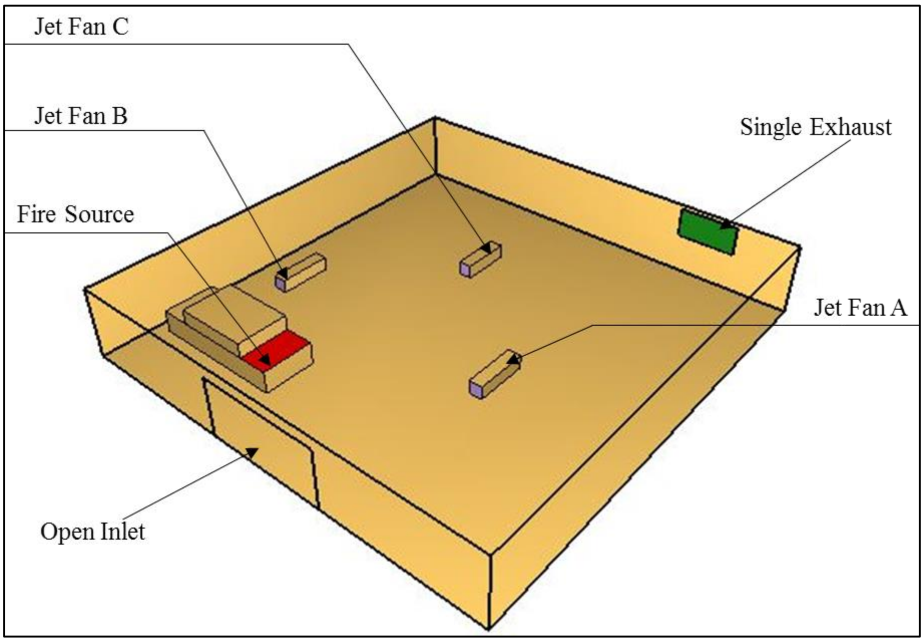


Figure 13 Test Run 4

4.3.2 BOUNDARY CONDITION

Boundary Condition	Value
Software used	FDS
Fluid	Mixture of N_2, O_2, CO_2 and H_2O
Turbulence model	LES
Inlet Condition	Open
Fire HRR (MW)	4
Jet Fan Flow rate (m^3/s)	1.28
Extraction Fan (CFM)	Single Exhaust = 62793 Double Exhaust = 126812

4.4 TAGUCHI METHOD

Taguchi methods are statistical methods are used to improve the quality of manufactured goods, and more recently also applied to, engineering, biotechnology, marketing and advertising. Professional statisticians have welcomed the goals and improvements brought about by Taguchi methods, particularly by Taguchi's development of designs for studying variation, but have criticized the inefficiency of some of Taguchi's proposals. Taguchi addresses quality in two main areas: off-line and on-line quality control (Roy, 2001) the primary goals of Taguchi methodology can be described as:

- A reduction in the variation of a product or process design to improve quality and lower the loss Imparted to society.
- A proper product or process implementation strategy which can further reduce the level of variation

4.4.1 ORTHOGONAL ARRAYS

Orthogonal array testing is a systematic, statistical way of testing. Orthogonal arrays can be applied in user interface testing, system testing, regression testing, configuration testing and performance testing. The permutations of factor levels comprising a single treatment are so chosen that their responses are uncorrelated & hence each treatment gives unique piece of information. Net effect of organizing the experiment in such treatments is that the same piece of information is gathered in minimum number of tests. All orthogonal vectors exhibit orthogonality (Douglas, 2007). Orthogonal vectors exhibit the following properties:

- Each of the vectors conveys information different from that of any other vector in the sequence, i.e., each vector conveys unique information therefore avoiding redundancy.
- On a linear addition, the signals may be separated easily.
- Each of the vectors is statistically independent of the others, i.e. the correlation between them is nil.
- When linearly added, the resultant is the arithmetic sum of the individual components.

4.4.2 BENEFITS OF TAGUCHI METHODS

Following are the benefits that are specially associated with the orthogonal array:

- Provides uniformly distributed coverage of the test domain.
- Concise test set with fewer test cases is created.
- All pair-wise combinations of test set created.
- Arrives at complex combinations of all the variables.
- Simpler to generate and less error prone than test sets created manually.
- Reduces testing cycle time.

4.4.3 LIMITATIONS OF TAGUCHI METHOD

Some of the limitation associated with Taguchi method are:

- In Taguchi's arrays, interactions are confounded and difficult to resolve.
- Statisticians in response surface methodology (RSM) advocate the "sequential assembly" of designs. In the RSM approach, a screening design is followed by a "follow-up design" that resolves only the confounded interactions judged worth resolution. A second follow-up design may be added (time and resources allowing) to explore possible high-order univariate effects of the remaining variables, as high order univariate effects are less likely in variables already eliminated for having no linear effect. With the economy of screening designs and the flexibility of follow-up designs, sequential designs have great statistical efficiency.
- It does not guarantee the extensive coverage of test domain

4.4.4 USES OF TAGUCHI METHOD

- Taguchi method has following major uses:
- Prediction always has some degree of uncertainty.
- Should have reasonable prediction throughout the experiment range.
- To determine factor levels that will simultaneously satisfy a set of desired specification.
- To determine the qualitative understanding of behavior over the region tested.
- To predict the product properties throughout the region.

4.4.5 L4 ORTHOGONAL ARRAY

When the parameters to be evaluated are more the number of experiment to be carried out increases. Taguchi method is a highly preferred technique that uses a special design of orthogonal arrays to understand the effects of parameters with least number of experiments. Taguchi method contains parameter design, system design and tolerance design procedure to achieve a robust design to ensure best product quality. Taguchi method uses the S/N (signal-to-noise ratio) to analyze the experimental data and find the optimal parameter configuration. Further ANOVA (analysis of variance) and regression analysis can be implemented to estimate error variance, contribution of parameters and a response equation respectively. The Taguchi array is a balanced array of parameters and levels such that the individual parameter values are confounded. The experiments were conducted for two level, L4 orthogonal array. The orthogonal array is shown in Table 5.

Table 5 Taguchi design of experiment approach

Test Run.	**Factors**		
	Angle of Jet Fan	**Number of Jet Fan**	**Type of Exhaust**
1	0	2	Single Exhaust
2	0	3	Double Exhaust
3	45	2	Double Exhaust
4	45	3	Single Exhaust

Chapter 5

RESULTS AND DISCUSSIONS

The CFD results of this study is separated in two parts,

- Stage1 - Comparative study was analysed using two simulation software FLUENT and FDS.
- Stage 2 – Design optimization of ventilation system

In stage1 case CFD analysis was carried out using Ansys Fluent V14.5 and Pyrosim 15. The difference in numerical results of both software were carried out.

In stage 2 case optimisation in design of ventilation system for time require to clear the car parking space after 120s of fire was evaluated. Finally all the data of optimisation was statistically analysed using Minitab 17 to arrive at optimal parameter setting.

5.1 CFD RESULTS

5.1.1 SOURCES OF ERRORS IN CFD CALCULATIONS

There is always error in a CFD analysis. It is important to know the sources of these errors and take precautions accordingly. The major source of error for a CFD analysis is due to the selected numerical method to solve Navier-Stokes equations. Some of the numerical techniques employed in CFD are Finite Difference Methods, Finite Element Methods and Finite Volume Methods. The one FLUENT uses is the Finite Volume Method and the source of error here arises when discretizing the transport equations. Interpolations are made to find values at the cell faces, whereas all the information is stored at the cell centres. This is the main approximation of the Finite Volume Method.

Second type of error is at the Boundary Condition definitions. It is up to the user how to define the boundary conditions, therefore the results will be as correct as the user defines them. The physical models employed may also be a source of error. Choosing the right turbulence model, density calculation method or radiation calculations affect the results.

All iterative solvers should run long enough to minimize the numerical error. Solver can be terminated at any time but great attention must be taken for achieving converged results. Default convergence criteria or predefined tolerances do not always assure converged results. Even when the residuals fall below the convergence criteria, more iterations may be needed for the convergence. In order to understand when the results are converging, it is essential to open extra convergence monitors for some scalars in FLUENT.

