# Modeling and Simulation of the Fuel Injection Process in an Engine


**By**

**Mian Noman**

**School of Chemical and Materials Engineering**

**National University of Sciences and Technology**


**2021**

# Modeling and Simulation of the Fuel Injection Process in an Engine

Names: Mian Noman

Reg.No:00000277559

**This thesis is submitted as partial fulfillment of the requirements for the degree of**

**MS in Process Systems Engineering**

**Supervisor Name: Dr. Muhammad Ahsan**

**School of Chemical and Materials Engineering (SCME)**

**National University of Sciences and Technology (NUST)**

**H-12 Islamabad, Pakistan**

**April, 2021**

# THESIS ACCEPTANCE CERTIFICATE

Certified that final copy of MS thesis written by Mr Mian Noman Registration No 00000277559), of School of Chemical & Materials Engineering (SCME) has been vetted by undersigned, found complete in all respects as per NUST Statues/Regulations, is free of plagiarism. errors. and mistakes and is accepted as partial fulfillment for award of MS degree. It is further certified that necessary amendments as pointed out GEC members of the scholar have also been Incorporated in the said thesis.

Signature:____________________________

Name of Supervisor: Dr Muhammad Ahsan

Date:____________________________

Signature (HOD):____________________________

Date: ____________________________

Signature (Dean/Principal):____________________________

Date: ____________________________

Form Thi-I

National University of Sciences & Technologv (NUST)
MASTER'S THESIS WORK
Formulation of Guidance and Examination Committee

Name:Mian Noman
Department; SCME
Credit Hour Completed€)

NUST Regn No: 00000277559
Specialization: Process System Engineering
CGPA 3.38

Course Work Completed

3.31

| S/NO | Code | Title | Core/Elective | CH | Grade |
|---|---|---|---|---|---|
| | PSE-801 | Process Systems Theory | Core | 03 | B |
| | PSE-852 | Process Modeling and Simulation | Core | 03 | |
| | PSE-823 | Advanced Process Dynamics and Controls | Core | 03 | |
| 4 | PSE-802 | Optimization and Decision Analysis | Core | 03 | B |
| 5 | CSE-801 | Computational Fluid Dynamics | Elective | 03 | |
| 6 | EME-902 | Numerical Methods in Chemical Engineering | Elective | 03 | |
| 7 | ENE-809 | Waste Water Treatment and Design | Elective | 03 | |

Date 25-07-2019

Student's Signature

**Thesis Committee**

1. Name: Dr. Muhammad Ahsan (Supervisor) Signature:
   Department: Chemical Engineering
2. Name: Dr. Iftikhar Ahmad Signature:
   Department: Chemical Engineering
3. Name: Dr. Umair Sikander Signature:
   Department: Chemical Engineering

Date: 1/8/2019 Signature of Head of Department:

APPROVAL

1/8/2019 Date:

Dean/Principal

1. Distribution

Ix copy to E)€am Branch, HQ NUST

Ix copy to PGP Dte, HQ NUST

6//ec%1'vQ—

Additional

Form: TH-04

National University of Sciences &Technology (NUST)

**MASTER'S THESIS WORK**

We hereby recommend that the dissertation prepared under our supervision by

Regn No & Name: 00000277559 Mian Noman

Title: Modeling & Simulation of the Fuel Injection Process in an Engine.

Presented on: 04 Mar 2021 at: 1400 hrs in SCME on MS Teams

Be accepted in partial fulfillment of the requirements for the award of Master of Science degree in Process System Engineering.

Guidance & Examination Committee Mem bers

Name: Dr Iftikhar Ahmad Signature:

Name: Dr Umair Sikander Signature:

Supervisor's Name: Dr Muhammad Ahsan Signature:

Dated: 09.03.2021

Head of Department

Date 15/3/21

**Dean/Principal**

Date 16.3.2021

School of Chemical &Materials Engineering (SCME)

# Dedication

Dedicated to my Beloved Parents, family, and friends.

# Acknowledgments

All acclaim and eminence be to "ALLAH" a definitive creator of this universe, who endowed us with the ability to comprehend and made us curious to investigate this entire universe. Infinite greetings upon the leader of this universe and hereafter "HOLY PROPHET HAZRAT MUHAMMAD (PBUH)": the wellspring of beneficial information and blessings for whole humankind and Uma.

My deepest thanks to Ex-Principal SCME Dr. Arshad Hussain for being an excellent enthusiasm source and providing all these research work facilities.

I am highly thankful to my respected supervisor, Dr. Muhammad Ahsan, for the guidance and assistance during this project. It is his consistent motivation that helps to achieve this landmark. His guidance helped a lot in all research phases, starting from learning tools until the final composing of this thesis.

I am highly obliged to my family for their eternal love. Thanks for supporting me in each step of my research work. It was not impossible without their support.

# Abstract

The low efficiency of the internal combustion engine has always been a virulent issue. Researches are being carried out for decades to improve efficiency and eliminate consequent problems. This study focuses on using computational fluid dynamics to analyze the effect of inlet valve geometry and Spray angle on internal combustion engines' performance. Computational fluid dynamics (CFD) analysis considers snapshots of fuel flow in internal combustion at critical points. Combustion efficiency is affected by phenomena like swirl and tumble. CFD analysis is used to study these phenomena. It considers the role of intake port geometry and sprays angles in creating squish and swirl. Phenomena like swirl and tumble are vital to combustion of fuel, increasing the efficiency of combustion and engine overall performance. In the study, CFD is being utilized to verify an experimental study with ANSYS fluent solver. The results are being compared to the experimental values of the previous study. The information predicted by Fluent is being discussed. This study analyzes the variation of flow parameters like pressure, temperature, and velocity using the finite volume method (FVM) solver with the standard k-ε turbulence model in computational fluid dynamics (CFD) with different spray angles. Geometry has been designed using Design Modeler and CFD analysis of injection has been carried out using ANSYS Fluent. Four different cases were applied to all the designs to investigate the performance. Based on the standard performance parameters available in the literature, Parameters with better performance were selected. The best design was further validated with the existing design in the literature by giving the same boundary conditions. This study's design has shown better performance in all parameters than the design available in the literature.

**Keywords:** spray angle; Diesel engine; AHRR; CFD; Fluent; Shockwave; standard k-ε model

# Table of Content

# List of Tables

# List of Figures

# Abbreviations

AHRR (Apparent heat release rate)

CFD(Computational fluid dynamics)

CNG(Compressed natural gas)

IVC(Intake valve open)

EVO(Exhaust valve close)

# Chapter 1

## 1. Introduction

One of the most important revolution in the history of humankind is the invention of internal combustion Engine. In today's world Engines are primary devices utilized for converting the heat energy obtained from fossil fuels into useful work. Engines have extremely widespread applications ranging from Electricity generators to Jet planes[1]. Ever since engines are invented, researchers have tried to increase the efficiency of engines by adopting better engine cycles. Global warming and environmental pollution are intensifying problems in the current era, which threaten living beings' existence [2].

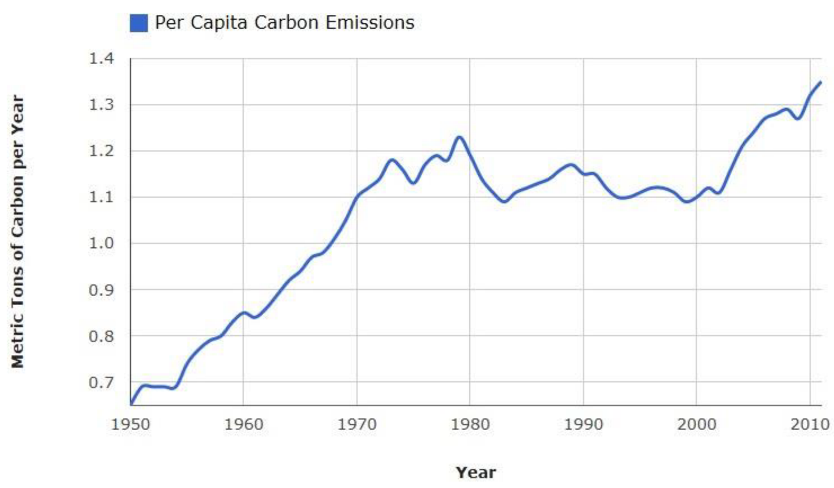



Figure 1 Global Per Capita Carbon Emission Estimates

In the past two centuries, non-renewable sources (fossil fuels) are extensively used to meet the world's energy demand. Almost 90% are consumed for transportation and energy generation. Extensive use of petroleum products is the actual cause of pollution and global warming [3].

In recent years increase in petroleum consumption and pollution, the usage of more efficient engines seeks much attention. Therefore, governments are pushing towards strict regulatory measures. Although scientists like Carnot laid strong theoretical foundations for ideal cycles, Otto, Diesel, etc. but real engines' technology has always been struggling to achieve better efficiency. The thermodynamic processes cannot be done with 100 percent efficiency due to inherent irreversibility. Ever since the invention, researchers are making efforts to improve its efficiency[4].

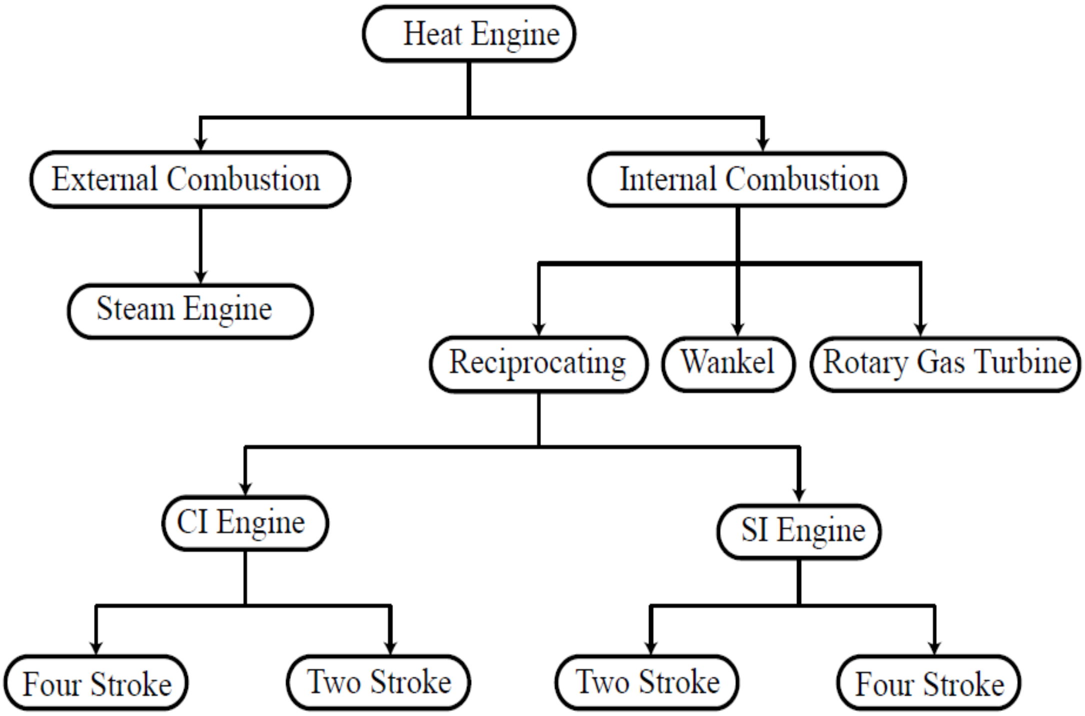



Figure 2 Classification of Engine

Chemical energy is converted into mechanical energy in an engine. However, according to the second law of thermodynamics, 100% conversion of heat, i.e., low-grade energy (LGE) into work, i.e., an (HGE) is never possible due to entropy and energy losses. This challenge drives the quest for exploring ways to improve the performance of the engine[5]. Combustion put a vital part in improving engine efficiency. Combustion of any fuel in the engine involves turbulent mass transfer, heat transfer, radioactive heat transfer, and chemical and physical processes. All these phenomena are steer by the geometry of engine inlet valve port. Since the geometry of engine plays a vital role in enhancing the performance of power generation plants and vehicles, it has been a research subject[6].

## 1.1 Classification of engines

Engines can be classified on the basis of combustion, ignition, or cycles. The classification of engine is shown in Figure 2.

### 1.1.1 Based on combustion type:

- External combustion
- Internal combustion

### 1.1.2 External combustion

In an external combustion engine, steam is generated using Many types of fuels like gasoline, diesel, wood and then it is used to move the piston. Combustion is carried outside in the engine cylinder and is usually called a steam engine[7].

### 1.1.3 Internal combustion (IC)

In IC engine, combustion occurs inside the engine, which is used to generate pressure, which causes the piston to move. Air insides the cylinder is heated with combustion, with the increase in temperature air pressure increases. High-pressure air moves the piston and rotates the crank[8].

### 1.1.4 Based on fuel type:

- Diesel/Biodiesel engines
- Gasoline engines
- Gas/CNG engines

### 1.1.5 Diesel/Biodiesel Engines

In diesel engines, diesel or high boiling hydrocarbons are used as a fuel for combustion and mostly, they are internal combustion engines. It can be designed for two or four-stroke cycle engines. They are primarily used for heavy-duty engines with low speed[9].

### 1.1.6 Gasoline Engine

In the gasoline or petrol engine, low boiling hydrocarbon is used as a fuel for combustion. SI engines and designing the engine are possible using two different ways, such as four or two-stroke cycle engines. They are used for low-duty engines with high speed.

### 1.1.7 Gas/CNG engines

Butane, propane, or other low boiling hydrocarbons are used as fuel. They mostly spark-ignition engines.

### 1.1.8 Bases of fuel ignition:

- SI engines
- Compression ignition (CI) engine

## 1.2 SI engine

In the SI engine, fuel and air are mixed and then drawn through injection nozzles into the engine's cylinder. Ignition of fuel is started using spark. Spark is generated by spark plug/point, which causes burn the fuel. These engines are often called fix or constant volume combustion engines.

### 1.2.1 Compression ignition (CI) engine

Compression ignition engine are those in which ignition is done by compressing the air. Air is compressed when the piston moves upwards, and its temperature rises to 600 to 700 C then the diesel or gasoline is sprayed using spray nozzles. High temperature in the cylinder cause ignition and the combustion is called constant pressure combustion because ignition pressure is constant at the time of ignition pressure[2].

### 1.2.2 Bases of the working cycle

- Engine with two strokes
- Engine with four strokes

## 1.3 Engine with four strokes

In this type of engine, two rotation of the crankshaft is complete in 4 strokes or one cycle. These events, which are taking place in four-stroke engines

- Air intake/suction ($1^{st}$ stroke)
- Compression/upward movement of the piston ($2^{nd}$ stroke)
- Power/downward movement of the piston ($3^{rd}$ stroke)
- Exhaust/emission ($4^{th}$ stroke)

## 1.4 Engine with two strokes

In a two-stroke engine, one cycle is completed in two strokes or one revolution of the shaft. These events take place in a two-stroke engine

- Air intake/fuel intake and compression (1st stroke)
- Power/downward movement of the piston and exhaust/emission (2nd stroke)

## 1.5 Computational Fluid Dynamics

Generally, CFD is defined as numerical analysis of the fluid motion using many equations such as mass conservation of fluid, energy conservation and momentum conservation equations. CFD uses the PC as a computational instrument to utilize scientific models and strategies to find the arrangement of complex liquid mechanics and discrete numerical issues. The Numerical solution is a calculation method in which discrete estimations are used, which generally follow the steps shown in Figure 3 [2].

