A REVIEW ON CFD ANALYSIS OF FUEL INJECTOR WITH PRESSURE

[1] Mr. B. Phanindra Kumar, [2] B. Sai Kishore, [3] A. Vaishnavi, [4] D. Manisha, [5] K. Sravan Sai

[1] Assistant professor, Dept of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, India

[2] B. E U. G (IV year), Dept of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, India

[3] B. E U. G (IV year), Dept of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, India

[4] B. E U. G (IV year), Dept of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, India

[5] B. E U. G (IV year), Dept of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, India

Abstract: Fuel injection is a critical process in internal combustion engines, as it plays a key role in determining engine performance and emissions. The effectiveness of diesel engine depends greatly on the fuel injector nozzle. The size and geometry of the fuel injector affect the combustion characteristics of the diesel engine as well as the performance over the lifetime of the engine and the mechanical durability of the injector. So, in this paper a computational fluid dynamics (CFD) simulation of fuel injection with pressure analysis was performed. The aim of the simulation was to investigate the fuel injection process and its effect on the pressure distribution in the combustion chamber. CATIA software is used to design the fuel injector model. The simulation was carried out using ANSYS Fluent software, and the results were analysed to determine the fuel spray characteristics and pressure distribution during the injection process.

Keywords: Fuel Injector, Computational fluid dynamics (CFD), Combustion characteristics, CATIA, ANSYS Fluent

1.INTRODUCTION:

Diesel engine is well known for its higher efficiency and fuel economy as compared to any other type of internal combustion engines. Fuel injected in a diesel engine has to mix with air and form a combustible mixture. The combustion process in a diesel engine is an exceedingly complex phenomenon. In diesel engines the complexity of the combustion process is compounded by the non-uniform fuel distribution within the combustion chamber and the fuel–air mixing process. Internal combustion engines use fuel injection systems to inject fuel into the combustion chamber. Before ignition, the fuel is atomized and combined with air, resulting in more efficient combustion and enhanced engine performance. The fuel injection system consists of a fuel injector, which sprays gasoline into the combustion chamber, and a fuel delivery system, which supplies fuel to the injector.

1.1 Direct and Indirect duel injection system

There are mainly two types of fuel injection systems. They are,

- Direct fuel injection system
- Indirect fuel injection system

Unlike traditional petrol engines, which are also known as spark-ignition (SI) engines with carburettors, both diesel engines have fuel injection systems. Injection methods are classified as either Direct Injection (DI) or Indirect Injection (IDI). Direct injection refers to all systems in which fuel is injected directly into the combustion chamber, as the name indicates. In IDI engines, a relatively rich fuel combination is first lit in a small pre-chamber or turbulence chamber, and this burning mixture then goes into the main combustion chamber, where it mixes with the remaining compressed air and burns it very effectively [1].

In a direct injection diesel engine, combustion begins at the dynamic injection point and is divided into two different phases: the ignition delay and the heat release period. The ignition delay is the time between the actual dynamic injection point and the ignition start. The ignition delay is determined by factors such as cylinder pressure, temperature, and fuel chemistry. The ignition delay in DI diesel engines is determined by the compression ratio, swirl, and engine speed.

The heat release rate of the fuel creates a fluctuation in gas pressure and temperature within the engine cylinder, which has a significant impact on the engine's fuel economy, power output, and emissions. It provides a good understanding of the combustion process that occurs in the engine. When the injection timing is advanced, the smoke level decreases. However, with excessively advanced injection timings, the smoke level rises due to a decrease in brake thermal efficiency, resulting in more fuel intake at any given output efficiency [2].

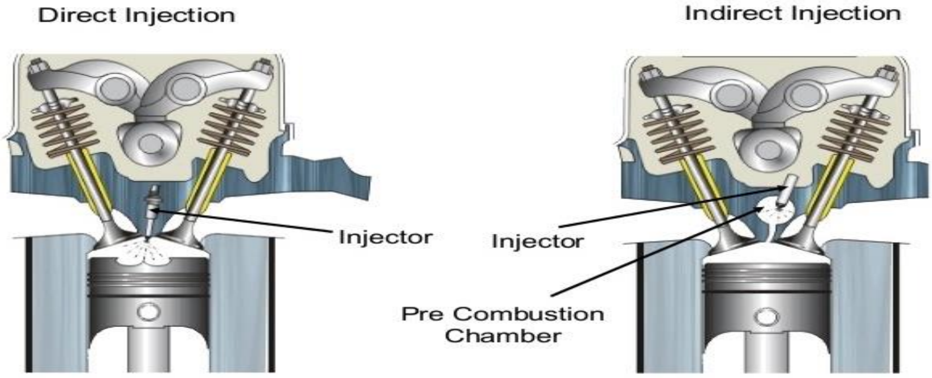

Figure 1: Direct and indirect fuel injector

1.2 Types of fuel diesel engine fuel injection systems

There are several types of fuel injection systems used in diesel engines. The choice of fuel injection system depends on factors such as engine design, performance requirements, emissions regulations, and cost considerations. Here are some commonly used types of diesel engine fuel injection systems:

- Single fuel injection system
- Multi fuel injection system
- Common rail fuel injection system
- Sequential fuel injection system

1.3 Computational Fluid Dynamics (CFD)

Computational fluid dynamics is the branch of fluid mechanics that involves using numerical methods and algorithms to simulate and analyse the behaviour of fluid flow.CFD simulations can predict and visualize fluid flow patterns, pressure distributions, temperature profiles, turbulence characteristics and other important parameters. These predictions normally occur under those conditions defined in terms of flow geometry, the physical properties of a fluid, boundary conditions and initial conditions of flow field. Such predictions generally concern sets of values of the flow variables, for example, velocity, pressure, or temperature at selected locations in the domain and for selected times. The predictions may also involve evaluations of overall flow behaviour, such as the flow rate or the hydrodynamic force acting on an object in the flow.

Computational Fluid Dynamics (CFD) has become an essential too in the design of the fuel injections systems in internal combustion engines. It allows for a better performance and reduced emissions.CFD simulations involve dividing the fluid domain into a grid or mesh of discrete elements. The properties of the fluid such as velocity, pressure, and temperature are calculated at each point in the mesh using the numerical algorithms. [3].

By using CFD software, it has the capability to evaluate and display fluid quantities such as:

- Velocity, the speed and direction of particles inside or outside the model.
- Temperature
- Pressure

These results can be calculated and displayed at specific locations in a model for the maximum or minimum values on a surface or a component. When displayed in the fluid the results can be depicted as color contours, particles, a direction field or streamlines.

1.4 Applications of CFD Analysis

CFD analysis has numerous applications across various industries. Some key applications of CFD analysis include:

- **Aerospace:** CFD is extensively used in the aerospace industry for aerodynamics analysis, aircraft components and propulsion system which helps to improve aircraft performance and fuel; stability.
- **Automotive Industry:** CFD is used for aerodynamic analysis of vehicles to enhance fuel efficiency, and improve vehicle stability.
- **Energy and Power Generation**: CFD plays a crucial role in the design and analysis of energy systems, such as wind turbines, gas turbines. It helps optimize the efficiency and performance of these systems by analysing fluid flow, heat transfer, and combustion processes.
- **Chemical and Process Industry:** CFD is used to analyse and optimize processes in the chemical and process industry which helps improve process efficiency, reduce energy consumption, and optimize equipment design.
- **HVAC and Building Ventilation**: CFD is used to analyse airflow, thermal comfort, and pollutant distribution in HVAC systems and buildings. It helps in optimizing air distribution, reducing energy consumption, and improving indoor air quality.

1.5 Advantages of CFD analysis

CFD analysis offers several advantages in the field of engineering and design. Some of the key advantages of CFD analysis include:

- **Cost-Effectiveness:** CFD analysis allows engineers to evaluate and optimize designs virtually, reducing the need for expensive physical prototypes and testing.
- **Prediction of Performance Parameters:** CFD simulations allow for accurate prediction of performance parameters such as pressure distribution, temperature profiles, velocity distribution, and turbulence characteristics. This information helps engineers assess the performance of the design and make necessary adjustments.
- **Improved Design Insight:** CFD provides detailed insights into fluid flow behaviour, heat transfer, and other related phenomena. It helps engineers understand complex flow patterns, identify potential issues, and make informed design decisions.
- **Faster Design Iterations:** With CFD analysis, engineers can quickly iterate and refine designs. They can evaluate multiple design options, make modifications, and re-run simulations in a relatively short time frame, accelerating the design process.
- **Visualization of Flow Phenomena:** CFD analysis provides visual representations of fluid flow behaviour. This allows engineers and stakeholders to better understand and communicate complex flow patterns and phenomena, aiding in design discussions and decision-making processes.

Overall, CFD analysis offers a range of benefits, from cost and time savings to improved design performance and enhanced understanding of fluid flow behaviour. It has become an essential tool in engineering design, optimization and decision making process.

2. LITERATURE SURVEY

In 2016, F.J. Salvador, D. Jaramillo, J.-V. Romero, M.-D.Rosello [4]. In this paper, the behaviour of internal nozzle flow and cavitation phenomena was numerically analysed in this article for non-conventional Diesel convergent-divergent nozzles in order to identify their flow characteristics. The degree of convergence divergence of the orifices varies amongst nozzles, but they always have the same diameter in the center of the nozzle orifice.