One more important aspect to reduce the error in CFD calculations is to have a grid-independent solution. Grid must be fine enough to capture all flow features and analysis results must not change when the models are run with finer meshes. If the results are changing as the number of cells used are increased, then finer mesh should be created for grid independency.

For some big models, the mesh resolution at critical locations cannot be transformed to the whole domain. Then non-conformal interface can be used. This is the case where the high density grid in and around the capacitor is not spread to the whole computational domain. This introduces some error. The reason is that, single cell on one side of the non-conformal interface corresponds to several cells on the other side. So, an interpolation is done at the interface.

If the flow is turbulent then the error increases compared to the laminar case. Since we do not have enough computational resources to solve turbulence with Direct Numerical Simulation we have to model it. This modelling brings another type of error to the solution. There are turbulence models like Large Eddy Simulation, Detached Eddy Simulation or even Reynolds Stress Modelling, but they are far too expensive for our computational resources. Therefore we use Reynolds Averaged Navier Stokes equations to solve for the turbulence.

5.1.1.1 DISCRETISATION

The first source of error comes from the theory of finite volume method. Interpolations have to be done for discretization. There are numerous schemes for this and the easiest is the first order upwind. The advantage of this scheme is easy convergence. However, it is only first order accurate. It is suggested to use second order schemes for unstructured grids (Fluent user guide).

5.1.1.2 CONVERGENCE ISSUES

Only a well converged, well posed and grid independent simulation can give reliable results. Convergence is determined by the order of magnitude residuals drop. It should be noted that, convergence criteria must assure that the results do not change as the iterations proceed. There is a common way of implementing this. Scalar change of some values like temperature is displayed as well as the residual monitors. When the scalar values stay at a certain number and do not change as the iterations continue, then it can be stated that the solution is converged. It was seen that this trend is achieved when the continuity and momentum residuals fell below 10-4 and energy residual fell below 10-7 convergence criteria of Therefore all the models use the convergence criteria of 10-4 for the flow variables and 10-7 for the energy.

5.1.1.2 EXPERIMENTAL RESULTS

The 4 tests were conducted according to the DOE array. The responses considered in this study were time require to clear smoke from whole domain. The results of the tests are given below in Table 6.

Table 6 Results of tests

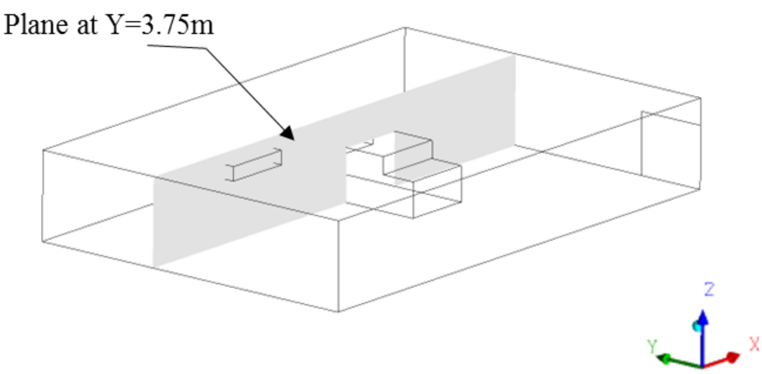
Test Run.	Factors			Clearance Time
	Angle of Jet Fan	Number of Jet Fan	Type of Exhaust	
1	0	2	Single Exhaust	271s
2	0	3	Double Exhaust	206s
3	45	2	Double Exhaust	176s
4	45	3	Single Exhaust	218s

5.1.2 CFD POST PROCESSIGN RESULTS

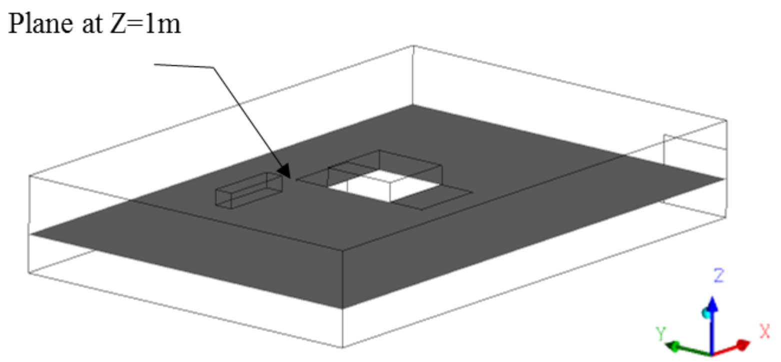
5.1.2.1 STAGE 1- COMPARITIVE RESULTS OF FLUENT & FDS

CFD NOMENCLATURE

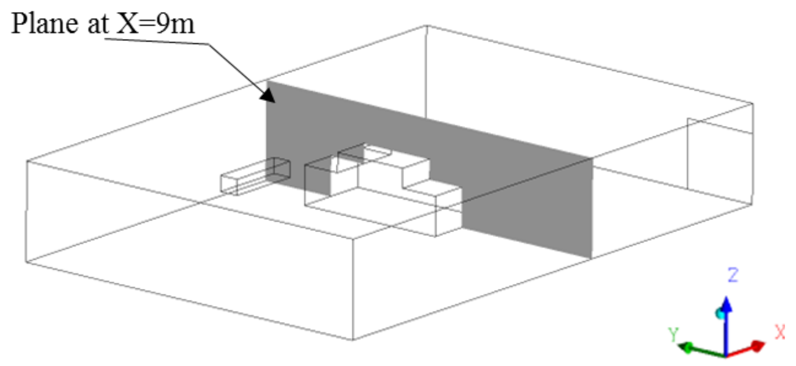
- Plane middle of jet fan body to plot velocity distribution of Jet fan



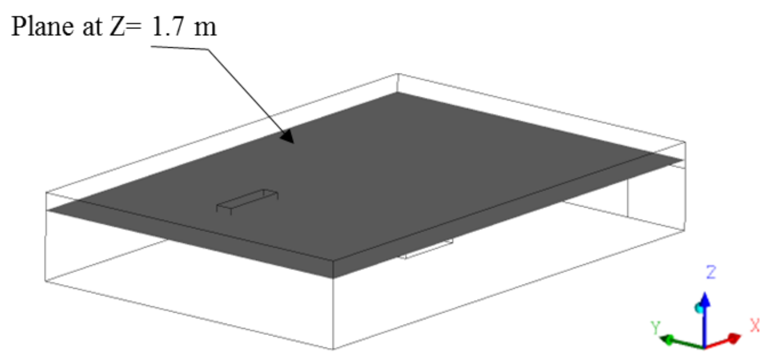
- Plane at the height of Z= 1 m in CFD domain to plot velocity distribution of air flow



- Plane at middle of car body to temperature distribution of fire



- Plane at Z=1.7 (As per NBC coding) to plot visibility contour



CFD RESULTS FOR STAGE 1

Both software uses same simulation time of 350s. The results of comparison for 350 s is shown from figure 14 to figure 19