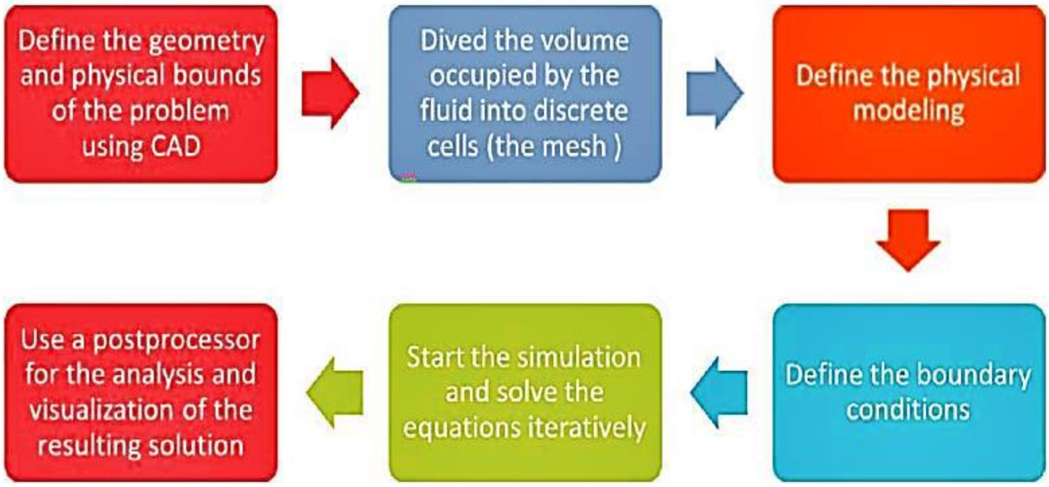



Figure 3 Simulation flow

Computational fluid dynamics (CFD) can be used for simulations and investigate the effect of different geometries on the phenomenon like swirl, tumble, and wake and vortex formation in a flow field. Computer-based software like ANSYS FLUENT etc., has increased the computational speed and theoretically simulates any physical condition[10].

CFD of CI engines has been researched over the years. CI engine's mathematical modeling is undoubtedly complex and therefore, it involves various interacting sub-models. Even today, CFD cannot be used for the absolute value calculation of C.I engine operational parameters.

Since CFD itself is subject to validation research, we will also view our subject matter. The development of steps involved in CFD is discussed for research done on CI engines. Brief comparisons in the early CFD models and their results versus the latest developments are presented[11].

### 1.5.1 Mesh Generation for FEM

Mesh generation is required to apply FEM. In simple words, to solve a problem, FEM requires a subdivision of a large surface divided into small parts called finite elements. Modeling these small parts is done using basic equations and after that, combining these equations helps solve the whole problem[12].

### 1.5.2 Types of meshes

Based on the generation process, meshes can be broadly classified as:

- Structured/pattern meshing
- Unstructured/Random Meshing
- Hybrid/mixed Meshing

### 1.5.3 Structured/pattern Meshing:

These meshes consist of hexahedral units, which are implicitly connected. Automatic mesh generation algorithms have difficulty in the generation of hexahedral elements for complex geometry. Therefore, for complex geometry, hexahedral mesh generation is an extensive time process in which the 3D model is divided into many regions that depend upon the type of analysis and geometry[13].

### 1.5.4 Unstructured/random mesh

An unstructured mesh has tetrahedral elements, which are connected explicitly. Automated mesh generation algorithms in most modern software generate tetrahedrons by default for most of the geometry.

### 1.5.5 Comparison

Structured meshes take more effort, time, and understanding to construct but provide more accurate solutions whereas unstructured meshes are more favorable to construct but compromise accuracy.

A choice between structured and unstructured mesh depends on the requirement of the problem at hand. In the CFD analysis of CI engines, a refined structured mesh is favorable in transient analysis types, described later in the article. Grid independence studies can give an idea about mesh requirements in different analysis types[14]. The mesh of the engine sector is shown in Figure 4.

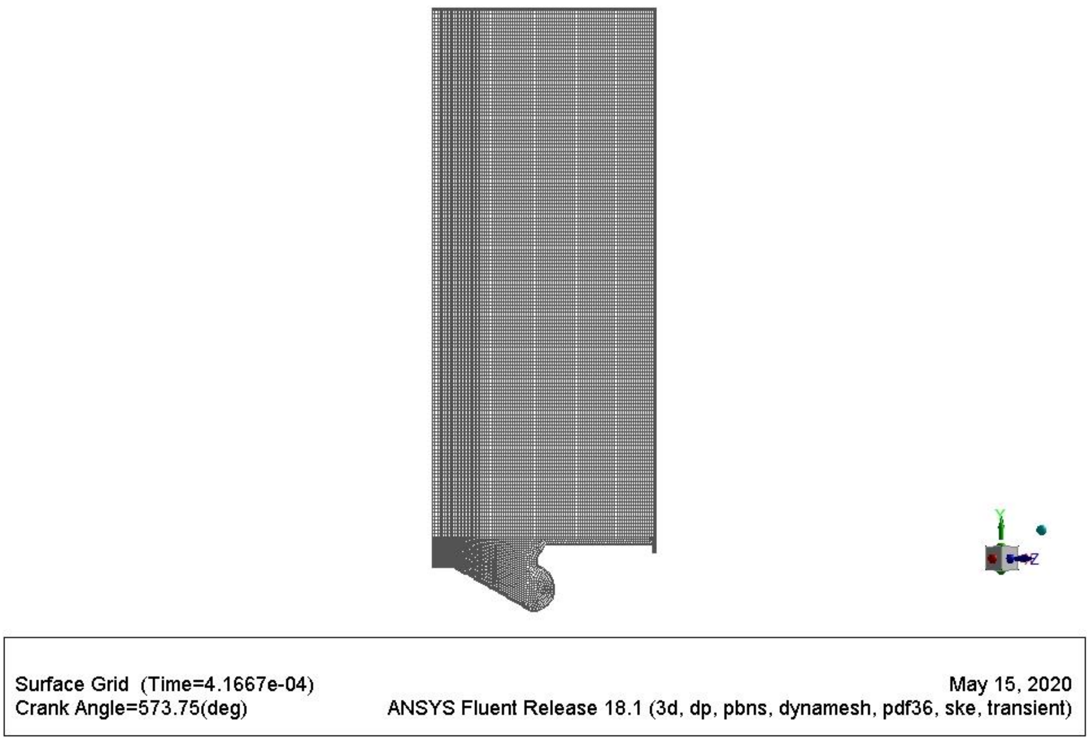



Figure 4 Mesh of Engine Sector

In the hybrid meshing, unstructured and structured are combined to get better results, some modern mesh generation programs are adopting hybrid meshing. Different cells are used in different regions, such as hexahedral or prismatic in the viscous region and tetrahedral for any other region[15]. In Viscous parts, a smaller number of cells are formed than other regions, such as in unstructured mesh. In hybrid meshing, there is no restriction for the faces or edges. The hybrid meshing more flexible than other types [6].

Unstructured and structured mesh both have their advantages and disadvantages. Unstructured mesh has more flexibility in solving complex problems. Structured meshes take more effort, time, and understanding to construct but provide more accurate solutions whereas unstructured meshes are more favorable to construct but compromise accuracy.

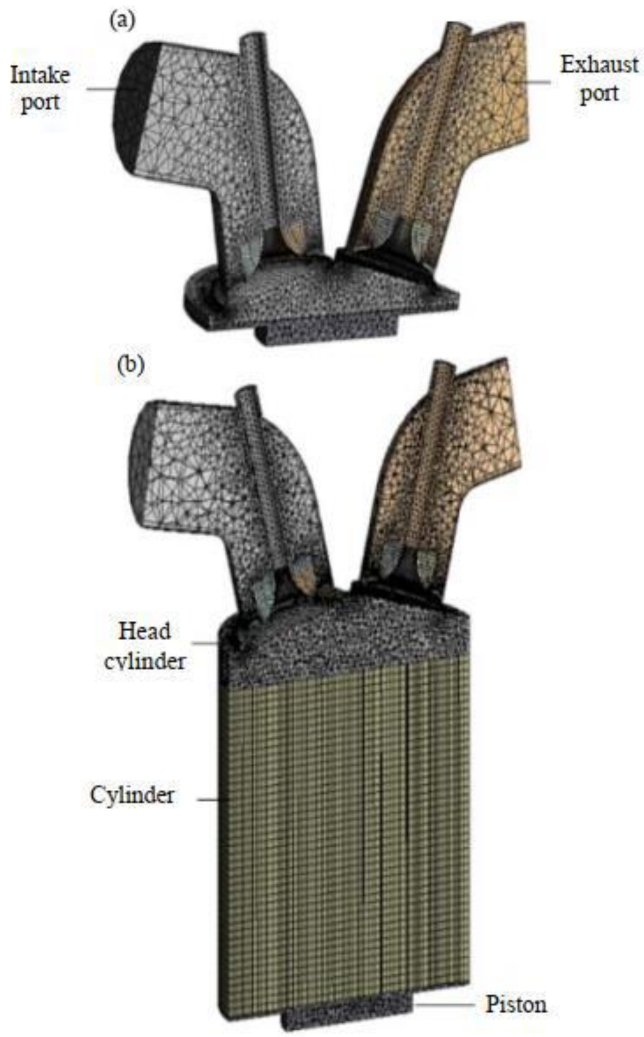



Figure 5 Hybrid meshing, structured mesh consisting of hexahedrons are visible on straight face whereas, in rest of is, tetrahedrons indicating unstructured patterns are visible

However, to join the boundaries of different meshes and their proper function interpolation relation problem must be solved carefully. In some cases, structured mesh gives better results. J L Gary smith explored using Computational Fluid Dynamics (CFD) to design a Jaguar Cars Ltd engine. He also developed some methods to make the mesh generation process fast and accessible [16]. He proposed that Computational fluid dynamics (CFD) can be used for the design's final stage instead of on the initial stage of designing engines. He used the prediction of Computational fluid dynamics (CFD) to investigate the engine's flow behavior inside the cylinder. J L Gray smith recommends the company dedicate lab/workshop for the CFD and train the worker, which are draughting engines to develop CFD models using computer-aided methods. His study summed up significant problems in the CFD of CI engines at that time[17].

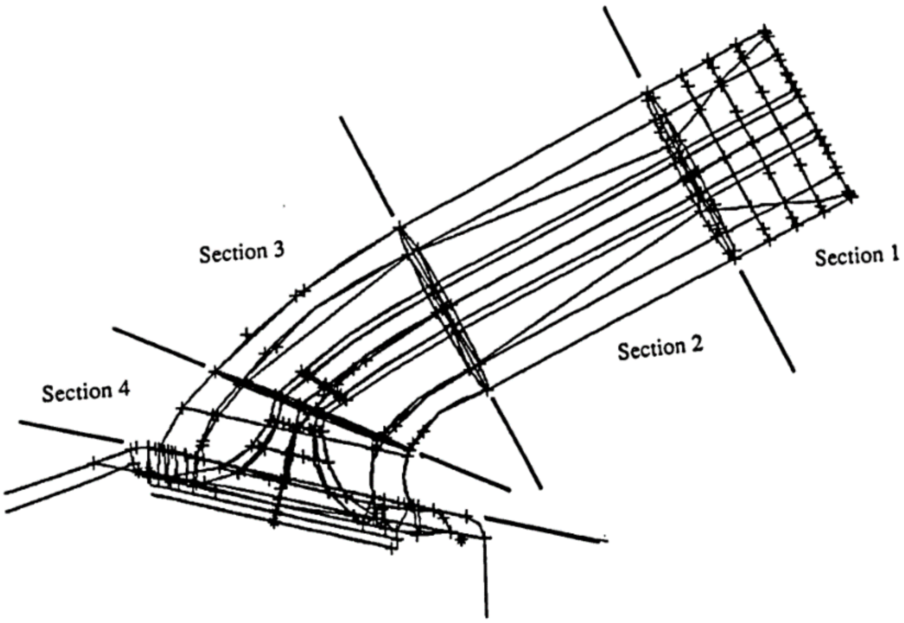



Figure 6 mesh J L Gary smith

Although modern meshing techniques also require simplification of geometry, meshing has been primarily automated and caters to earlier researchers' most challenges. Ahmad Al makky has shown various complex meshes generated using modern meshing tool (Figure 7).

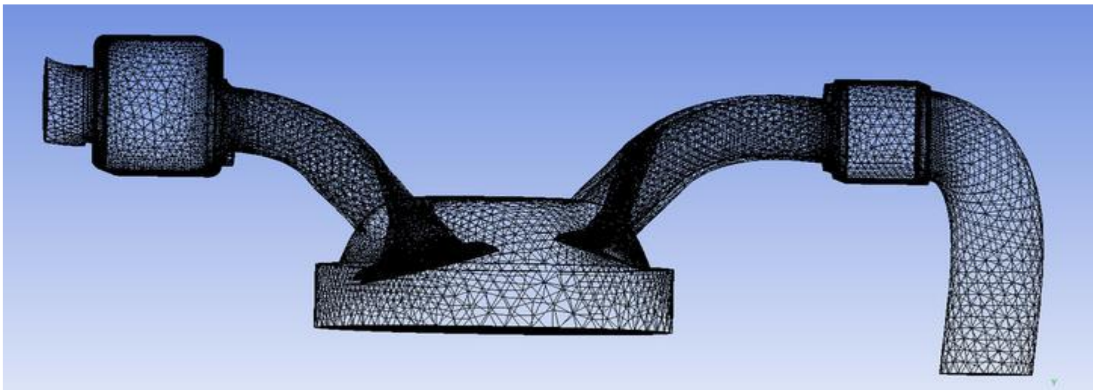


Figure 7 Mesh generated with Ansys

## 1.6 Types of Engine CFD simulation:

After meshing, it is necessary to identify the simulation types into which engine CFD is divided. Based on the type of simulation preprocessing is done. There are several types of CI engine's CFD simulation in the present era. Because of the complexity of the engine cycle all its phenomena cannot be simulated at once. Therefore, the relevant simulation is developed and studied[11].

The main classification of simulations presently done is

- Simulation using Cold flow
- Simulation of combustion
- Port flow

### 1.6.1 Simulation using cold flow

Cold flow prediction contains the flow of air model without considering chemical reactions in a transient engine cycle. Cold flow simulation is done so that process of mixture formation is captured precisely by studying fluid dynamics in the dynamic geometry. The features, which are changing the airflow jet, which produces tumbles and swirls through intake opening and closing of valves, and compression and squish-based turbulence production, can be studied[18].

This knowledge is vital; it makes sure of circumstances and conditions perfect for flame progression, propagation, and combustion. If the turbulence levels high, it takes part in quick flame propagation and the combustion process in a power stroke[19]. An appropriate proportion of turbulent airflow is crucial to ensure correct air and fuel ratio in the combustion process. Charge stratification can be assessed by using the CFD.