In 2015, Pathak, V. K., & Gupta. S [5]. The aim of this paper is by changing the shape and fuels of the nozzle injector to investigate its performance. The geometry of a diesel injector orifice is significant in establishing the performance and emission characteristics of a diesel engine. Many studies are also being conducted to reduce the emissions created by diesel engines. Using CFD software, this article compares dimethyl ether (DME) to diesel fuel. As a result, improved fuel air mixing and atomization of the fuel are obtained, which improves engine performance. Because of its characteristics, dimethyl ether reduces soot, SOx, and NOx emissions significantly.

In 2016, V. Lazarev, G. Lomakin, E. Lazarev [6]. The perfection of the diesel engine performance parameters is taken into account as a result of the increase in rail pressure and the modernization of the nozzle Tribo systems have a high (up to 300 MPa) fuel pressure value. The updated configuration nozzle and extra (bottom) precision guiding interface are used and hydrodynamic injection parameters are evaluated. Computational fluid-dynamic (CFD) modelling is used to estimate hydrodynamic fuel flow and force distribution parameters within the "needle-nozzle body" framework. The outcomes of the injection modelling are defined and the parameters for the changed design of nozzle precision interfaces are communicated. The ways of accelerating the steadiness of needle position within the nozzle body with perfection of parameters of fuel injection system are presented.

In 2014, Sanghoon Lee, Sungwook Park [7]. This paper analyses, as a comparison, the spray features of a group-hole nozzle as opposed to the characteristics of a single-hole nozzle in terms of spray operation and atomization phase. Phase Doppler Particle

Analysers (PDPA) and spray visualization experiments have been performed using a customizable GDI injector that can adopt a particular nozzle shape under free spray conditions.

P. Parthiban, M. Robert Sagayadoss, T. Ambikapathi [8]. A rocket nozzle is a mechanical device which is designed to control the rate of flow, speed, direction and pressure of stream that exhaust through it. There are various types of rocket nozzles which are used depending upon the mission of the rocket. This paper contains analysis over a convergent divergent rocket nozzle which is performed by varying the number of divisions in mesh. Also the various contours of nozzle like Cell Equiangular skew, Cell Reynolds number, Pressure, Velocity, Mach Number, and above are calculated at each type of mesh using CFD analysis software ANSYS Fluent.

A. Jhonann (2013) [9]. In this they have developed a model based on the Hiroyasu's multi zone combustion model. Nozzle injection (spray) parameters, induction swirl, air and fuel composition was considered in the model. Sub models pertaining to zone velocity, air entrainment rate, droplet evaporation rate, combustion rate etc., were taken from the latest literature. The model simulation was used to extract the parameters like cylinder pressure, heat release rate and emissions (NOx and soot). The predicted results of zone velocity and spray tip penetration are compared with the predictions that are reported by Hiroyasu. The predicted results showed good agreement with the experimental data.

Mobasheri, R. & Peng Z. (2013) [10]. A modified parameter named ‘‘Homogeneity Factor (HF) of in-cylinder charge’’ has been introduced as a new measure for supporting the understanding of the air–fuel mixing and combustion process in diesel engines. The effects of this parameter on engine performance and pollutant emissions have been studied using various EGR and split injection schemes on a DI diesel engine. In this study, twelve different injection strategies for which two injection pulses with different fuel amount for each pulse (up to 30% for the second pulse) and different separation between two pulses (up to 30 CA) were evaluated using CFD modelling. Results show the higher Homogeneity Factor will result in higher rate of air–fuel mixing and more complete combustion process. It was found when the dwell delay between injection pulses becomes longer, as it leaves more time for the air–fuel mixing during the late compression stroke, the maximum amount of Homogeneity Factor will be obtained at a later stage of combustion process and it can cause a reduction of NOx formation. This suggests, by optimizing split injection configurations accompanied with appropriate EGR rate, pollutant emissions can be decreased and controlled more accurately.

Tanner, F. X., & Srinivasan, S. (2009) [11]. A computationally efficient computational fluid dynamics (CFD)-based optimization method with the capability of finding optimal engine operating conditions with respect to emissions and fuel consumption has been developed. The application of this optimization tool is demonstrated for a non-road, medium-speed DI diesel engine which, for these simulations, utilizes a multi-orifice, asynchronous injection system. It has been demonstrated that this new injection method has a large potential for reducing emissions while maintaining a low fuel consumption. In addition, this optimization approach is computationally very efficient when good enough initial values are available.