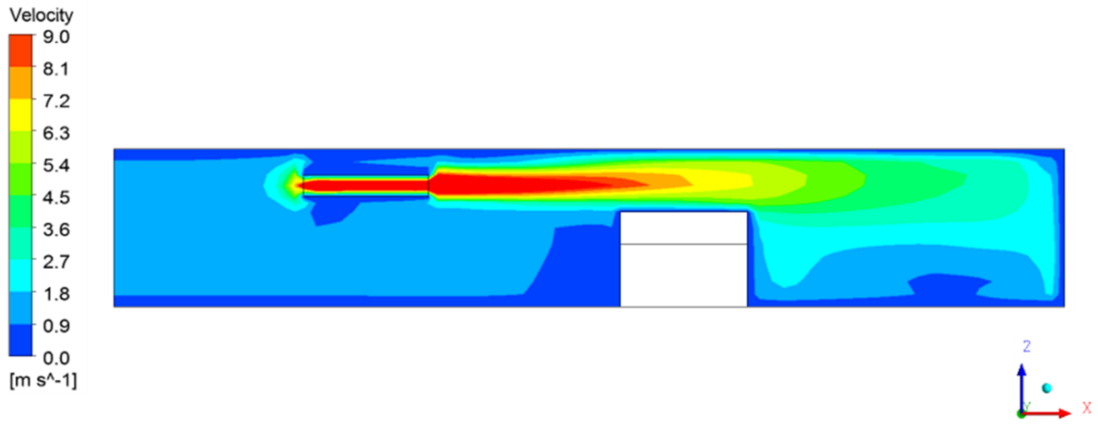


Figure 14 FLUENT velocity distribution at middle of jet fan at 350 s

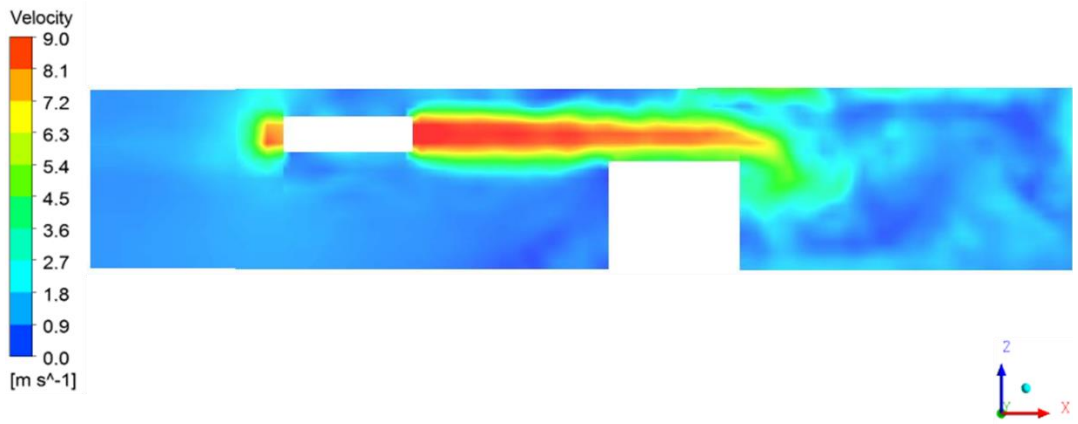


Figure 15 FDS velocity distribution at middle of jet fan at 350 s

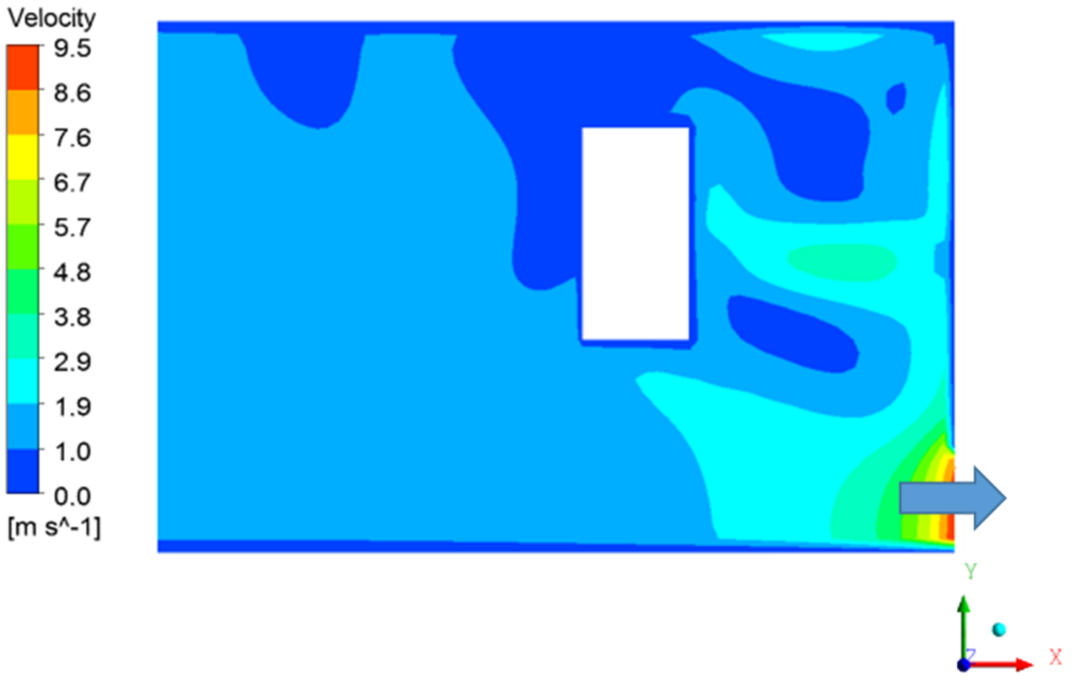


Figure 16 FLUENT velocity distribution at middle of height Z=1 m

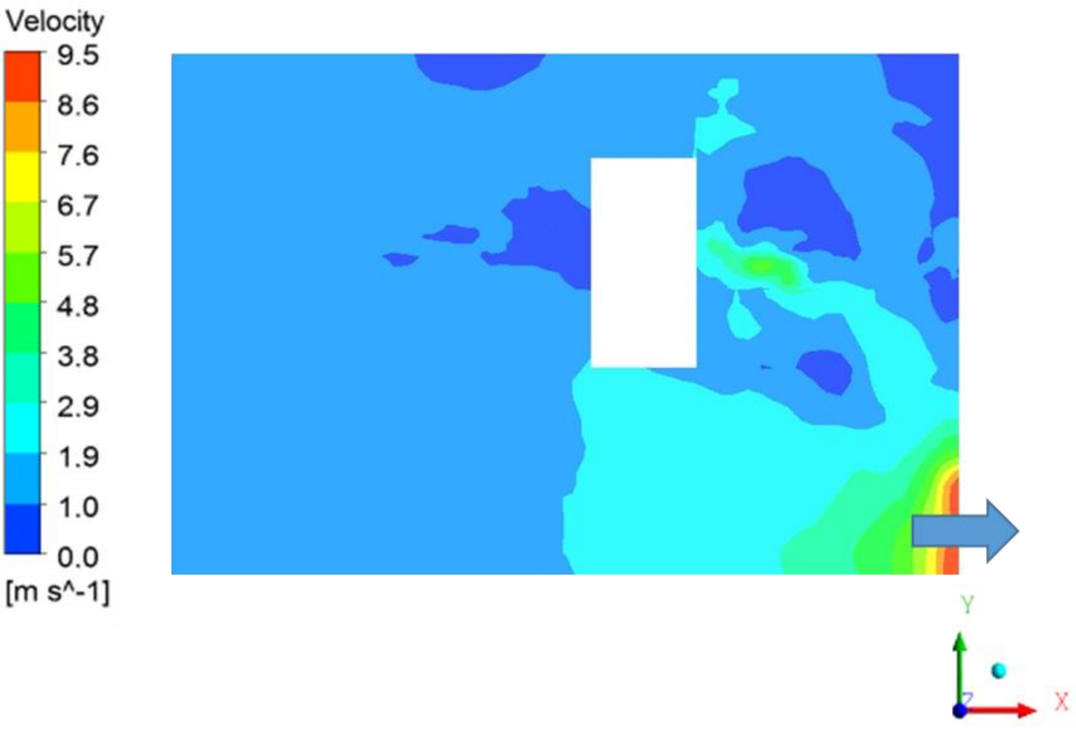


Figure 17 FDS velocity distribution at middle of height

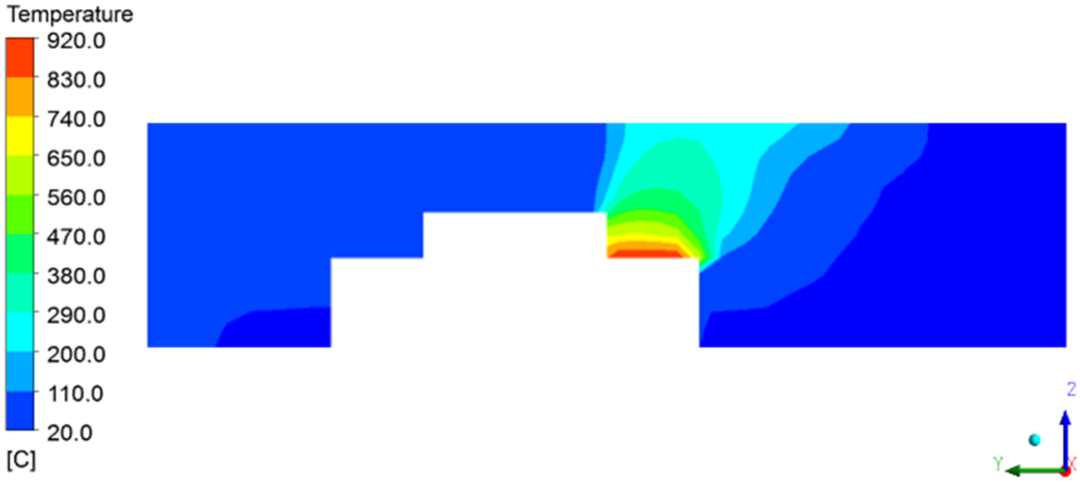


Figure 18 FLUENT temperature distribution at middle of fire

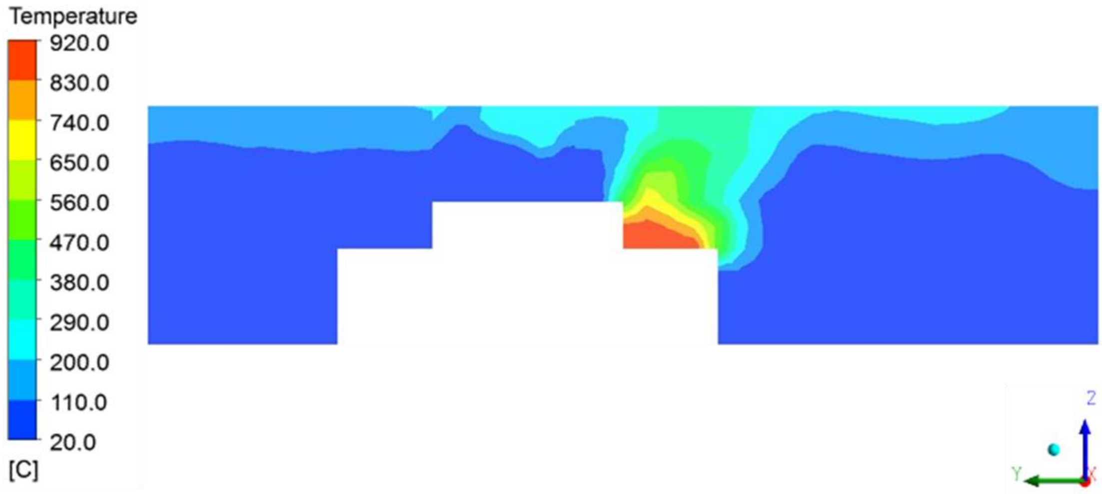


Figure 19 FDS temperature distribution at middle of fire location

As per the numerical results from figure 14 to figure 19 it shows that results of FDS are approximately similar to results of FLUENT at different planes. But FDS require less simulation time compared with FLUENT and it is cheaper as well as flexible for modelling of fire, hence for building simulation application FDS is more appropriate than FLUENT.

5.1.3 CFD RESULTS FOR STAGE 2

FDS used in simulation of fire scenarios using computational fluid dynamics (CFD). It can predict smoke, temperature, carbon monoxide, and other substances during fires. The results of optimisation runs are shown from Figure 20 to figure 31