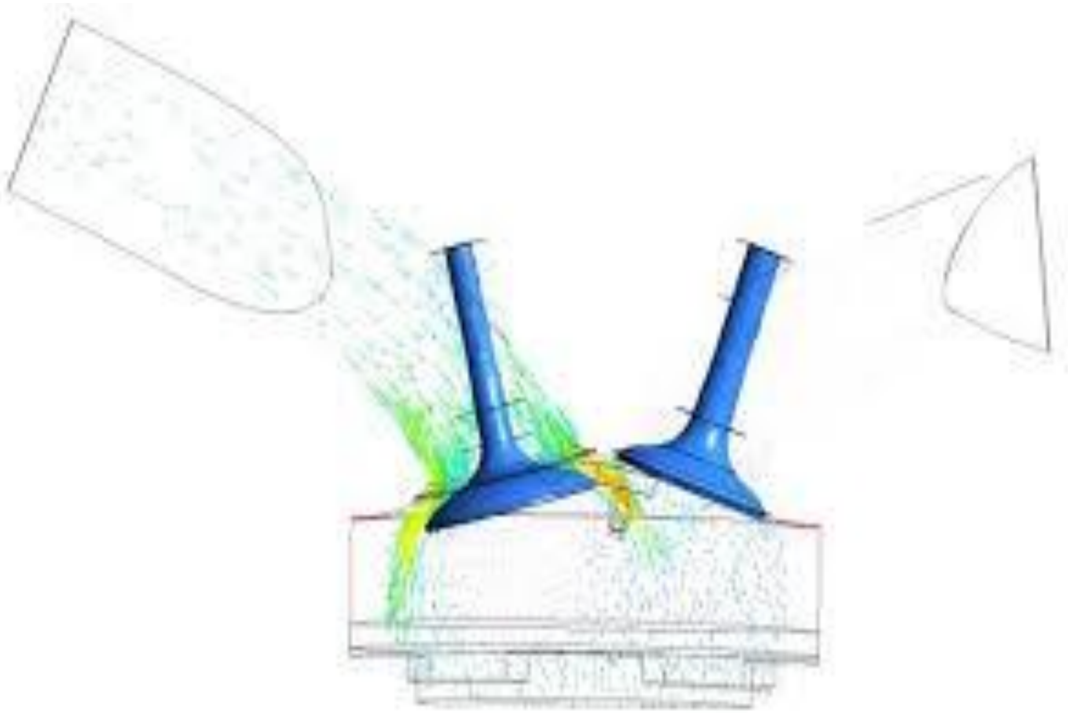


Figure 8 cold flow simulation

### 1.6.2 Simulation using combustion

This type of simulation involves the stroke in which combustion takes place. It starts from the intake valve close (IVC) and ends at the exhaust valve open (EVO). When combustion occurs, both intake and outtake valve remain closed; the only piston moves from top dead center to bottom dead center. This type of simulation analysis of parameters such as exhaust gases, pressure, apparent heat release is investigated. Usually, combustion simulation geometries are less complex than other types.  The geometries are usually symmetric and often have one dominating quality, which influences the whole calculation[20].

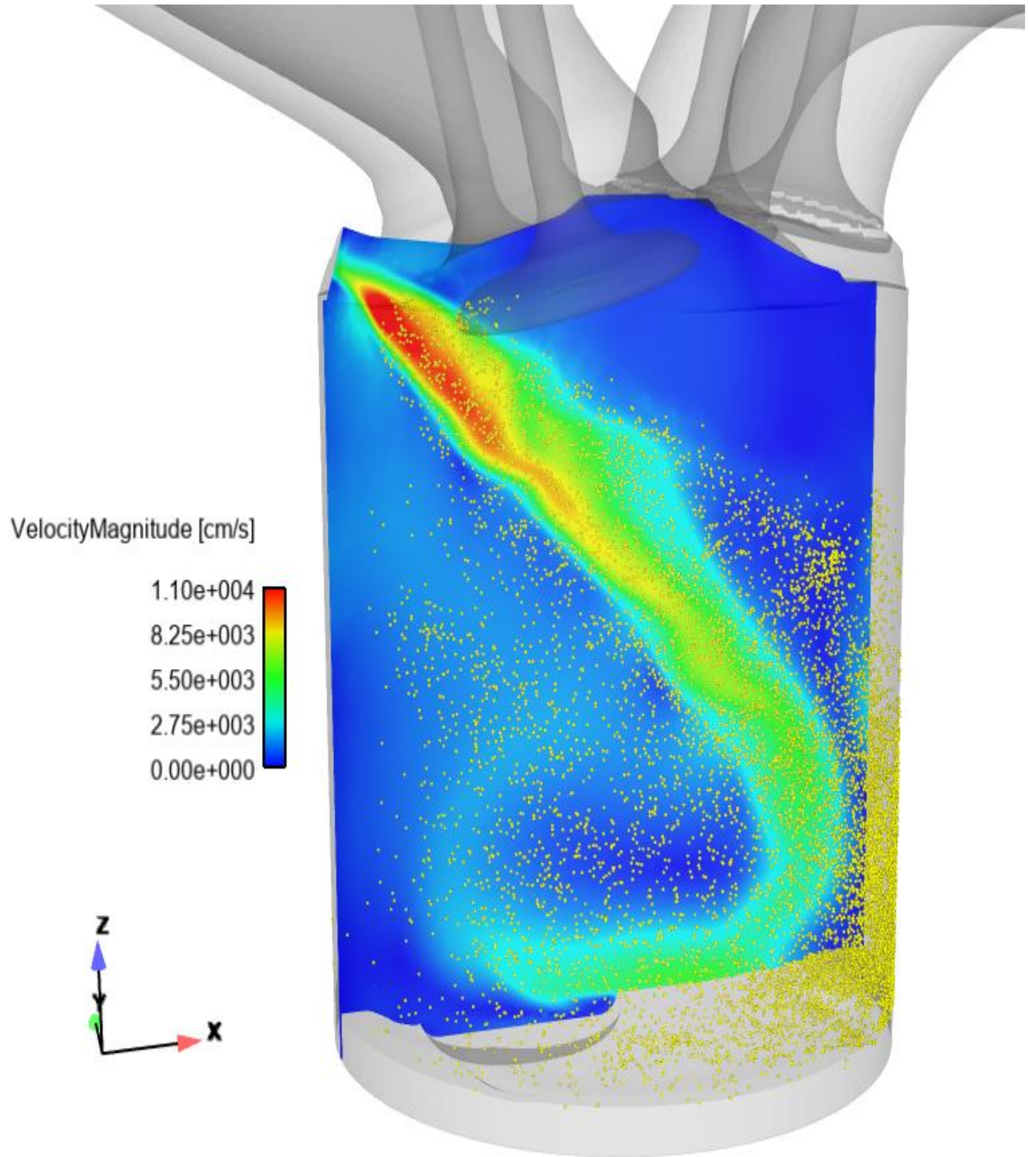



Figure 9 Combustion Simulation

### 1.6.3 Port Flow Simulation

In an engine, the cycle symmetry of cylinders and port valves is frozen at crucial points and with the help of CFD, airflow is studied and comprehended in the process of port flow analysis. It is also used to analyze flow rate volume and swirl through the engine. Other fluid dynamics mechanisms and phenomena can be comprehended[21].

This study allows better comprehension and modification of the ports' geometry to get desired results during the engine cycle. Different techniques like Laser Doppler velocity are used to measure velocities, turbulence and flow rates. Expansion and compression of air during the movement of the piston are not visible in the results[14].

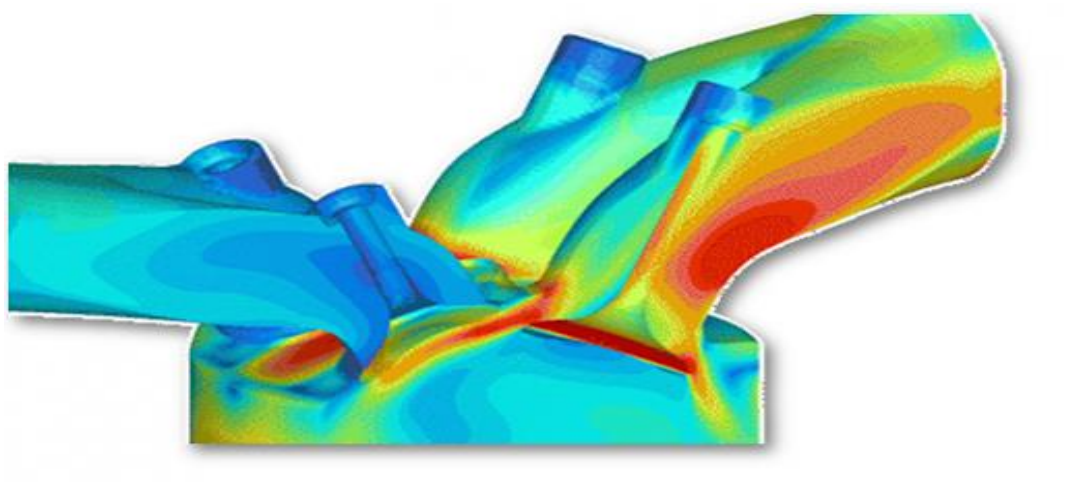


Figure 10 Port flow simulation

Practically Because Of CFP software, one-point port flow analysis is comparatively easy cause of its geometry. Both cylinder and port valves are set as a specific point, so the mass flow rate is determined then assigned for turbulence models and results are obtained. RANS approach is used to calculate the effects of turbulence. For a model setup and accuracy, experimental data is used.

With an increase in the number of positions and number of cases, problems become complex. A more significant number of static cases bear chances for errors[22].

### 1.6.4 Inside the cylinder flow model

At the very beginning of the engine cycle, air motion inside the cylinder can affect the performance. The incoming air produces flow structures with big-level turbulent motions in cylinders to determine the mixing between residuals and the new charge[23]. Field of flow has a significant effect on initial intake and conditions for burning of fuel. Transfer of heat, highest flame temperature and thermal stress levels are affected by flow field during combustion. The nature of fluid motion determines the extent of pollutant emissions in the post-combustion phase. Flow patterns also determine cyclical variations and some other processes.

An engine that uses a four-stroke intake path length can affect the performance motored at 1000-3000 rpm. Measurement of spontaneous fall in pressure and flow rate through intake path is filed as the quality of engine speed[24].

In the engine cycle's intake stroke, large flow structures are being dominated by the incoming jet's shape and direction. With Optimum manifold design and cylinder geometry, these structures may help control the degree of stratification quite accurately and tailor mixture preparation to purpose[25].

# Chapter 2

## 2. Literature Review

As discussed in Chapter 1, there are many design parameters of the Engine injection that are interrelated. On the other hand, we require to optimize the injection by changing different parameters. It can change one design parameter at once and observe its effect on the performance keeping other parameters constant following the design. Many studies were carried out to simulate the diesel engine to achieve the maximum output pressure and less NOx and other pollutants. The engine's overall efficiency can be enhanced by using efficient injection and atomization of fuel, which can increase the overall engine performance [26].

Many CFD simulations of diesel engines are carried out using different software to study the different engine parameters. According to RD Reitz, engine performance is greatly affected by details of the intake flow process and mechanisms. Pressure, heat release, soot, and NOx, in different experiments with standard engines show that CFD results are compatible with experiments[27]. A model experiment can predict soot NO, trade-off a trend in a better and accurate way. However, there is significant room for improvement in predicting the accuracy of combustion with late injection. A model cylinder was created then the effect of this model on intake and outtake valves are studied. Flow rate does not affect predictions and measurements of the flow, but valve lift does affect.[28] A study carried out by Zhang et al. targeted to analyze the different geometries on the phenomena of combustion. The behavior of flame movements is studied by using the correlation of the cross method. The method used to note the combustion motion of flame and temp of burning with high-speed photographs of cylinders is two-color method. Pumping rate, injection nozzle size, to timings of combustion processes were studied[29].

Z. Mahmood et al. investigated the two-intake valve engine and study the effect of port design on fluid flow. STAR-CD CFD, k-ε model, SIMPISO solution algorithm employed. LDA for experimental visualization. In Figure 8, when both intake ports are entirely open, if we increase the α angle 0º to 32 º, it shows an increase in the tumble. When angle β is increased from 0.5 º to 32 º causes small increases to swirl inside the cylinder. When the spray/intake angle is increased and the port is deactivated, it causes many changes in flow patterns[28]. When the port is diactivated, an increase in intake angle causes changing the flow motion inside the cylinder and enhancing the swirl and tumble.

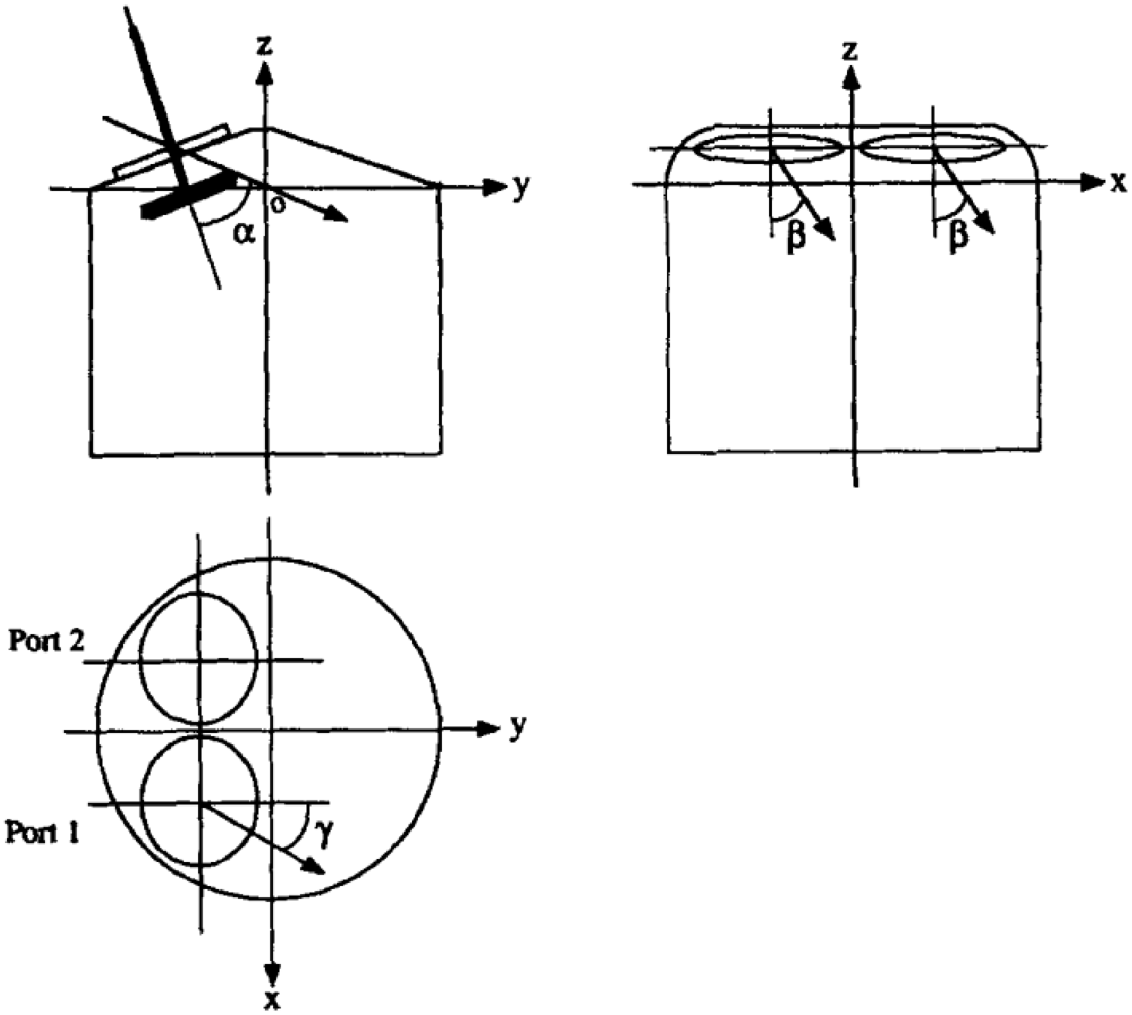



Figure 11 Configurations studied by Z. Mahmood

According to RD Reitz, engine performance is greatly affected by details of the intake flow process and mechanisms. Pressure, heat release, soot, and NOx, in different experiments with standard engines show that CFD results are compatible with experiments. A model experiment can predict soot NO, trade off-trend in a better and accurate way, though there is significant room for improvement in predicting combustion accuracy with late injection. A model cylinder was created and its interaction with valve and cylinder walls was studied. Flow rate does not affect predictions and measurements of the flow, but valve lift does affect[16].

A study carried out by Zhang et al. targeted the Pumping rate, injection nozzle size about timings of combustion processes were studied [30]. Chen, Dent conducted the experiments and computational studies to investigate the curve's effect in the inlet port. Flow patterns were analyzed using STAR-CD CFD code. Surface pressure maps of localized regions were obtained experimentally [21].