Vijayakumar, T., Raj, R. T. K., & Nanthagopal, K. (2011) [12]. The spray characteristics of the diesel fuel are greatly affected by the cavitation formed inside the injector due to the high pressure differential across the nozzle. Many researchers are exploring the potential of using diethyl ether and dimethyl ether as an alternate for diesel fuel to meet the strict emission norms. Due to the variation in the fuel properties the internal flow characteristics in injectors for ether fuels are expected to be different from that of the diesel fuel. In this paper the numerical investigation is used to study and compare the internal flow characteristics of diethyl ether, dimethyl ether and diesel fuel. The two phase flow model considering the fuel as a mixture of liquid and vapour is adopted for the simulation study. The injection pressure is varied from 100 bar to 400 bar and the flow characteristics of all three fuels are simulated and compared. Results indicate that all three fuels have distinct cavitation patterns owing to different property values. The dimethyl ether is found to be more cavitation than diesel and die-ethyl ether fuels as expected. The mass of fuel injected are found to be decreasing for the ether fuels when compared with diesel fuel at all injection pressures.

Based on the literature review of CFD analysis on fuel injectors in different we came to know that there are several findings and observations. In order to improve fuel atomization, reduce emissions and increase engine efficiency, computational fluid dynamics simulations have been applied for the optimizing injector designs and operating conditions. CFD models have been used successfully to simulate fuel injection processes to understand how injector design, injection parameters, and operating circumstances affect fuel spray characteristics.

CFD analysis of fuel injectors under pressure provides useful information about fuel spray parameters, combustion performance, and emission formation. Simulations aid in the optimization of injector designs, operating conditions, and injection techniques in order to improve engine performance, fuel efficiency, and emissions control.

3.ACKNOWLEDGEMENT

We are delighted to present the initial project report on 'CFD Analysis of fuel injector with pressure'. Our heartfelt thanks go to our internal guide for their unwavering assistance and guidance throughout the project. We are truly appreciative of their valuable suggestions, which proved to be extremely beneficial. We would also like to express our gratitude to the HOD, Dr. B. Vijaya Kumar, Guru Nanak Institute of Technology, Hyderabad, India, for their invaluable support and recommendations, which have been instrumental in the success of this project.

4. REFERENCES:

1. Kumar Tiwari, A., & Kumar Sahu, P. (2021). CFD Analysis of Fuel Injection System using ANSYS CFX. International Research Journal of Engineering and Technology.
2. Ramos JI. Internal combustion engine modeling. New York: Hemisphere Publishing Corporation; 1989.
3. Hu, H. H. (2012). Computational Fluid Dynamics. In Fluid Mechanics (pp. 421–472). Elsevier. https://doi.org/10.1016/B978-0-12-382100-3.10010-1
4. F.J. Salvador, D. Jaramillo, J.-V.Romero, M.-D.Rosello, "Using a homogeneous equilibrium model for the study of the inner nozzle flow and cavitation pattern in convergent–divergent nozzles of diesel injectors s", Journal of Computational and Applied Mathematics.
5. Pathak, V. K., & Gupta, S. (2015). Study of Nozzle Injector Performance Using CFD. *International Journal of Recent Advances in Mechanical Engineering*, *4*(3), 153–160. https://doi.org/10.14810/ijmech.2015.4312
6. V. Lazarev, G. Lomakin, E. Lazarev, "Modeling of Injection Parameters for Diesel Engine Injector Nozzles with the Additional Precision Guiding Interface", Procedia Engineering 150 (2016) 52 – 60
7. SanghoonLee ,Sungwook Park, "Spray atomization characteristics of a GDI injector equipped with a group-hole nozzle", Fuel 137 (2014) 50–59
8. P. Parthiban, M. Robert Sagayadoss, T. Ambikapathi, Design And Analysis Of Rocket Engine Nozzle by using CFD and Optimization of Nozzle parameters, International Journal of Engineering Research, Vol.3., Issue.5., 2015 (Sept.-Oct.).
9. A. Johann, Macroscopic and Microscopic Spray Pattern for High Pressure Common – Rail Diesel Injection, Journal of Society of Automotive Engineering of Japan. 67 (2013).
10. Mobasheri, R., & Peng, Z. (2013). CFD investigation into diesel fuel injection schemes with aid of Homogeneity Factor. *Computers and Fluids*, *77*, 12–23. https://doi.org/10.1016/j.compfluid.2013.02.013
11. Tanner, F. X., & Srinivasan, S. (2009). CFD-based optimization of fuel injection strategies in a diesel engine using an adaptive gradient method. *Applied Mathematical Modelling*, *33*(3), 1366–1385. https://doi.org/10.1016/j.apm.2008.01.023
12. Vijayakumar, T., Raj, R. T. K., & Nanthagopal, K. (2011). Effect of the injection pressure on the internal flow characteristics for diethyl and dimethyl ether and diesel fuel injectors. *Thermal Science*, *15*(4), 1123–1130. https://doi.org/10.2298/TSCI100717091V