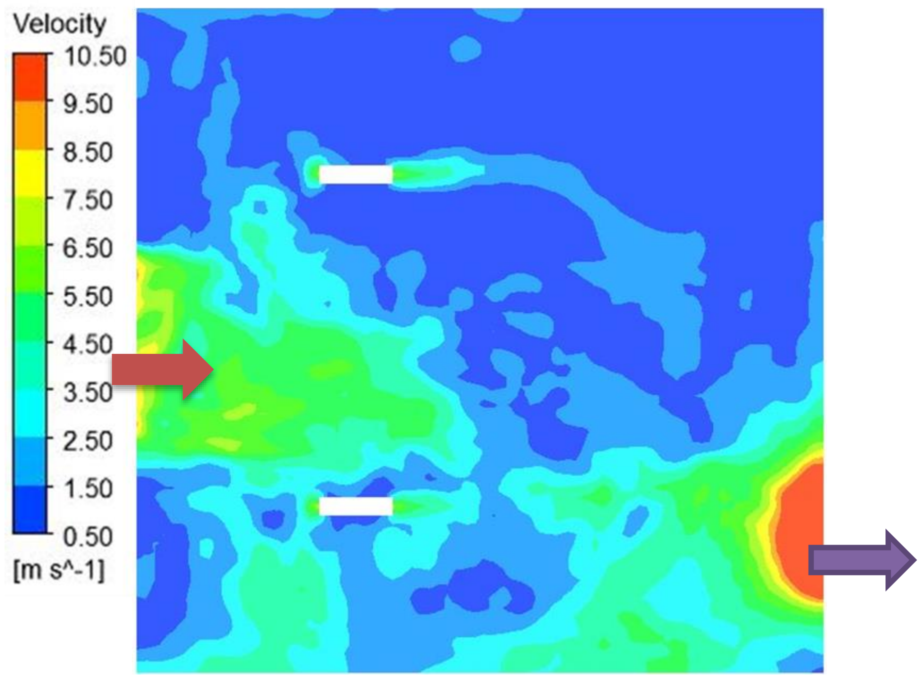


Figure 20 Velocity distribution in test run 1

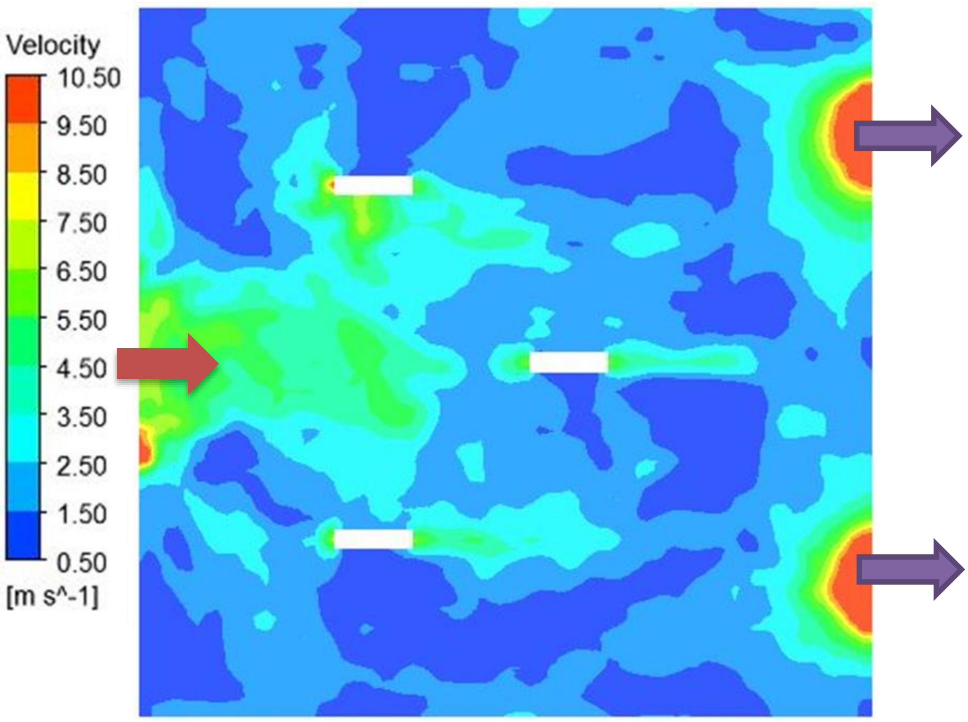


Figure 21 Velocity distribution in test run 2

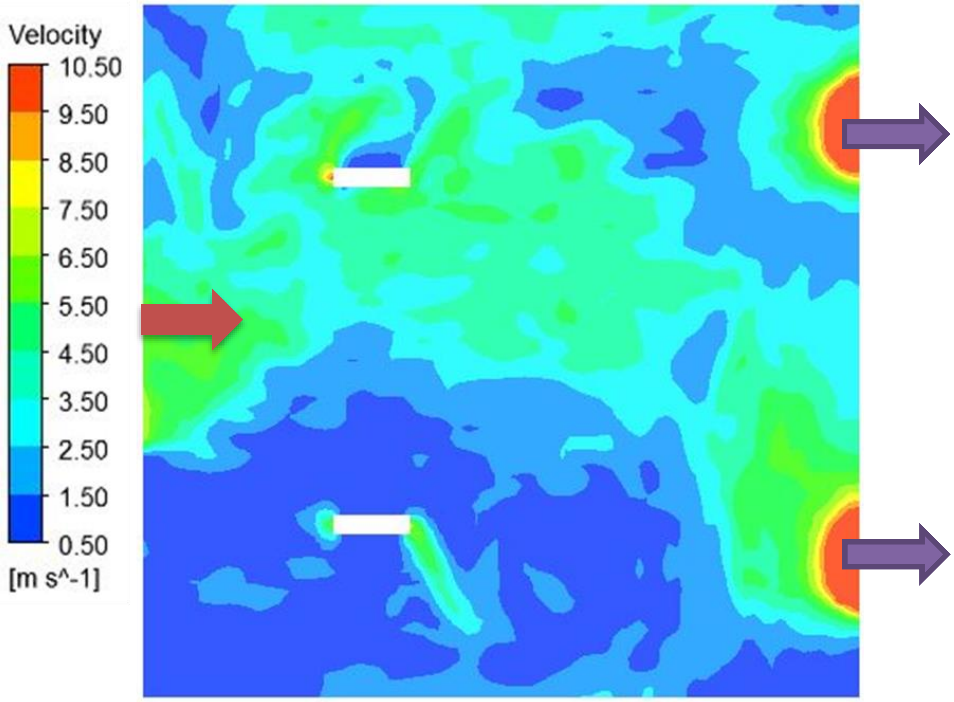


Figure 22 Velocity distribution in test run 3

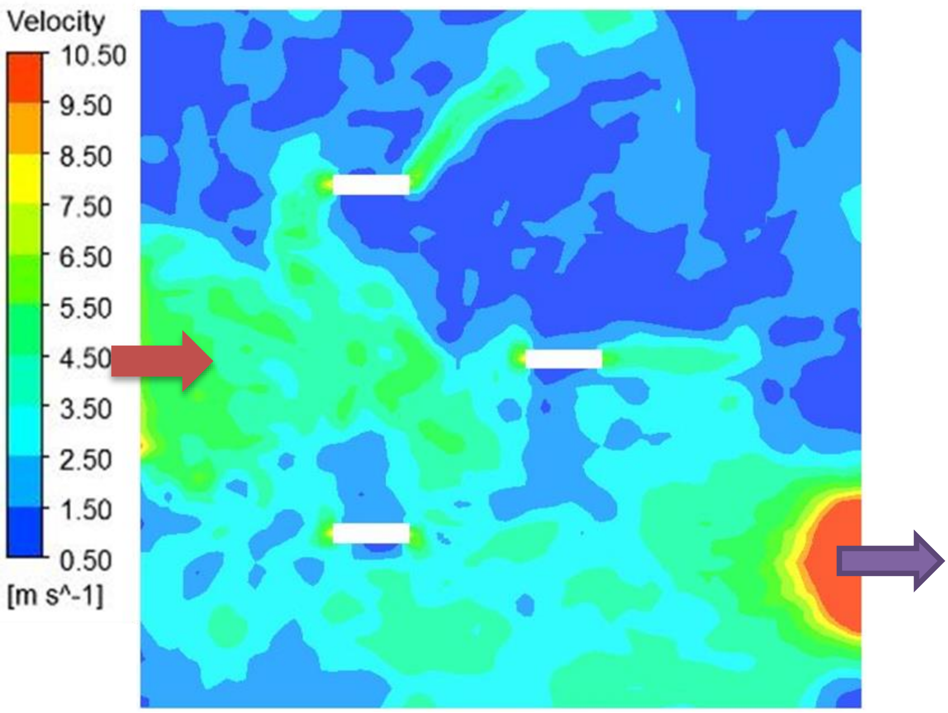


Figure 23 Velocity distribution in test run 4

From results of velocity distribution in all test runs shows that test run 3 has high air circulation than others.