C.W. Hong et al. helped design an advanced engine with optimized in-cylinder air motion and controllable turbulence level to meet strict emission requirements. The analysis comprised of turbulence effect produced by mixture flow inside the cylinder, direct measurement using simultaneous methods, CFD using KIVA-3 and k–ε turbulence model. Measurement of integral length with computational length scale shows reasonable similarity in both tendency and magnitude [30].

M.Reeves studied the cyclic flow variations and their effect on the burning process. A turnkey, high-speed digital image is used in combination with PIV. It provided the visual of the process Synchronization scheme allows the acquisition of hundreds to millions of stream visualization series at the same crank angle, enabling very detailed visualization.

Leylek et al. Loss pockets examined the intake section and variation in the valve lift are predicted and identified using mathematics to investigate the total pressure losses. Details of the prediction for the high valve lift scenario are put forth. The conclusion is that a significant effect occurs in the valve's clearance; around 30% loss of pressure occurs upstream of the valve clearance section[26].

Using the CFD code, the precision of forecast for air motion and their spray properties in a CI diesel is evaluated by Bo et al. The Laser Doppler Anemometry (LDA) uses the compression and intake strokes until ignition of the fuel to determine velocity field. The conclusion drawn is that the forecast for the velocities of gases, droplets, and their penetration length mostly match experimental results, but further studies and examination are required to check the spray modeling results, especially concerning effects on gas temperature and fuel vapor[23].

Chen et al. use STAR-CF version 2.21 to simulate the fluid flow for the intake stroke and the flow is transient. In case turbulent flow and its explanation of boundary layer behavior is done using the standard k-e model[30].

Auriemma et al. has done in-cylinder measurements with motoring conditions. The air velocity parameters that are radial or tangential were measured for crank angle totaling 95-degree top dead center. Using various averaging methods, estimation is done for the shear stress, motion, and integral time[3].

Beard et al. studied the fluid flow in the compression ignition engine by carrying out experiments and using computations in tandem. The author modified the KIVA II code to perform simulations and consider the optical piston elasticity because of high pressure[31].

By applying the first law of heat, the leading cause of error in gross heat release estimations is evaluated by authors Brunt and Platts. They find that the leading cause of the error results from using the erroneous heat transfer ratio through the wall and their specific heat. The authors then propose a substitute model of the release of heat, called a polytrophic model. Using this model, a comparison is made with the results obtained through direct compression ignition engine experiments[32].

Olakzai et al. discussed a functional design for direct diesel intake and combustion engines. For this, an upgrade of the CFD approach is examined. The model of Tetrahedron cells is adequate to obtain a corresponding examination of steady-state flow. A previous study using different simulation software is shown in table 1.

Table 1 Previous computational studies on engine

| Year | Author | Research Objectives | Experiments or Simulation | Main Results |
|---|---|---|---|---|
| 1995 | R. D. Reitz | Use of improved KIVA code and its experimental validation [33] | KIVA-3 intake flow simulation, experiments on single-cylinder pressure, heat release, soot, and NOx, in different experiments with standard engines, show that CFD results are compatible with experiments. PIV used. | The intake of fuel affects the efficiency of the engine. A model experiment can predict soot NO, trade off-trend in a better and accurate way, though there is big room for improvement in predicting combustion accuracy with late injection. |
| 1996 | Z MAHMOOD ET AL | investigated the two-intake valve engine and study the effect of port design on fluid flow | STAR-CD CFD, k-ε model, SIMPISO solution algorithm employed. LDA for experimental visualization | STAR-CD CFD, k-ε model, SIMPISO solution algorithm employed. LDA for experimental visualization. See Figure 8. When both intake ports are entirely open, we increase the α angle 0º to 32 º show an increase in tumble when angle β is |

| | | | | |
|---|---|---|---|---|
| | | | | increased from 0.5 º to 32 º. |
| 1998 | C.W. Hong | to help design an advanced engine with optimized in-cylinder air motion and controllable turbulence level to meet strict emission requirements. [24] | He helped design an advanced engine with optimized in-cylinder air motion and controllable turbulence level to meet strict emission requirements. | The analysis comprised of turbulence effect produced by mixture flow inside the cylinder, direct measurement using simultaneous methods, CFD using KIVA-3 and k–ε turbulence model. |
| 1999 | M. Reeves | the cyclic flow variations and their effect on the burning process. | A turnkey, high-speed digital image is used in combination with PIV. It provided a visual of the process | Synchronization scheme allows the acquisition of hundreds to millions of stream visualization series at the same crank angle, enabling very detailed visualization. |
| 2003 | F. Payri | Comparison of different piston configurations and assess the effect of piston geometry on intake flow | Flow field measurements using LDV, CFD with discretized Naiver-Stokes equations and k–ε turbulence model, pressure | The conclusion is that a significant effect occurs in the clearance of valve; around 30% loss of pressure occurs the upstream of the valve clearance section. |

| | | | | |
|---|---|---|---|---|
| | | | correction method and PISO algorithm | |
| 2011 | Fredrik Erling | computations of the flow over the valve design to determine how it affects the volumetric and fuel conversion efficiencies | Using the CFD code, the precision of forecast for air motion and their spray properties in a CI diesel engine. | the valve new valve design see fig, functions correctly in four strokes, however, its breathing mechanism requires further analysis |
| 2011 | Auriemma et. al | Optimization of the intake manifold by observing flow structures | The parameters of air velocity, which are radial or tangential, were measured for crank angle totaling 95-degree top dead center | Using a variety of averaging methods, estimation is done for the shear stress, motion, and integral time |
| 2012 | Chen et al | al use STAR-CF version 2.21 to simulate the fluid flow | In case turbulent flow and its explanation of boundary layer behavior are done using the standard k-e model. | The conclusion drawn is that the forecast for the velocities of gases, droplets and their penetration length mostly match experimental results |
| 2013 | H. Sushma and | Investigated the effect of different positions of piston on engine efficiency[34]. | CFD simulation using<br>ANSYS fluent 13.0 | Piston B creates a higher swirl inside and volumetric efficiency. |

| | | | | |
|---|---|---|---|---|
| | Jagadeesha. K. B | | | |
| 2014 | Beard et al | studied the fluid flow in the compression ignition engine by carrying out experiments and using computations in tandem | The author modified KIVA II code to perform simulations and taking into consideration the optical piston elasticity because of high pressure. | Using this model, comparison is done with the results obtained through experiments of direct compression ignition engine |
| 2017 | Hiregoudar Yerrennagoudaru | discussed a functional design for direct diesel intake and combustion engines[35] | For this, an upgrade of CFD approach is examined. Model of Tetrahedron cells is adequate to obtain a corresponding examination of steady-state flow | Modified piston id predicted to have better life and less thermal strain. Also, the Platinum coating will provide more uniform temperature distribution, which will help burn fuels with a lower cetane value |
| 2018 | P. Ragupathi | Investigate air pressure and flow structures inside of the intake manifold, especially for the plenum, for a purposed design. See fig 12 | CFD performed using ANSYS. CAD made with reverse engineering | They find that the leading cause of the error is the result of using the erroneous ratio of heat transfer through the wall and their specific heat |

# Chapter 3

## 3. CFD Modeling and Simulation Setup

This chapter describes the brief background of the flow solver and the basic governing equation being used. The tool, which is used to carry out our simulations and the turbulence model is used, is also covered in this chapter. In the end, the Engine dimensions and its setup in software is described. ANSYS IC Engine was used for all types of numerical results. ANSYS design modeler and mesh generation tools were used for the engine's design and grid generation, respectively.

Computer simulation of the engine was done using computational fluid dynamics. CFD is defined as numerical analysis of the fluid motion using many equations such as mass conservation of fluid, energy conservation and momentum conservation equations. CFD uses the computing power of computer as a computational instrument to utilize scientific models and strategies to find the arrangement of complex liquid mechanics and discrete numerical issues. By doing the simultions with the help of Ansys fluent (IC Engine) it give us the different parameters of engine such as temperature , pressure and turbulane in multiphase flow. With these properties we can estimate the results of problems before physical experiments. Designing of engine is complex process which involve the complex geometry and equations and take lot of time, simulation helps us to peridict the results. Optimization of these results can be done by ploting results vs different boundary conditions [24].

### 3.1.1 Basic equations

The basic equation used for simulation are momentum conversation equation and mass conversation equations.

### 3.1.2 Conservation of mass

The differential form

$$\frac{\partial \rho}{\partial t} + \nabla . (\rho \vec{v}) = S_m \tag{9}$$

Conservation of momentum

as

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F} \qquad (10)$$

### 3.1.3 k-ε Turbulence Model

k- ε turbulane model is used of many engineering problems which make the prediction of trubulance properties easy. k- ε tubulance model can only used for turbulent flow because model assume that the flow is turbulent and other properties are negligible such as viscosity. Model used turbulent kinetic energy and rate of dissipation for calculations[36].

### 3.1.4 Transport equations for the standard k-ε model

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\frac{\mu+\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \qquad (11)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma}\right)\frac{\partial\varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \quad (12)$$

### 3.1.5 Modeling the turbulent viscosity

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \qquad (13)$$

Whereas, $C1\varepsilon$, $C2\varepsilon$, $C\mu$, $\sigma_k$ and $\sigma\varepsilon$ are the constants in the abovementioned equations and their default values are mentioned below.

$C1\varepsilon = 1.44$, $C2\varepsilon = 1.92$, $C\mu = 0.09$, $\sigma k = 1.0$, $\sigma\varepsilon = 1.3$

Computational Fluid Dynamics (CFD) is widely used for analyzing fluid flow by solving governing equations of fluid dynamics. Many CFD simulations of diesel engines are carried out using different software to study the different engine parameters. Performance of engine is depends upon the details of the intake flow process and mechanisms. Output pressure of engine, AHRR in different experiments with standard engines show that CFD results are compatible with experiments. A model experiment can predict soot NO, trade-off a trend in a better and accurate way. CFD analysis involve the naiver strokes equation and the general step involve to solve the problem are given below.

- Modeling
- Meshing
- PreProcessing
- Solver
- Post-Processing

Here, the modeling and meshing are done using ANSYS Mesh and Design Modeler. ANSYS Fluent is used as a solver tool.

## 3.2 Nozzle Modeling and Meshing

It is described in Chapter 1 that few of the design parameters, which directly can affects the overall effficency of the engine. Spray angle is one of those parameters. In this thesis, we have designed and studied 04 different models to investigate fuel injection behavior inside the engine. The overall length and the throat area are kept constant in all designs. However, the exit radius is a variable parameter. The proposed Geometry of the engine is shown in Figure 9. To reduce the computational time, this model uses 60-degree sector of the engine.[13]

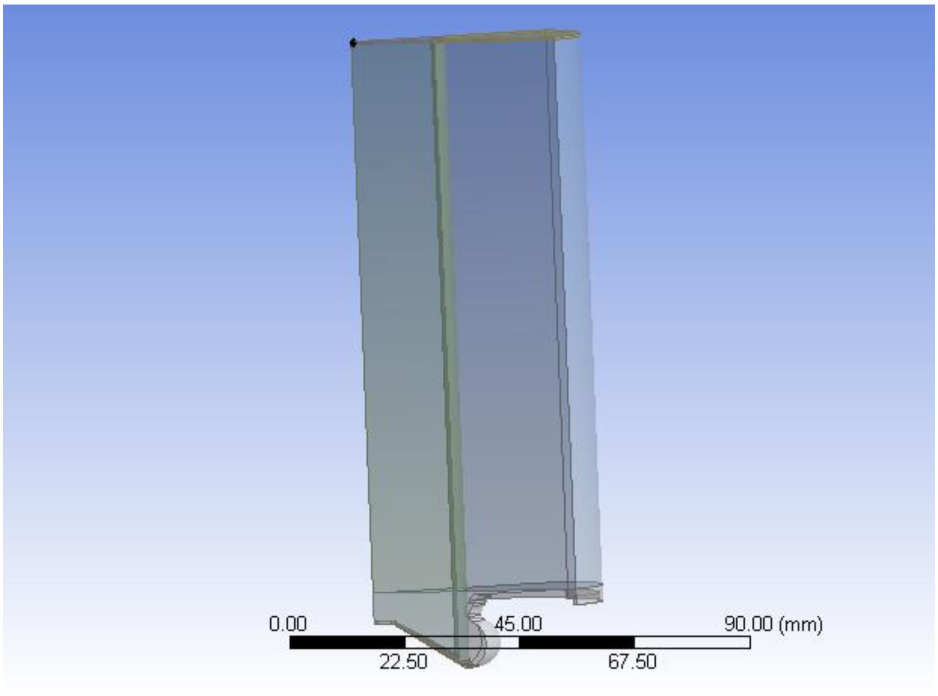



Figure 12 Geometry of engine sector

The engine sector's designing, modeling is done using Design Modeler. Design modeler has user friendly interface which help to create the geometry more ecurate than other available softwares. Mesh generation is done using ANSYS Mesh. Auto generation of meshing is used for meshing which is available in IC engine module. The meshing of all the designs was carried out and displayed in Figure 10.

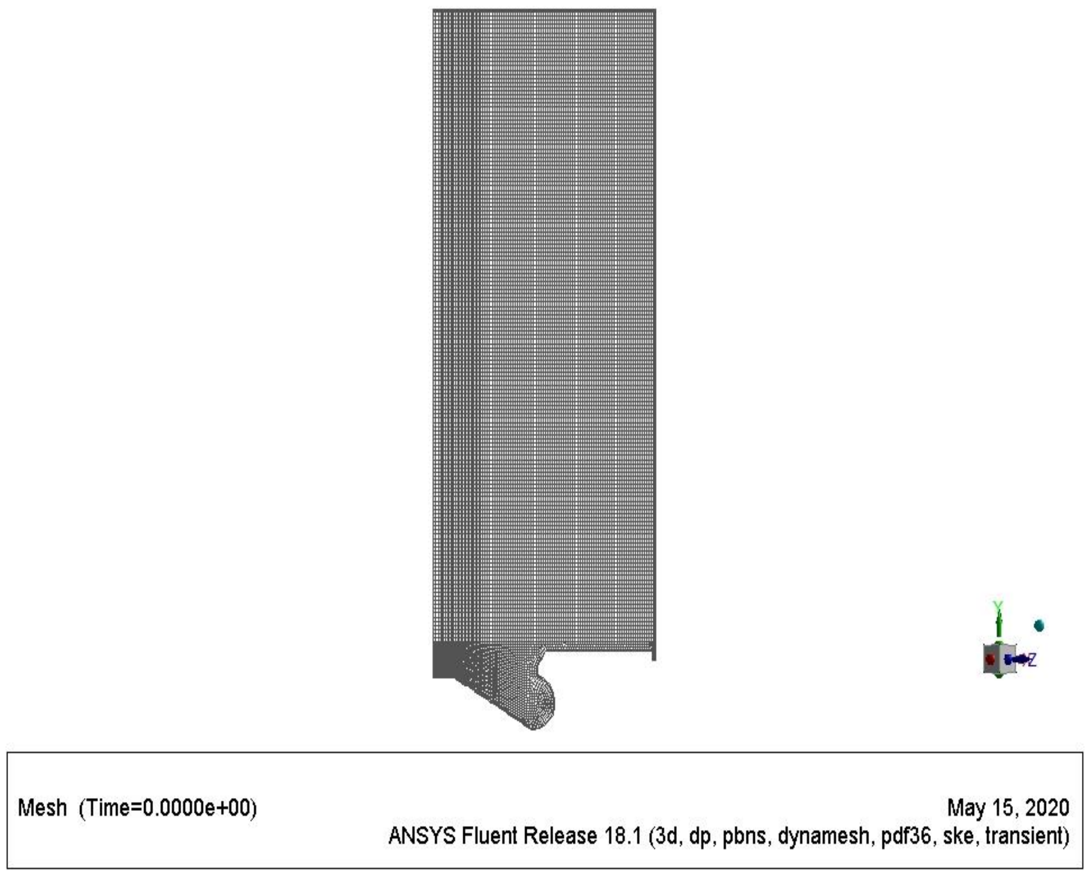



Figure 13 Mesh of engine

## 3.3 Solver Setup

Computational analysis of the fulid floe inside the engine is done using ANSYS Fluent/ IC Engine. Analysis is done to predict the many factors like maximum pressure inside the cylinder, maximum temperature, Apparent heat release rate (AHRR), velocity, swirl ratio, unburnt fuel, NOx. The solver procedure is shown in Figure 11.

