**Engineering and Technology Journal e-ISSN: 2456-3358**

**Volume 08 Issue 03 March-2023, Page No.-2062-2068**

**DOI: 10.47191/etj/v8i3.10, I.F. – 7.136**



# CFD Analysis of Sector Combustion Simulation in Four Stroke Direct Injection Diesel Engine

**Samuel Tamrat[1], Venkata Ramayya[2], Ramesh Babu Nallamothu[3], Yared Seifu[4]**

[1,3,4]Department of Mechanical System and Vehicle Engineering, Adama science and technology university, Adama, Ethiopia

[2]Department of Mechanical Engineering, Jima University, Jima, Ethioipia

**ABSTRACT:** In this article an attempt has been made in cylinder sector combustion CFD simulation of four stroke petrol engines using ANSYS fluent 2019 to obtain very useful results using a modelled engine. In order to improve engine quality, understanding on sector combustion properties in engine cylinder is of major importance because combustion condition significantly influences engine efficiency, Capabilities in providing optimum combustion conditions which include high optimum swirl ratio, and optimum tumble ratio in respective engine can result in enhancement of engine power and comfort, along with reduction of fuel consumption, exhaust emission and noise. The simulation is applied for a geometry with crank radius (55mm), connecting rod length (165mm), Engine speed (1800rpm), which is developed to study the effect combustion inside the cylinder. Piston starts from bottom dead center about 541degrees and the maximum pressure reaches at 716 degrees. The pressure increases from 542 crank angle degree to 716 crank angle degree and reaches its maximum value at 716 crank angle degrees. The Maximum pressure at the end of compression is 63.69Kpa and temperature is 1200K. The maximum penetration length of injection is occurred at crank angle 722 degree is 0.014mm.

**KEYWORDS:** CFD, Combustion, Diesel Engine, Simulation.

## 1. INTRODUCTION

The combustion process in the diesel engine should be controlled to avoid both excessive maximum cylinder pressure and an excessive rate of pressure rise, in terms of crank angle. (Kurniawan et al., 2007). Computational Fluid Dynamics (CFD) has emerged as a useful tool in understanding the fluid dynamics of IC Engines for design purposes (Pathak Yogesh et al., 2014). The task of reducing pollution may be accomplished by improving the engine design to achieve complete and perfect combustion which is highly dependent on the uniformity of the air–fuel mixture. (Zahid & Syed, 2021). Large scale flow structures such as swirl and tumble increase the level of turbulence at the time of ignition which, in turn, strongly affect both pollutant emissions and fuel efficiency (El-Adawy et al., 2017).

The performance of an IC Engine depends upon complex interactions between mechanical, fluid, chemical, and electronic systems (Pathak Yogesh et al., 2014). Air swirl is generated by adopting suitable changes in the design of inlet port. Swirl velocity can be increases by adopting suitable changes in the combustion chamber design (Gugulothu & Reddy, 2016). Swirl and tumble flows are commonly characterized by a dimensionless parameter employed to quantify rotational and angular motion inside the cylinder, which are known as swirl and tumble ratios, respectively (Karunakar et al., 2016).

The production of high turbulence intensity is one of the most important factors for stabilizing the propagation of flame, especially in the case of lean-burn combustion (Kurniawan et al., 2007). The use of a bowl piston combustion chamber will result in substantial swirl amplification at the end of the compression stroke, which is important for the engine performance and emission characteristics (Erdil & Kodal, 2007).Computational Fluid Dynamics (CFD) simulations are of paramount importance to integrate experimental studies for an efficient optimization and design of new Diesel combustion systems(Millo et al., 2020). CFD allow designers to simulate and visualize the complex fluid dynamics with lower cost and lower analysis duration, providing that the simulation fulfil the optimized setup (Shafie et al., 2017). Fuel spray and atomization play a critical role in the performance and emission characteristics of a direct injection diesel engine (Raju & Rao, 2015). Rao et al,2015 also did CFD simulations using CONVERGE software to minimize NOx and Soot emissions, while maintaining the same efficiency of baseline configuration. (Rao et al., 2016).