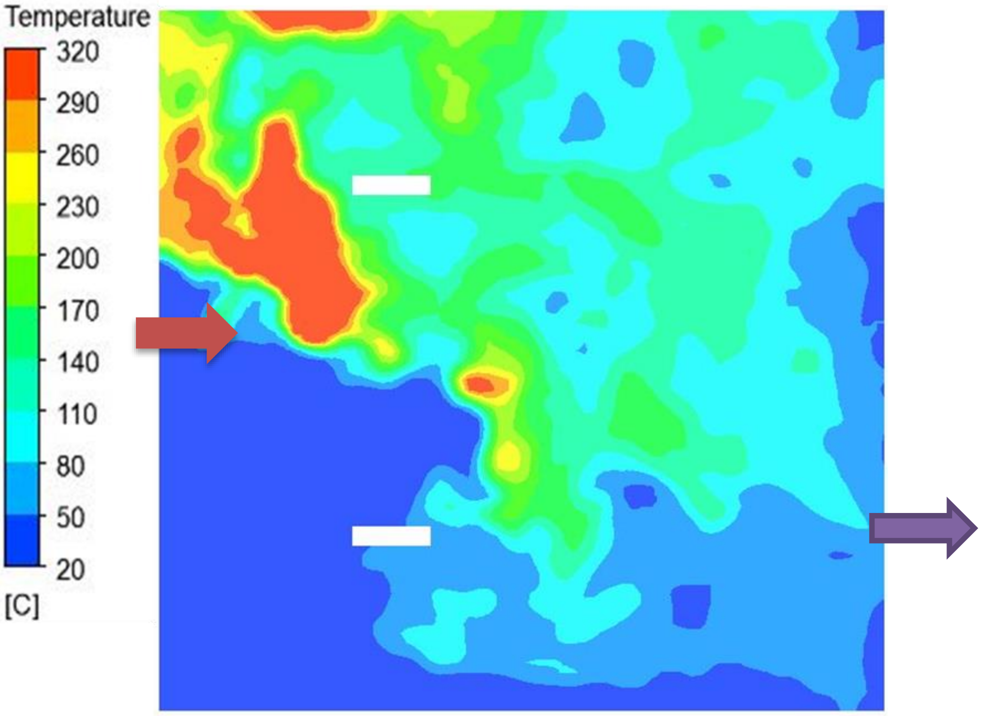


Figure 24 Temperature distribution in tests run 1

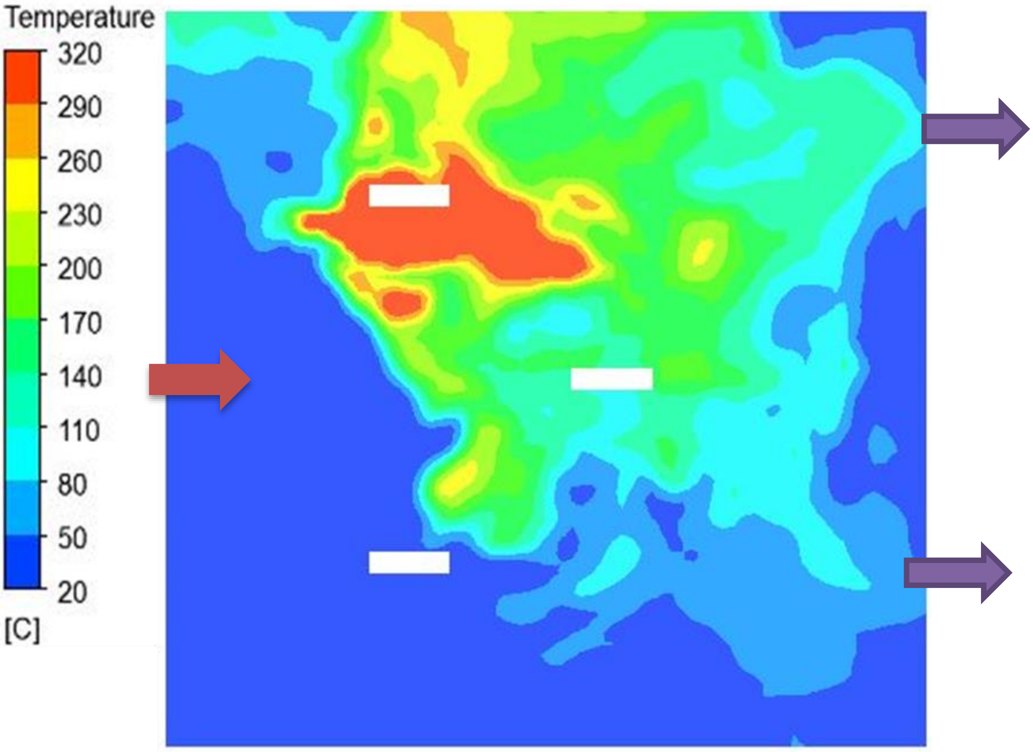


Figure 25 Temperature distribution in test run 2

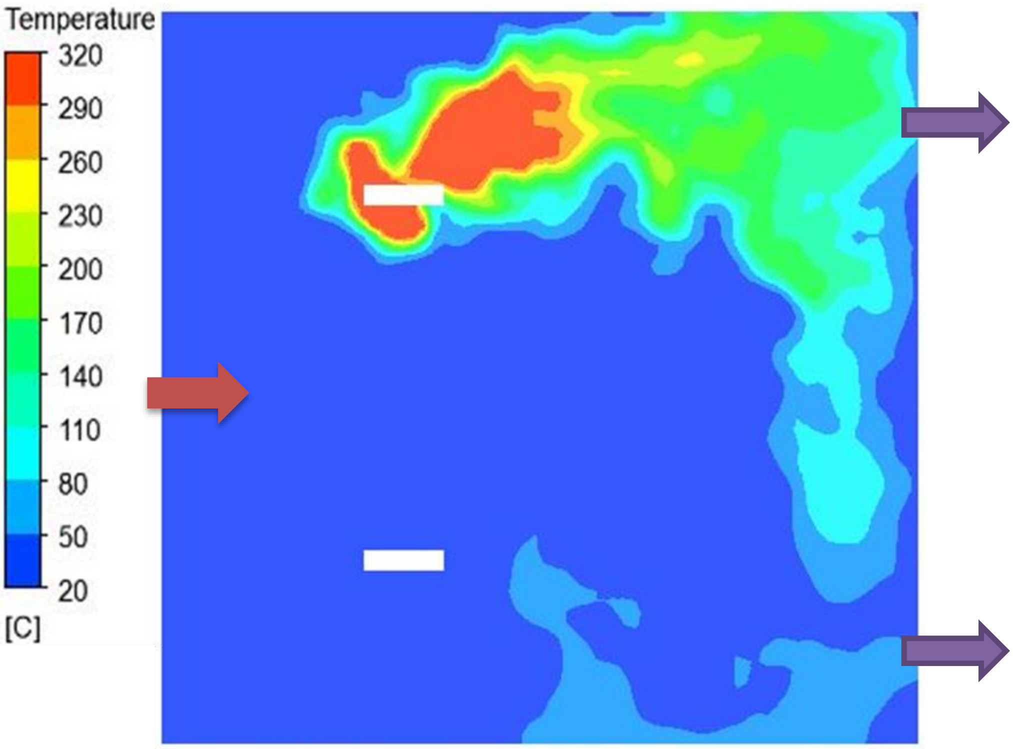


Figure 26 Temperature distribution in test run 3

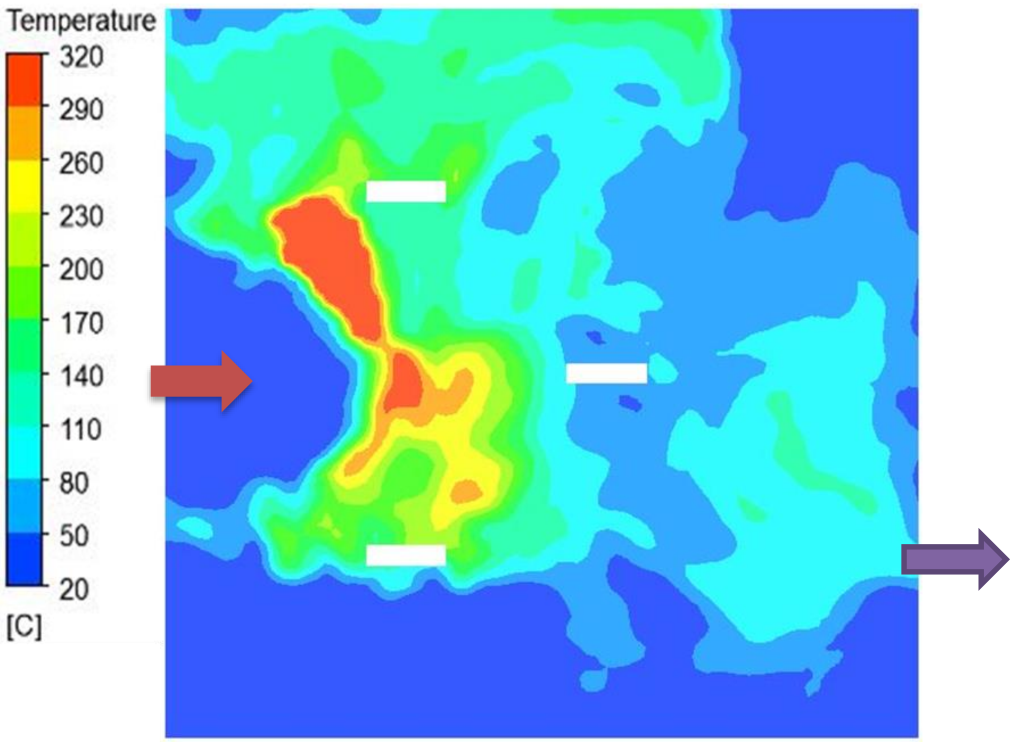


Figure 27 Temperature distribution in test run 4

Visibility plot used to show the clearance rate of smoke within specific time. The red colour used to represent clear view and blue colour represent smoke view at the specific time 120 s in each experiment. The detailed visibility plot is shown in figure