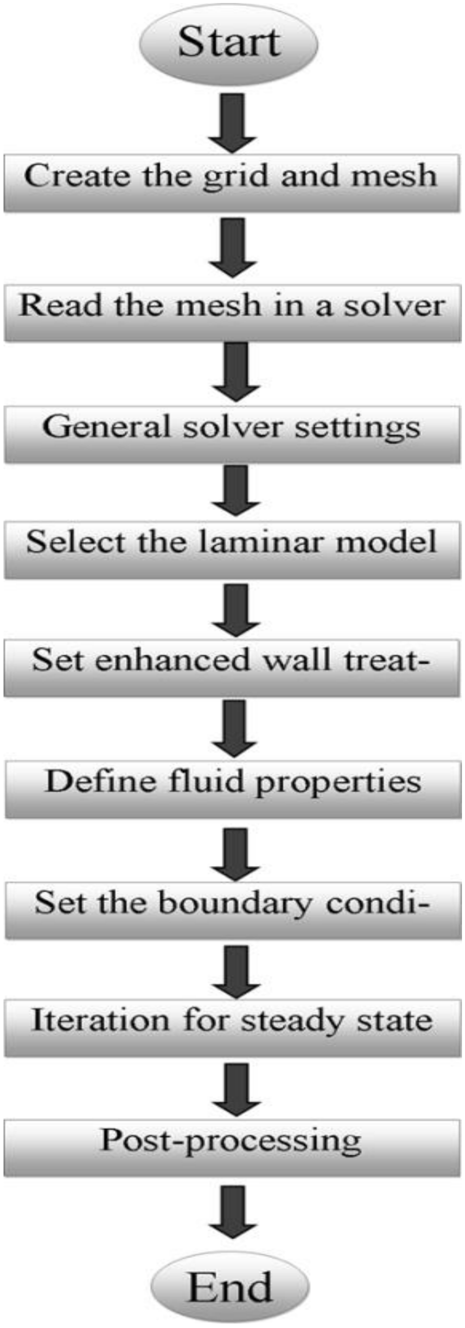



Figure 14 solver setup

In the Preprocessing the theoretical information of the is given to software such as the type of fluid, material properties and there boundary conditions. Theses properties and boundary conditions are shown in table 2. The boundary conditions were given according to a standard available in the literature to validate any small-scale Engine design. [4].

Table 2. General Setup Information

| **General Setup** | |
|---|---|
| Type of solver | Based on pressure |
| 3D | Dynamic |
| Time | Steadystate |
| Engine | 60 ° sector |
| **Case 1 Boundary Conditions** | |
| Inlet pressure | 3.5 **bar** |
| Inlet temperature | 404K |
| Injection angle | 50 ° |
| **Case 2 Boundary Conditions** | |
| Inlet pressure | 3.5 bar |
| Inlet temperature | 404K |
| Injection angle | 60 |
| **Case 3 Boundary Conditions** | |
| Inlet pressure | 3.5bar |
| Inlet temperature | 404K |
| Injection Angle | 70 ° |
| **Case 4 Boundary Conditions** | |
| Inlet pressure | 3.5 **bar** |
| Inlet temperature | 404K |
| Injection angle | 80 ° |

The abovementioned conditions were given to all the ANSYS Fluent designs to analyze the flow and performance parameters. The design with better results was selected and then it was compared with already available design in literature with the same boundary conditions to validate our results.

# Chapter 4

## 4. Results and Discussion

In this chapter, results obtained from the simulation are discussed. The geometry of the engine is cut into a 60-degree sector to reduce the computational time. The calculation starts from the intake valve close (IVC) 570-degree crank angle and ends at Exhaust valve open (EVO) 833-degree crank angle. Different parameters are analyzed by varying the spray angle. Geometrical information of the engine is shown in table 3.

Table 3 Geometrical Characteristics

| **Geometrical Characteristics** | |
|---|---|
| Combustion simulation type | Sector (60 °) |
| Connecting rod length | 165 |
| Crank radius | 55 |
| Starting crank angle | 570 ° (IVC) |
| End crank angle | 833 ° (EVO) |
| Compression Ratio | 15.75 |
| **Boundary Conditions for Case 1** | |
| Spray Angle | 50 ° |
| **Boundary Conditions for Case 2** | |
| Spray Angle | 60 ° |
| **Boundary Conditions for Case 3** | |
| Spray Angle | 70 ° |
| **Boundary Conditions for Case 4** | |
| Spray Angle | 80 ° |

Results are shown using contours of velocity, temperature, and graphs of AHRR, maximum pressure, maximum temperature, and Swirl ratio.

The spray cone of all cases is shown in Figure 12.

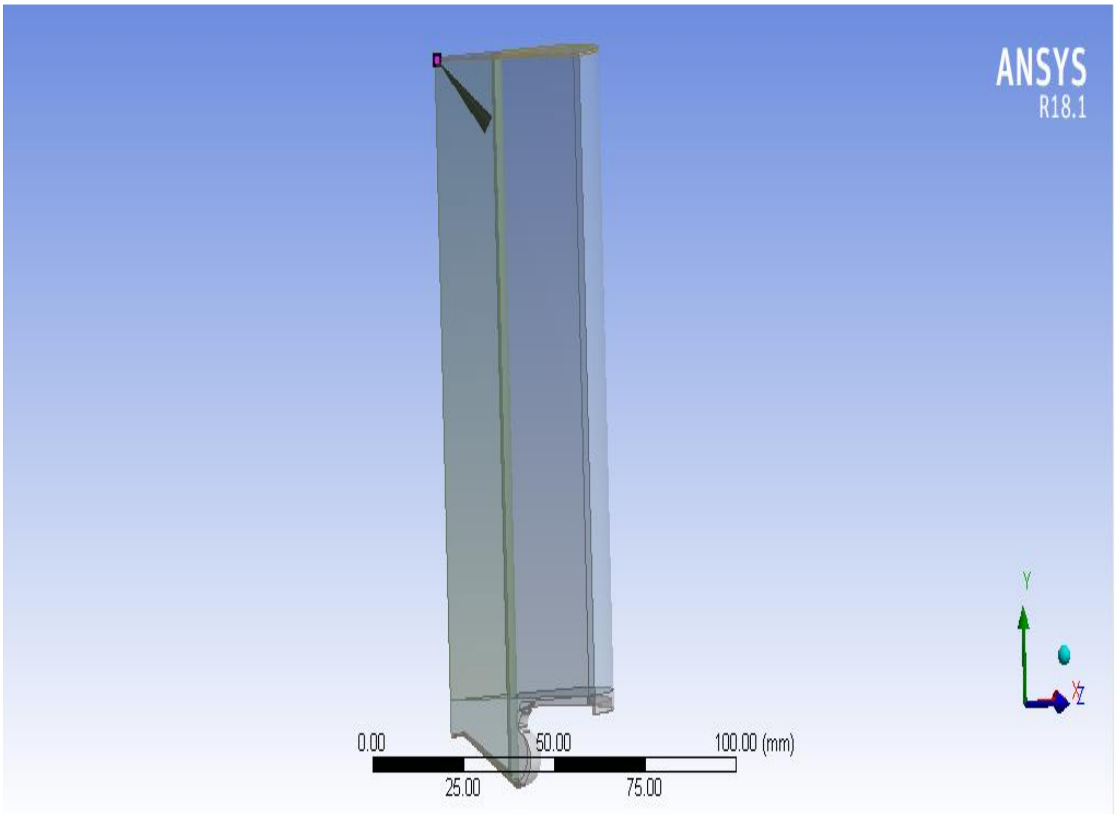



Figure 15 cone at 50 ° spray angle

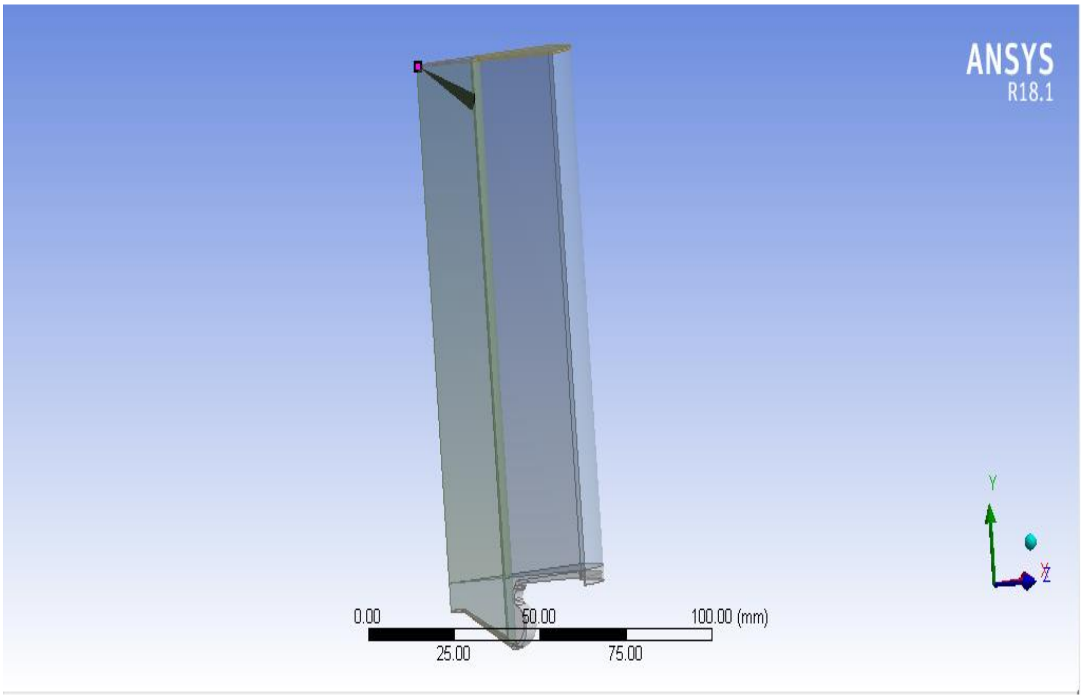



Figure 16 Cone At 60 ° Spray Angle

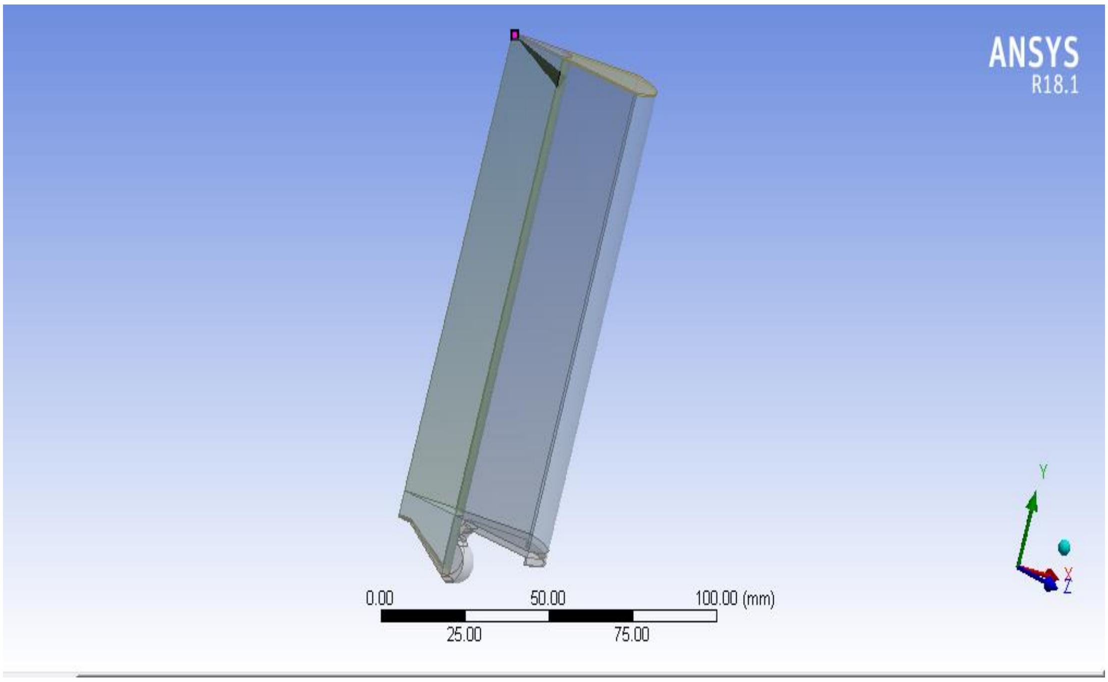



Figure 17 Cone At 70 ° Spray Angle

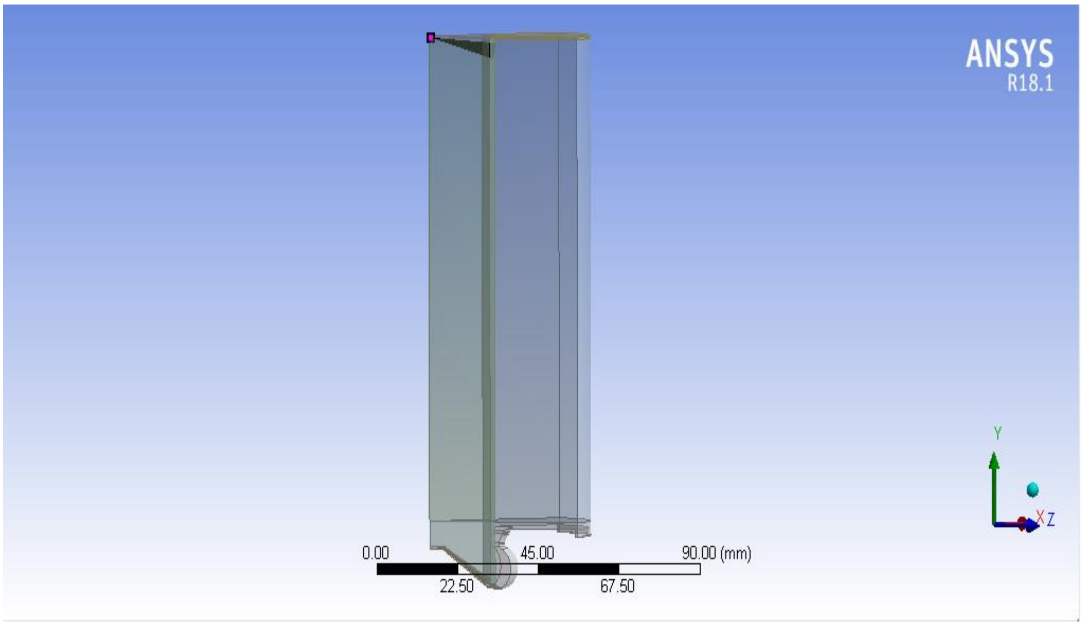



Figure 18 Cone At 60 ° Spray Angle

## 4.1 Meshing and conditions

This case was established to study the effect of spray angle on different parameters of the engine. The initial pressure was set to 3.5bar. However, the initial temperature was kept at 404K. The standard k-ε turbulence model was used to analyze the spray behavior and its ignition process. These conditions and turbulence models were applied to all designs. The behavior of pressure, temperature and velocity contours is discussed below. The particle traces of injected fuel inside the cylinder is also shown and discussed for all the designs. The spray angle in case 1 is 50°. Spray height and radius are 3 mm, 5mm, respectively. Engine speed is kept at 1500 rpm and the diesel filament model is selected from the species models list. The initial swirl ratio of the internal combustion engine is 1.3.

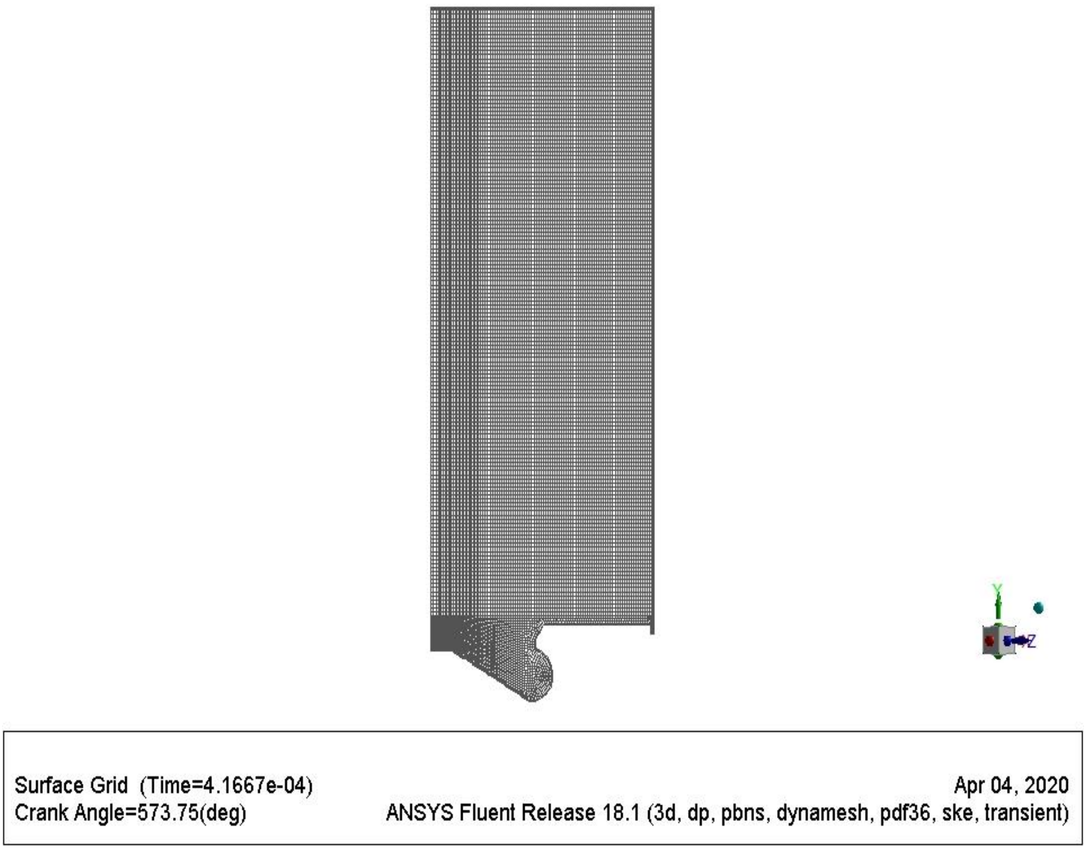



Figure 19 Mesh

## 4.2 Temperature contours

Comparison of temperature contours using four spray angle 50°, 60°, 70°, 80° degree at crank angle 570° (IVC), 720° (start of injection), 732.95° (end of injection), 832.95° (EVO).Temperature contours are shown below in figures 14, 15, 16.

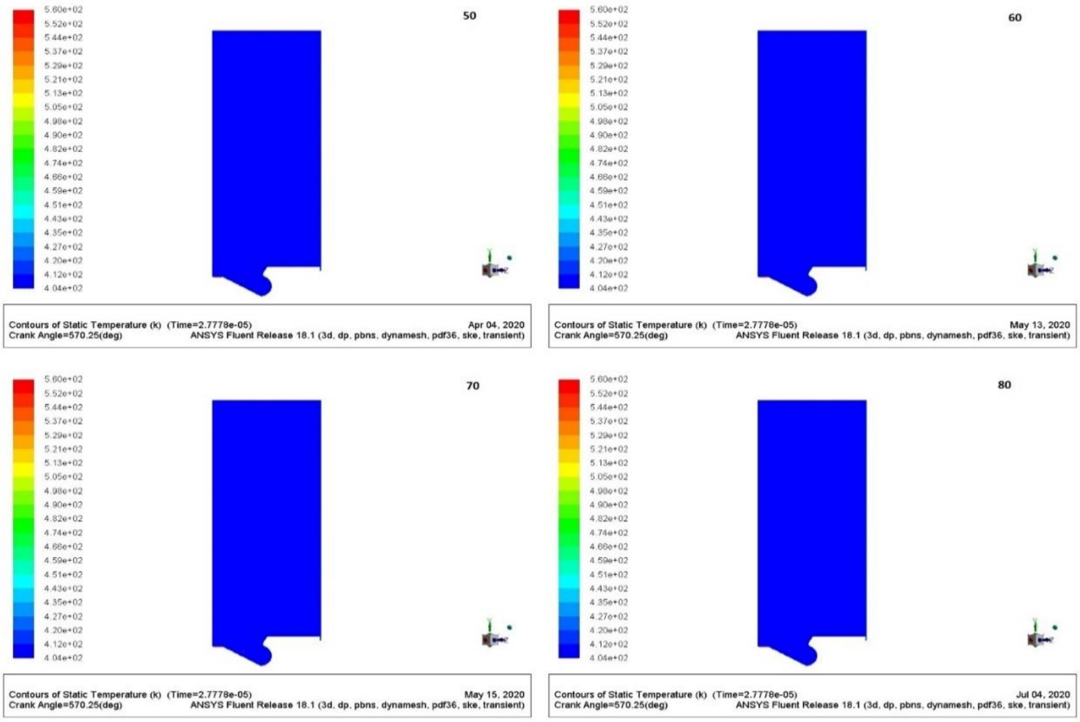



Figure 20 Temperature contours at 570° CA

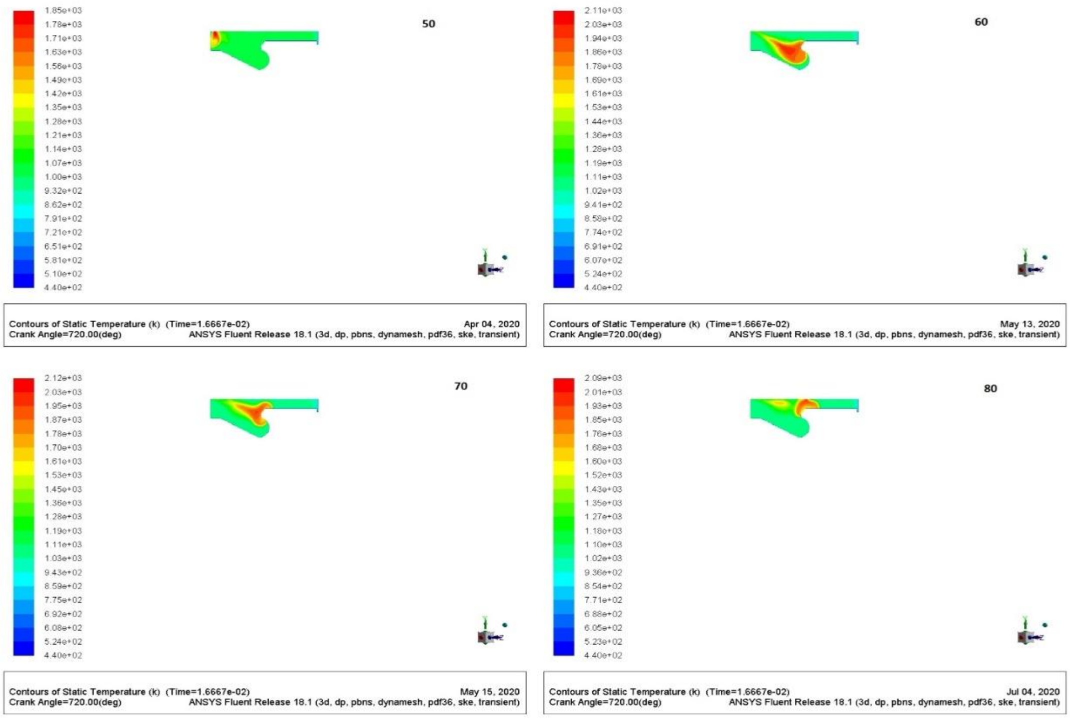
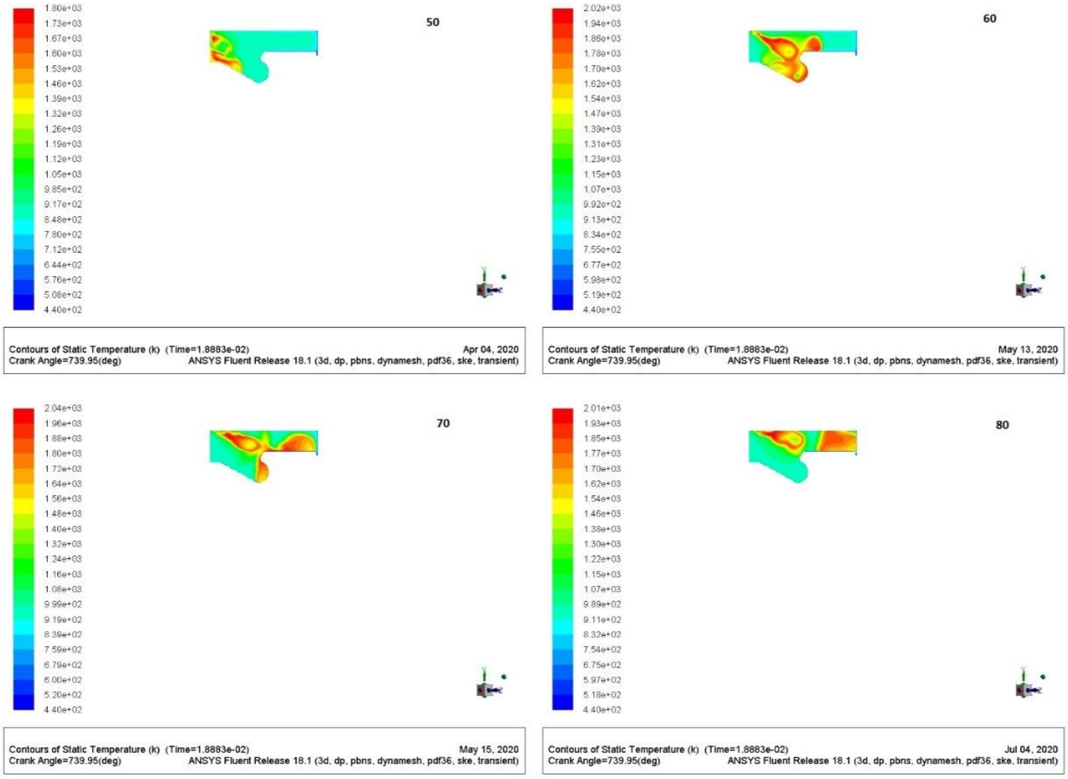



Figure 21 Temperature contours at 720° CA



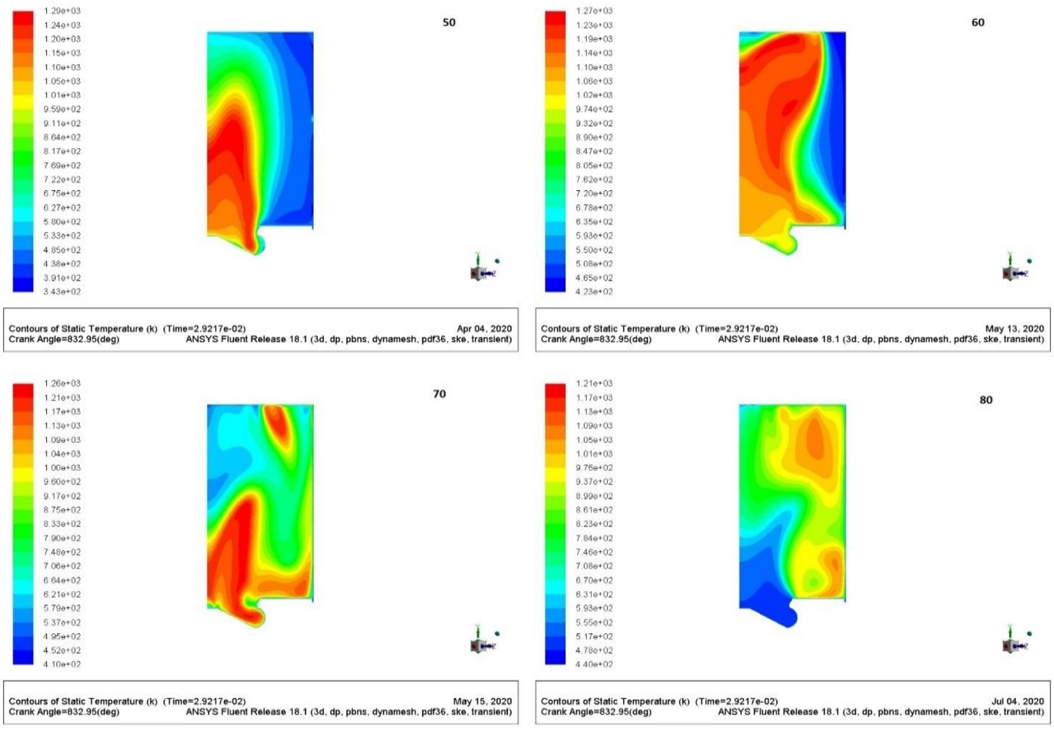




Figure 22 Temperature contours at 832.95° CA

In figure 14 All the contours are same because at that point (570° CA) calculation start. In the figure 15 at 720° CA injection of fuel start there is increase in temperature when spray angle increase from 50° to 60° and increases till 80°. In figure 16 at 732.95 CA Injection of fuel end. There is clearly shown better combustion is at 70°. In figure 16 Temperature contours at 832.95° (end of calculation). At 50° combustion occur on Right side and left side have low temperature. At 60° combustion is better than 50° but still one side of cylinder have less temperature. Maximum Temperature with 50°, 60°, 70° and 80° are 1800K, 2100K, 2400K, and 2200K, respectively.

## 5. Particle traces of injection

Particle traces at spray angle of 50°, 60°, 70° and 80° are shown in figure 17. Particle traces are colored by temperature. We can clearly see that at 80° flow is fully developed as compared to other.