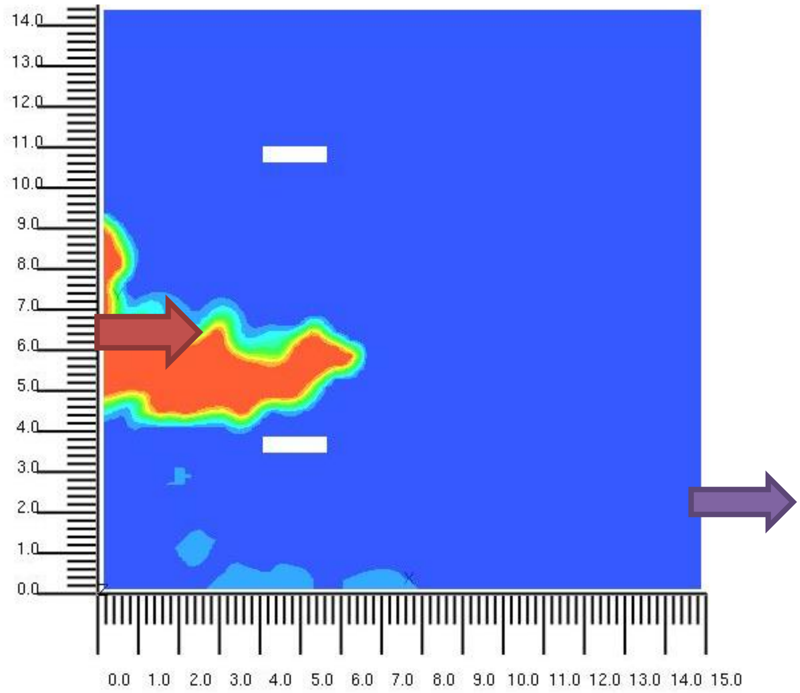


Figure 28 Visibility plot in test run 1

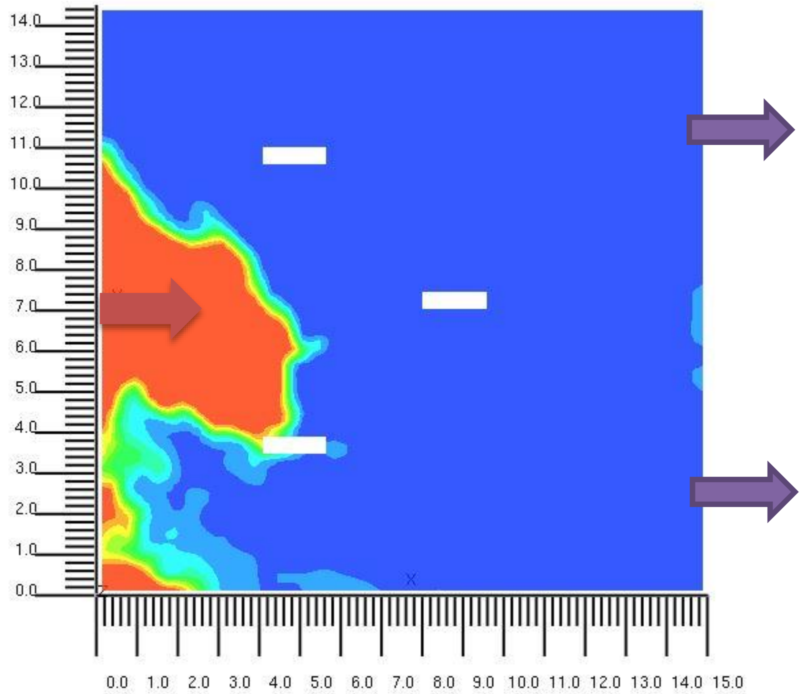


Figure 29   Visibility in test run 2

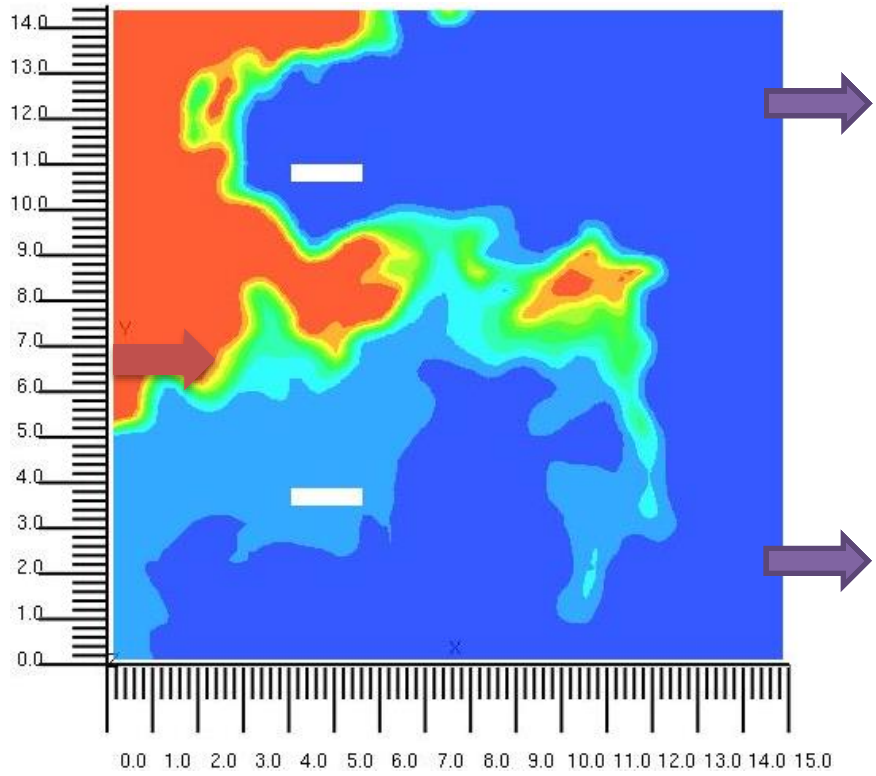


Figure 30 Visibility plot in test run 3

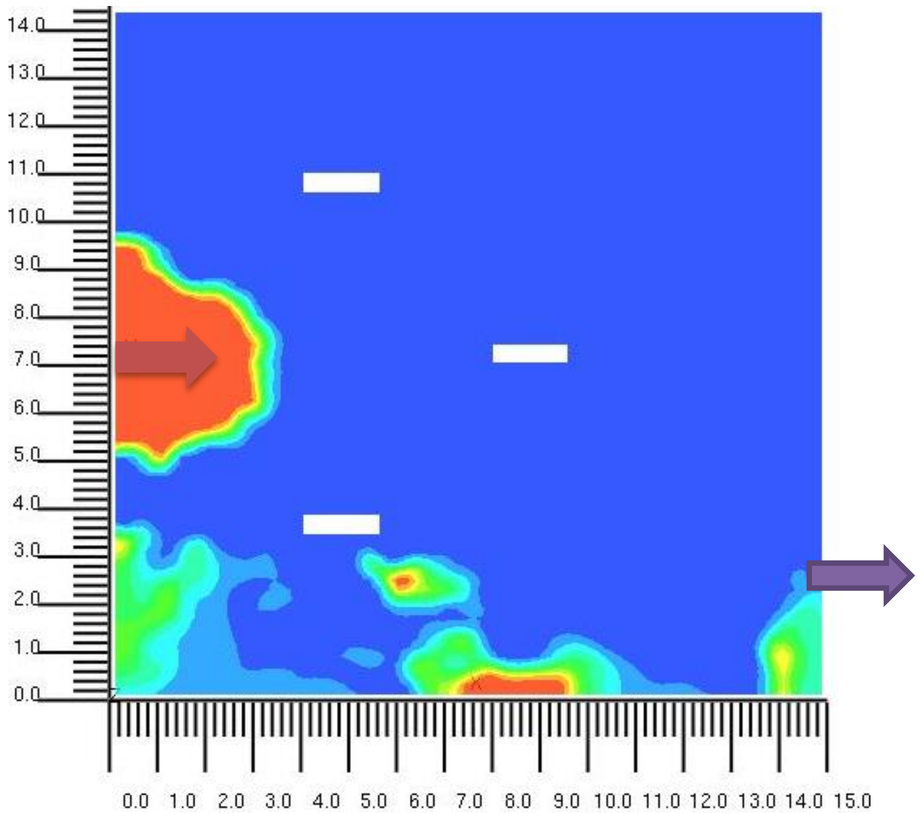


Figure 31 Visibility plot in test run 4

5.2 STATISTICAL ANALYSIS OF RESULTS

5.2.1 SIGNAL-TO-NOISE RATIO (S/N) RATIO

The Taguchi approach is one experimental design which has achieved a great deal of success. The overall aim of the Taguchi design is to find factor levels that maximize the S/N ratio. In statistical terms, "S" is called a "signal" and "N" is called a "noise". The higher the S/N ratio, the better the quality. S-N ratio is a measure to identify the parameter setting that minimizes the effect of noise on the response. There are three categories of quality characteristics in the analysis of the S/N ratio, i.e. the smaller the better, the larger the better, and the nominal the better.

Smaller-the-better: This is usually the chosen S/N ratio for all undesirable characteristics like defects, etc. for which the ideal value is zero.