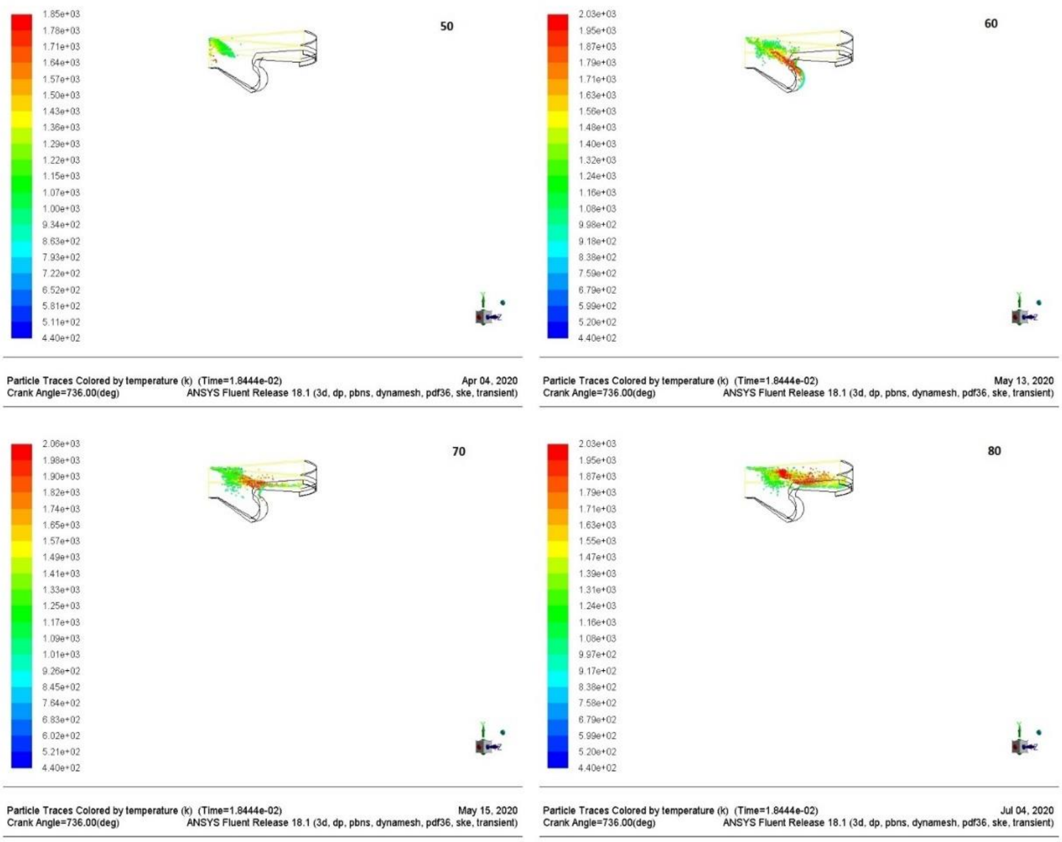



Figure 23 particle traces of injection colored by temperature

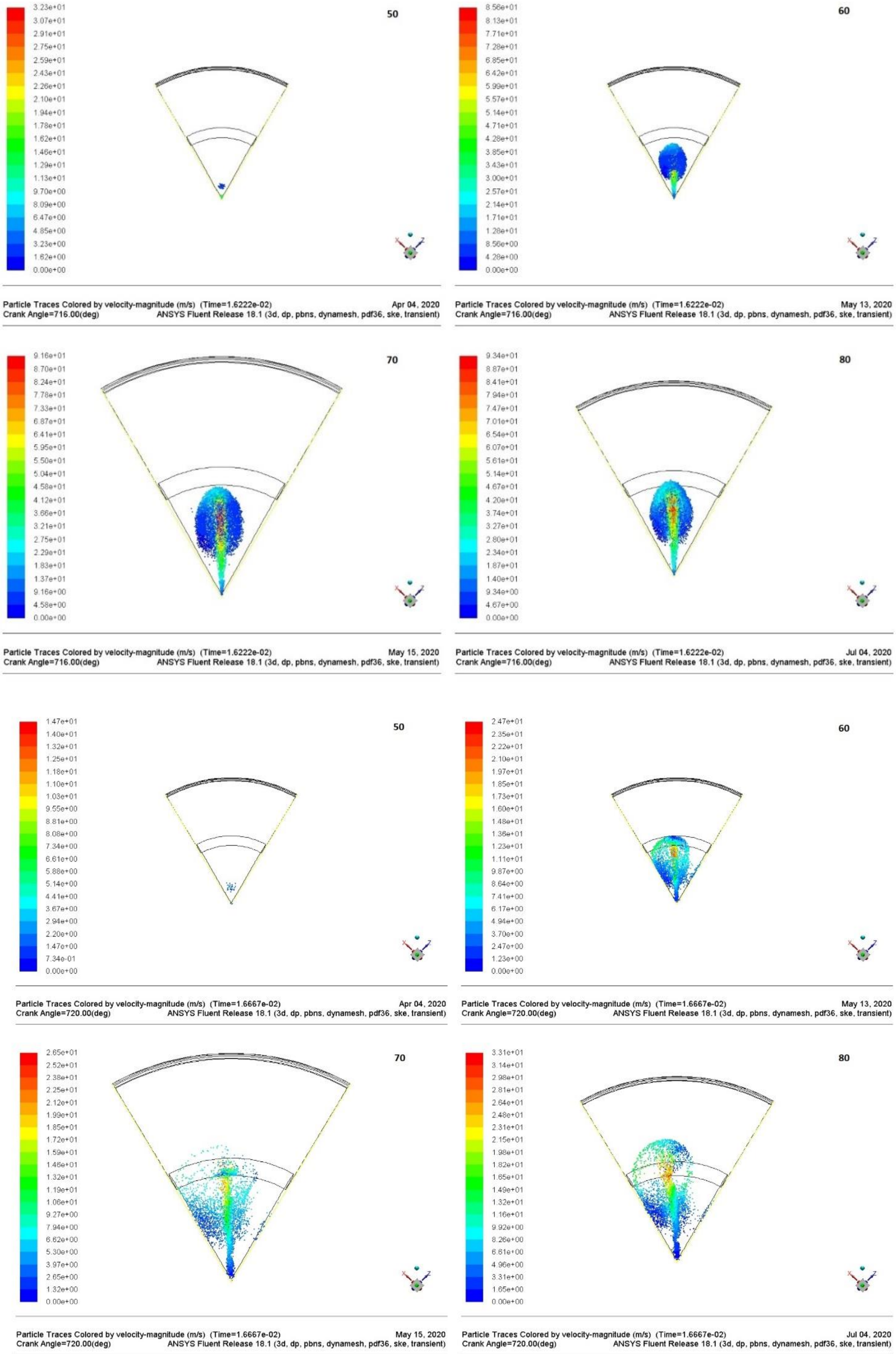



Figure 24 particle traces colored by velocity

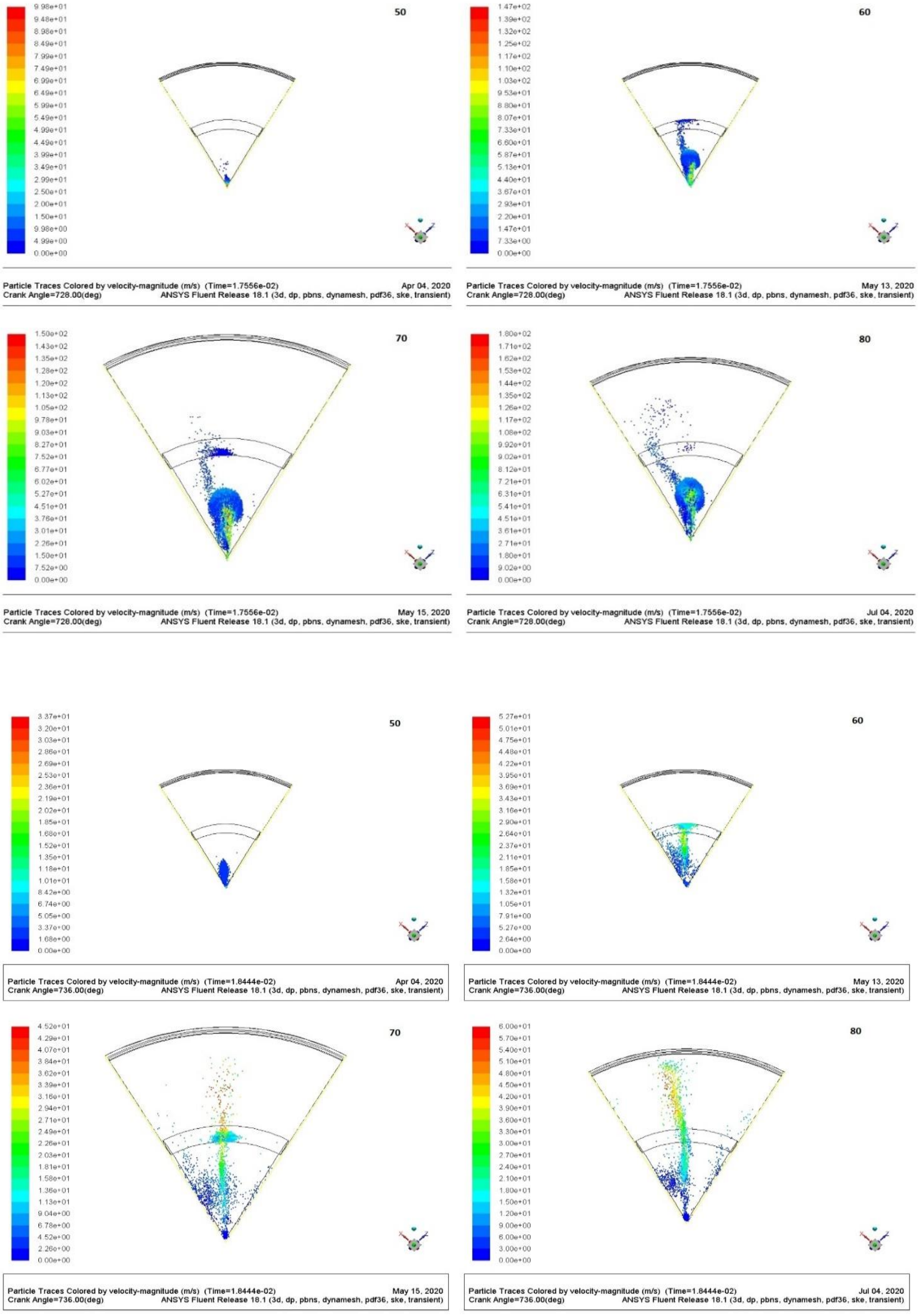



Figure 24 Particle traces of velocity

Apparent heat release rate at all four cases are shown in figures 20 Maximum AHRR is at 80-degree spray angle. At the 50-degree spray angle maximum heat release rate is only 10. While increasing the spray angle peak of the heat release rate is increased.

There are three peaks for the 60 and 70 degree. First two peaks are almost like each other but in third peak, it has higher value of heat release. Pressure inside in cylinder start increasing when the spray angle is increased from 50° to 70°, after the 70° further increase in spray angle causes to slightly decrease the pressure inside the cylinder. Results show that to optimize the overall efficiency of engine, spray angle and the shape of the combustion chamber needs to optimize together. Pressure shows in figure 22.

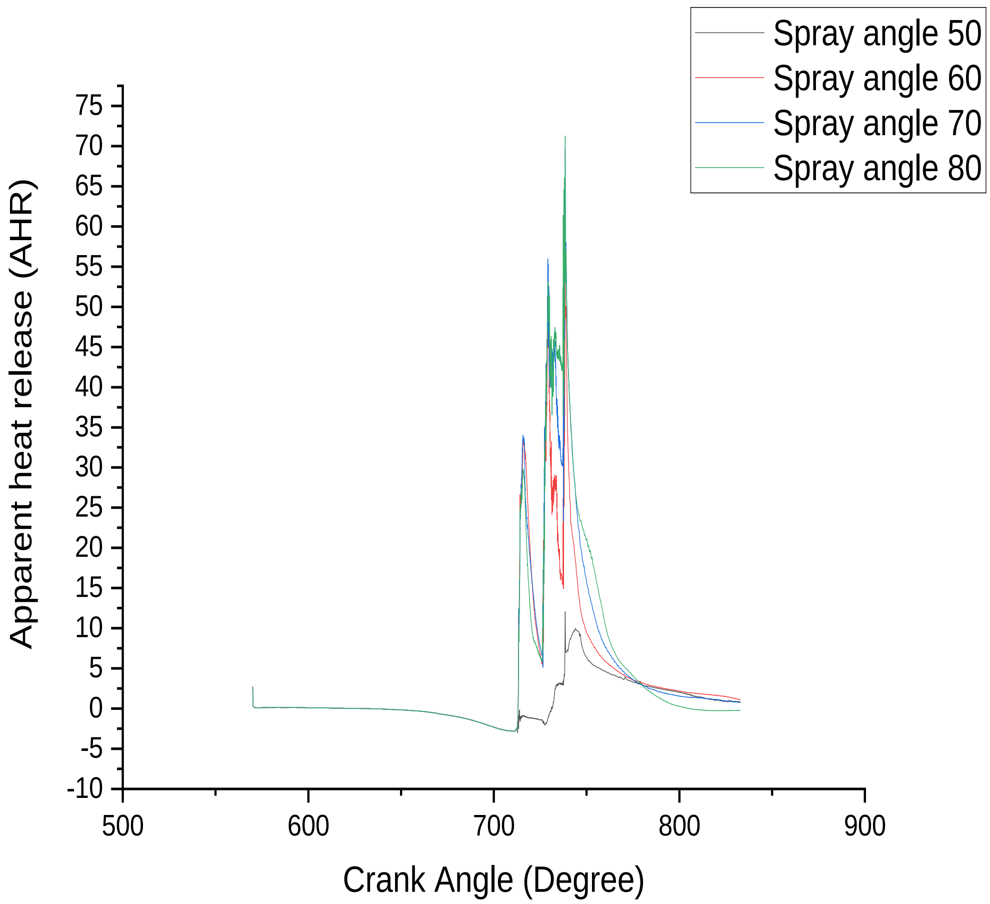



Figure 25 Apparent Heat Release Rate vs Crank Angle

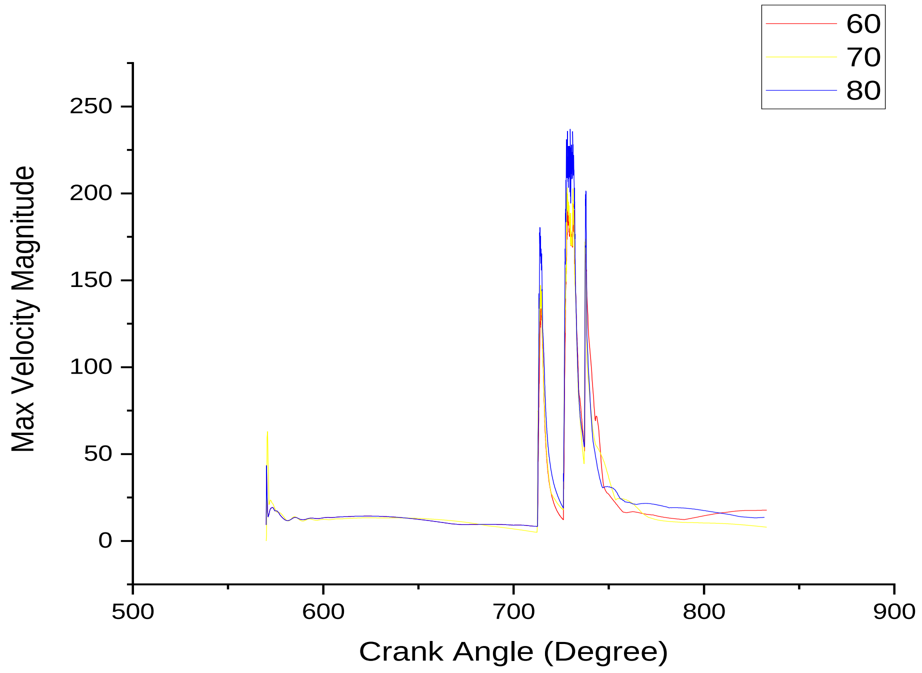



Figure 26 Maximum velocity vs Crank Angle

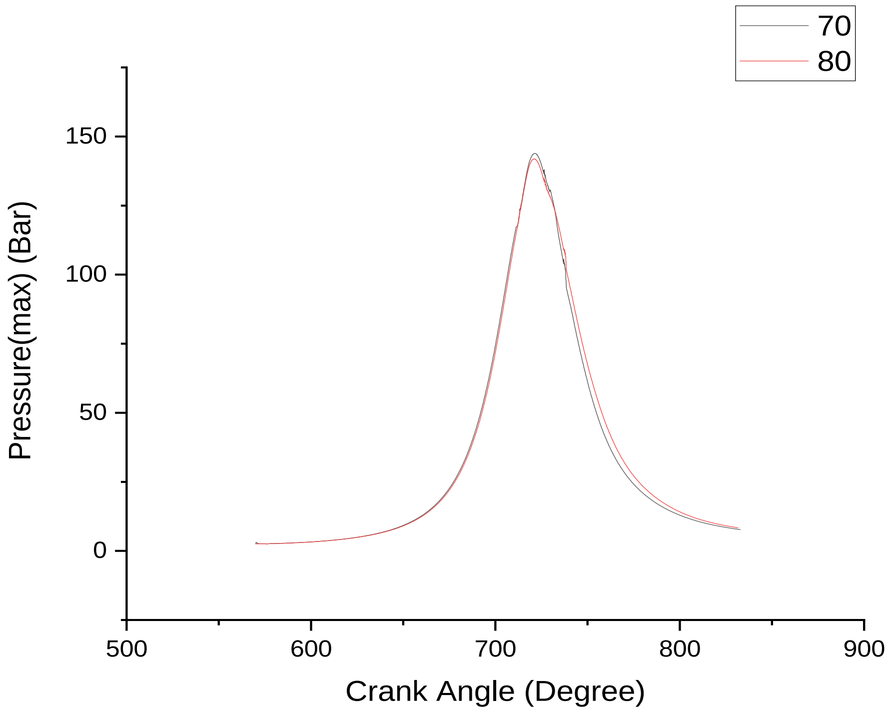



Figure 27 Maximum pressure vs Crank Angle

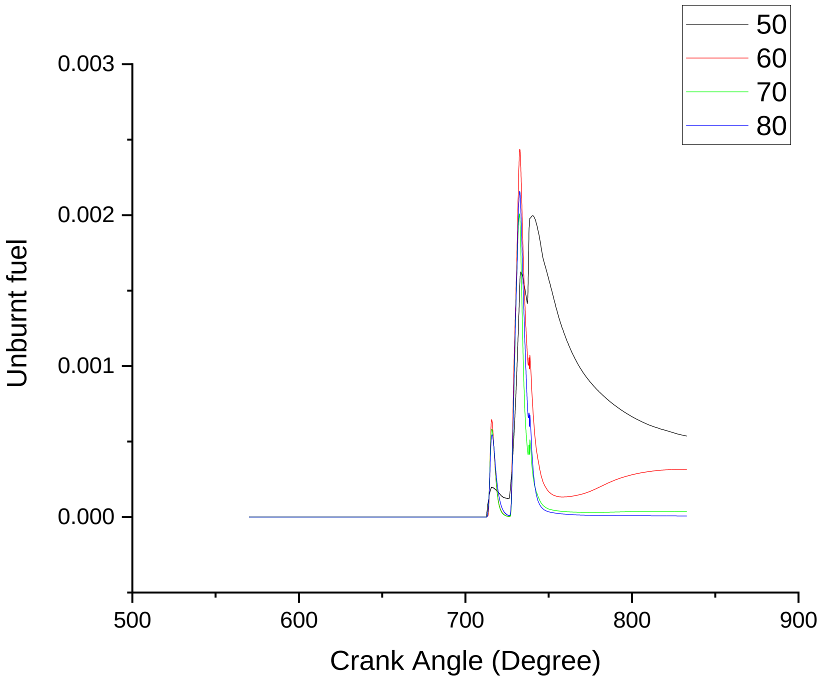



Figure 28 Unburnt fuel Vs Crank Angle


Figure 29 Penetration Length VS Crank Angle

## 5.1 Conclusions

In this work, the effect of injection spray angle on the combustion and overall diesel engine performance was investigated using computational fluid dynamics. Data used for the model is collected from literature and the CFD model is build based on that data. By analysis of simulation results following conclusions are drawn.

1) The pressure inside the in-cylinder starts increasing when the spray angle is increased from 50° to 70°. After the 70° further increase in spray angle causes to slightly decrease the pressure inside the cylinder. Results show that to optimize an engine's overall efficiency, spray angle and the combustion chamber's shape need to optimize together.
2) With an increase in the spray angle from 50° to 80°, the cylinder's temperature rises.
3) At 80°, penetration of fuel is high as compare to other cases. Penetration length increases when the spray angle increases from 50° to 80°.
4) Results show that the lowest unburnt fuel is at a spray angle of 70°. Other cases show a high-unburnt fuel value than 70°.

# References


[1] T. Cerri, A. Onorati, and E. Mattarelli, "1D engine simulation of a small HSDI diesel engine applying a predictive combustion model," *J. Eng. Gas Turbines Power*, vol. 130, no. 1, 2008, doi: 10.1115/1.2747258.

[2] J. Liu, H. Zhao, J. Wang, and N. Zhang, "Optimization of the injection parameters of a diesel/natural gas dual fuel engine with multi-objective evolutionary algorithms," *Appl. Therm. Eng.*, vol. 150, no. November 2018, pp. 70–79, 2019, doi: 10.1016/j.applthermaleng.2018.12.171.

[3] D. T. Hountalas and A. D. Kouremenos, "Development of a fast and simple simulation model for the fuel injection system of diesel engines," *Adv. Eng. Softw.*, vol. 29, no. 1, pp. 13–28, 1998, doi: 10.1016/S0965-9978(97)00042-2.

[4] P. Olmeda, J. Martin, A. Garcia, D. Villalta, A. Warey, and V. Domenech, "A Combination of Swirl Ratio and Injection Strategy to Increase Engine Efficiency," *SAE Int. J. Engines*, vol. 10, no. 3, pp. 1–13, 2017, doi: 10.4271/2017-01-0722.

[5] M. Zhao, M. Wei, P. Song, Z. Liu, and G. Tian, "Performance evaluation of a diesel engine integrated with ORC system," *Appl. Therm. Eng.*, vol. 115, no. 5, pp. 221–228, 2017, doi: 10.1016/j.applthermaleng.2016.12.065.

[6] B. W. Davis, J. T. Diep, and S. Jose, "( 12 ) United States Patent," vol. 2, no. 12, 2017.

[7] C. V. Ngayihi Abbe, R. Nzengwa, R. Danwe, Z. M. Ayissi, and M. Obonou, "A study on the 0D phenomenological model for diesel engine simulation: Application to combustion of Neem methyl esther biodiesel," *Energy Convers. Manag.*, vol. 89, pp. 568–576, 2015, doi: 10.1016/j.enconman.2014.10.005.

[8] E. A. Yfantis, J. S. Katsanis, E. G. Pariotis, T. C. Zannis, and R. G. Papagiannakis, "First-Law and Second-Law Waste Heat Recovery Analysis of a Four-Stroke Marine Diesel Engine Equipped with a Regenerative Organic Rankine Cycle System," *5th Int. Symp. Sh. Oper. Manag.*, no. May, pp. 1–11,

2015.

[9] H. Sapra, M. Godjevac, K. Visser, D. Stapersma, and C. Dijkstra, "Experimental and simulation-based investigations of marine diesel engine performance against static back pressure," *Appl. Energy*, vol. 204, pp. 78–92, 2017, doi: 10.1016/j.apenergy.2017.06.111.

[10] V. Aesoy and E. Pedersen, "Modeling and Simulation for Design and Testing of Direct Injection Gaseous Fuel Systems for Medium-Speed Engines," *SAE Int. J. Fuels Lubr.*, vol. 4, no. 2, pp. 188–203, 2011, doi: 10.4271/2011-01-2401.

[11] M. Y. Anisimov, S. S. Kayukov, A. A. Gorshkalev, A. V. Belousov, R. E. Gallyamov, and Y. D. Lysenko, "Developing the Model of Fuel Injection Process Efficiency Analysis for Injector for Diesel Engines," *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 302, no. 1, 2018, doi: 10.1088/1757-899X/302/1/012055.

[12] M. Talbi and B. Agnew, "Energy recovery from diesel engine exhaust gases for performance enhancement and air conditioning," *Appl. Therm. Eng.*, vol. 22, no. 6, pp. 693–702, 2002, doi: 10.1016/S1359-4311(01)00120-X.

[13] J. Oefelein, R. Dahms, and G. Lacaze, "Detailed modeling and simulation of high-pressure fuel injection processes in diesel engines," *SAE Int. J. Engines*, no. March, 2012, doi: 10.4271/2012-01-1258.

[14] G. Derrico, T. Lucchini, F. Atzler, and R. Rotondi, "Computational fluid dynamics simulation of diesel engines with sophisticated injection strategies for in-cylinder pollutant controls," *Energy and Fuels*, vol. 26, no. 7, pp. 4212–4223, 2012, doi: 10.1021/ef3004784.

[15] Z. S. Filipi and D. N. Assanis, "A nonlinear, transient, single-cylinder diesel engine simulation for predictions of instantaneous engine speed and torque," *J. Eng. Gas Turbines Power*, vol. 123, no. 4, pp. 951–959, 2001, doi: 10.1115/1.1365122.

[16] M. G. Shatrov, V. I. Malchuk, S. D. Skorodelov, A. Y. Dunin, V. V. Sinyavski, and A. L. Yakovenko, "Simulation of fuel injection through a nozzle having different position of the spray holes," *Period. Eng. Nat. Sci.*, vol. 7, no. 1, pp. 458–464, 2019, doi: 10.21533/pen.v7i1.403.

[17] V. V. Sinyavski, M. G. Shatrov, A. Y. Dunin, I. G. Shishlov, and A. V. Vakulenko, "Results of Simulation and Experimental Research of Automobile Gas Diesel Engine," *2019 Syst. Signals Gener. Process. F. Board Commun. SOSG 2019*, vol. 7, no. 1, pp. 281–286, 2019, doi: 10.1109/SOSG.2019.8706756.

[18] F. Payri, J. Benajes, X. Margot, and A. Gil, "CFD modeling of the in-cylinder flow in direct-injection Diesel engines," *Comput. Fluids*, vol. 33, no. 8, pp. 995–1021, 2004, doi: 10.1016/j.compfluid.2003.09.003.

[19] A. Praptijanto, A. Muharam, A. Nur, and Y. Putrasari, "Effect of ethanol percentage for diesel engine performance using virtual engine simulation tool," *Energy Procedia*, vol. 68, pp. 345–354, 2015, doi: 10.1016/j.egypro.2015.03.265.

[20] K. K. Yum, B. Taskar, E. Pedersen, and S. Steen, "Simulation of a two-stroke diesel engine for propulsion in waves," *Int. J. Nav. Archit. Ocean Eng.*, vol. 9, no. 4, pp. 351–372, 2017, doi: 10.1016/j.ijnaoe.2016.08.004.

[21] G. Prabhakara Rao, V. R. K. Raju, and S. Srinivasa Rao, "Effect of Fuel Injection Pressure and Spray Cone Angle in di Diesel Engine Using CONVERGETM CFD Code," *Procedia Eng.*, vol. 127, pp. 295–300, 2015, doi: 10.1016/j.proeng.2015.11.372.

[22] R. A. Bakar, Semin, and A. R. Ismail, "Investigation of diesel engine performance based on simulation," *Am. J. Appl. Sci.*, vol. 5, no. 6, pp. 610–617, 2008, doi: 10.3844/ajassp.2008.610.617.

[23] M. Jafari, M. J. Parhizkar, E. Amani, and H. Naderan, "Inclusion of entropy generation minimization in multi-objective CFD optimization of diesel engines," *Energy*, vol. 114, pp. 526–541, 2016, doi: 10.1016/j.energy.2016.08.026.

[24] J. Shu *et al.*, "Effects of injector spray angle on combustion and emissions characteristics of a natural gas (NG)-diesel dual fuel engine based on CFD coupled with reduced chemical kinetic model," *Appl. Energy*, vol. 233–234, no. September 2018, pp. 182–195, 2019, doi: 10.1016/j.apenergy.2018.10.040.

[25] F. Baldi, G. Theotokatos, and K. Andersson, “Development of a combined mean value-zero dimensional model and application for a large marine four-stroke Diesel engine simulation,” *Appl. Energy*, vol. 154, pp. 402–415, 2015, doi: 10.1016/j.apenergy.2015.05.024.

[26] H. Omidvarborna, A. Kumar, and D. S. Kim, “Recent studies on soot modeling for diesel combustion,” *Renew. Sustain. Energy Rev.*, vol. 48, pp. 635–647, 2015, doi: 10.1016/j.rser.2015.04.019.

[27] H. Mohamed Ismail, H. K. Ng, and S. Gan, “Evaluation of non-premixed combustion and fuel spray models for in-cylinder diesel engine simulation,” *Appl. Energy*, vol. 90, no. 1, pp. 271–279, 2012, doi: 10.1016/j.apenergy.2010.12.075.

[28] Z. Liu, Q. Zuo, G. Wu, and Y. Li, “An artificial neural network developed for predicting of performance and emissions of a spark ignition engine fueled with butanol–gasoline blends,” *Adv. Mech. Eng.*, vol. 10, no. 1, pp. 1–11, 2018, doi: 10.1177/1687814017748438.

[29] E. Neshat, D. Honnery, and R. K. Saray, “Multi-zone model for diesel engine simulation based on chemical kinetics mechanism,” *Appl. Therm. Eng.*, vol. 121, no. April, pp. 351–360, 2017, doi: 10.1016/j.applthermaleng.2017.04.090.

[30] R. Mobasheri, “Influence of narrow fuel spray angle and split injection strategies on combustion efficiency and engine performance in a common rail direct injection diesel engine,” *Int. J. Spray Combust. Dyn.*, vol. 9, no. 1, pp. 71–81, 2017, doi: 10.1177/1756827716651514.

[31] E. Akbarian, B. Najafi, M. Jafari, S. F. Ardabili, S. Shamshirband, and K. W. Chau, “Experimental and computational fluid dynamics-based numerical simulation of using natural gas in a dual-fueled diesel engine,” *Eng. Appl. Comput. Fluid Mech.*, vol. 12, no. 1, pp. 517–534, 2018, doi: 10.1080/19942060.2018.1472670.

[32] C. D. Rakopoulos and E. G. Giakoumis, “Sensitivity analysis of transient diesel engine simulation,” *Proc. Inst. Mech. Eng. Part D J. Automob. Eng.*, vol. 220, no. 1, pp. 89–101, 2006, doi: 10.1243/095440705X69641.

[33] K. Qi, L. Feng, X. Leng, B. Du, and W. Long, "Simulation of quasi-dimensional combustion model for predicting diesel engine performance," *Appl. Math. Model.*, vol. 35, no. 2, pp. 930–940, 2011, doi: 10.1016/j.apm.2010.07.047.

[34] C. O. Katsanos, D. T. Hountalas, and T. C. Zannis, "Simulation of a heavy-duty diesel engine with electrical turbocompounding system using operating charts for turbocharger components and power turbine," *Energy Convers. Manag.*, vol. 76, pp. 712–724, 2013, doi: 10.1016/j.enconman.2013.08.022.

[35] S. Kook, C. Bae, P. C. Miles, D. Choi, M. Bergin, and R. D. Reitz, "The effect of swirl ratio and fuel injection parameters on CO emission and fuel conversion efficiency for high-dilution, low-temperature combustion in an automotive diesel engine," *SAE Tech. Pap.*, vol. 2006, no. 724, 2006, doi: 10.4271/2006-01-0197.

[36] V. Mrzljak, V. Medica, and O. Bukovac, "Simulation of a two-stroke slow speed diesel engine using a quasi-dimensional model," *Trans. Famena*, vol. 40, no. 2, pp. 35–44, 2016, doi: 10.21278/TOF.40203.

[37] Z. F. Tian and J. Abraham, "Development of a two-dimensional internal combustion engines model using CFD for education purpose," *Proc. - 20th Int. Congr. Model. Simulation, MODSIM 2013*, no. December, pp. 1575–1581, 2013, doi: 10.36334/modsim.2013.g6.tian.