Where

Y = responses for the given factor level combination

n = number of responses in the factor level combination.

Larger-the-better: This case has been converted from smaller-the better by taking the reciprocals of measured data

Where

Y = responses for the given factor level combination and,

n = number of responses in the factor level combination.

Nominal-the-best: This case arises when a specified value is most desired, meaning that neither a smaller nor a larger value is desirable.

Square of mean n = 10 Log10 ----------------- variance

S/N ratio is a measure to identify the parameter setting that minimizes the effect of noise on the response.

.

5.2.3 S/N RATIOS FOR CLEARNACE TIME

Minitab software was used to calculate the S/N ratios for clearance time. S-N ratio is a measure to identify the parameter setting that minimizes the effect of noise on the response. Depending on the goal of the experiment, which in this case is smaller-is-better for clearance time are chosen. Table 7 represent the response table for S/N ratio for clearance time

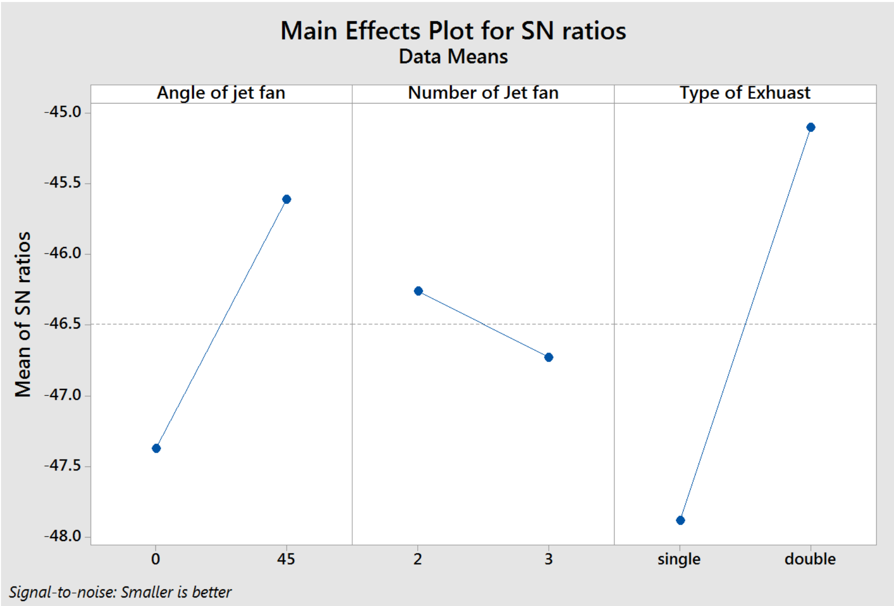


Figure 32 S/N ratio plot using Minitab

Table 7 Results of test on MINITAB

Level	**Angle of jet fan (A)**	**Number of jet fan (B)**	**Type of exhaust (C)**
1	-47.49	-46.49	-47.88
2	-45.84	-46.84	-45.45
Delta	1.64	0.35	2.43
Rank	2	3	1

From Table 7, it is observed that the optimum settings of parameters for minimum clearance time are A2-B1-C2 since the aim in a Taguchi experiment is to maximize the S/N ratio. The rank represents the parameter with the strongest influence which in this case is the type of exhaust.

Chapter 6

CONCLUSION

In this study CFD analysis was divided in to two stages

6.1 STAGE 1

Comparative research on car fire in parking garage was analysed by using two simulation software FLUENT and FDS. By comparing simulation results of FLUENT and FDS the conclusion is as follows:

- FDS is most appropriate software in field of building fire simulation because of its more flexibility in modelling of fire, jet fan, vents, less simulation time, single interface and accuracy to optimum solution.
- When the fire occurs in the middle of parking garage, in 350s, the simulation results of FDS is approximately similar to the simulation of FLUENT. From this we can conclude that FDS is capable of simulating fire and smoke and its accuracy is approximately similar to FLUENT.
- From underground car parking simulation studies, the two simulation software FDS and FLUENT , the impact of simulation results is decided by meshing accuracy, hence meshing refinement should be there, In order get more accurate solution.

6.2 STAGE 2

Design optimisation of car park using Taguchi Design of Experiment was simulated using FDS package. This stage has presented an application of the Taguchi method to the optimization of ventilation design in an electric drive. As shown in this study, the Taguchi method provides a systematic and efficient methodology for determining optimal parameters with far less work than would be required for most optimization techniques. The most important parameter affecting on clearance time is type exhaust fan. From result of Taguchi method it is concluded that A2-B1-C2 is the best combination for optimisation, i.e. 45° angle of jet fan – Two number of jet fans – Double exhaust is the best combination.

6.3 FUTURE SCOPE

Based on result and discussion summary, this project had achieved it main objective but an improvement still can done to improve the design ventilation in underground car park. Some of the suggestions to improve the result includes use of Taguchi optimisation in large scale level with complex geometry of car park. By increasing number of factors in optimisation, parametric model can be achieved.

REFERENCES

[1] A.D. Lemaire, J.L.M. Hensen (2013) Fire safety assessment of semi-open car parksbased on validated CFD simulations.

[2] Ali celen (April 2015), CFD Analysis of Smoke and Temperature Control System of an Indoor Parking Lot with Jet Fan. *Journal of Thermal Engineering, Yildiz Technical University Press, Istanbul, Turkey Manuscript Vol. 1, No. 2, pp. 116-130.*

[3] B. Merci(2013), Smoke and heat control for fires in large car parks: Lessonslearnt from research?, *fire Safety Journal 57.*

[4] Cai Yun, Li Lidan (2007). The Analysis of the Circulates Mode of Ventilation System in the Case of Fire of Subway [J]. *Journal of the Chinese People s Armed Police Force Academy, 23(2): 28-31.*

[5] (2007) CFD Modelling of Car Park Ventilation System, user guide for designers and regulators, *Federation of Environmental Trade Associations*.

[6] Eleni A (2009), “CO dispersion in a car repair shop” Seventh International Conference on CFD in the Minerals and Process Industries *CSIRO, Melbourne, Australia 9-11*.

[7] Lu **S**., Wang Y.H., Zhang R.F. and Zhang H.P. (2011) ‘’Numerical Study on Impulse Ventilation for Smoke Control in an Underground Car Park’’, *Procedia Engineering, Vol. 11, pp. 369–378.*

[8] Qian Zhang (2013), A CFD simulation of hydrogen dispersion for the hydrogen leakage from a fuel cell vehicle in an underground parking garage, International journal of hydrogen energy 38

[9] R.Ramkumar (2015), Enhancement of Fire Evacuation System in Car Parking Area Using CFD, *SSRG International Journal of Mechanical Engineering (SSRG-IJME) – volume 2 Issue 6.*

[10] Viegas J. (2010), The use of impulse ventilation for smoke control in underground car parks, *Tunnelling and Underground Space Technology, Vol. 25, pp. 42–53.*

[11] Wang H (2008), The design and implement of automatic control system for large-scale coal spontaneous combustion experiment unit. *Journal of Xi an University of Science and Technology, 28(1):6~10. (in Chinese)*

[12] Wang Binbin (2011), “Comparative Research on FLUENT and FDS’s Numerical Simulation of Smoke Spread in Subway Platform Fire” *Procedia Engineering 26 - 1065* – 1075.

[13] X. Deckers (2013), Smoke control in case of fire in a large car park: CFD simulations of full-scale configurations, *Fire Safety Journal 56.*

[14] YUAN Jian-ping (2011), Numerical Simulations on Sprinkler System and Impulse Ventilation in an Underground Car Park, *Procedia Engineering 11 634–639.*

[15] Zhang X.G., Guo Y.C., Chan C.K. and Lin W.Y. (2007), ‘’Numerical simulations on fire spread and smoke movement in an underground car park’’, *Building and Environment, Vol. 42, pp. 3466–3475.*

[16] Zhang Lin (2008), Effect of Ventilation System on Smoke and Fire Spread in a Public Transport Interchange, *Fire Technology, 44, 463–479.*