Dhyani et al., 2014 developed for n-cylinder simulations using CONVERGE, a CFD software to simulate the in-cylinder processes in a direct injection diesel engine (Dhyani et al., 2014). The swirl chamber diesel engine has low NOx emission, low noise, and good high speed (Wei et al., 2013). For conventional direct-injection diesel engines, the fuel distribution of multi-hole spray in the combustion chamber is highly nonuniform (Londhekar et al., 2019). Beard et al., have conducted experiments on various bowl shapes such as flat, W-shaped and with or without re-entrant (Bérd et al., 1998). Krishna et.al deals with experimental investigation of the in-cylinder tumble flow in an engine with a flat piston at the engine speed of 1000rpm using particle image velocimetry. (Krishna et al., 2010).Stone et.al have tried to establish the significance of axial swirl in diesel engines, and proved that axial swirl is used in medium and small DI diesel engines for improving the fuel and air mixing rate. (Stone & Ladommatos, 1992). From the literature survey, it is clear that in the sector combustion is very much dependent on the shape of the combustion chamber and piston bowl configuration. However, there is a very limited study on the effect of diffusion flame on the sector combustion characteristics.

The objective of the present study is to develop a multi-dimensional sector combustion to investigate parameters like; magnitude of velocity, static pressure and static temperature, apparent heat release rate, swirl ratio, tumble ratio and cross tumble ratio at an engine speed of 1800 rpm at various crank angle during suction and compression stroke. Based on these it is concluded that still there is a gap and it is proposed to fill the gap to the extent possible. CFD sector combustion analysis complements testing and experimentation by reducing total effort and cost required for experimentation and data acquisition.

## 2. METHODOLOGY

The methodology used for sector combustion CFD simulation of internal combustion engine processes include the following major steps.

- Design piston geometry by using Catia.
- The generation of the computational meshes required to cover the sector combustion fluid domain over the crank-angle interval of interest.
- The specification of the initial and boundary conditions, of the flow solver settings and of the physical and chemical models adopted to simulate the governing sector combustion processes.
- The post-processing and interpretation of the simulation results of sector combustion. All tools required to cover the above tasks to successfully perform an IC-engine sector combustion CFD calculation are accessible via a fully interactive graphical user interface

The sector combustion decomposition crank angle is set to 570 ° from the IVC (Inlet Valve Closed) setting, which entered in the properties view of ICE cell. The inlet valve closing (IVC) angle is chosen as the geometry decomposition angle, since for combustion simulation in the power stroke of the engine cycle, starting from closing of valves to the end of the compression stroke. The parameters of sector combustion are summarized in table 1 below. In this simulation 4 valves used per cylinder (2 intake and 2 exhaust) valves. Diesel unsteady flamelet model under combustion model with 721 degrees for start of CA, temperature 366.7k, end of CA 742.5, cone angle degree 9, cone radius $1.27 \times 10^{-4}$ m and magnitude of velocity 468m/s, total flow rates $1.3333\times 10^{-5}$ inlet mass mass in kg/s and $2.54\times 10^{-4}$uniform diameter selected.

**Table 1. Input parameters for sector combustion CFD Model used**

| No | Property | Value | Unit |
|---|---|---|---|
| 1 | Simulation Type | Combustion Simulation | - |
| | Combustion Simulation Type | Sector Combustion Simulation | |
| 2 | Engine Inputs | | |
| | Connecting Rod Length | 165 | mm |
| | Crank Radius | 55 | mm |
| | Engine Speed | 1800 | rpm |
| | IVC | 570 | degree |
| | EVO | 833 | degree |
| | Sector Decomposition Type | Complete Geometry | |
| | Spray Location, Height | 0.02 | mm |
| | Spray Location, Radius | 0.02 | mm |
| | Sector Angle | 60 | degree |
| | Spray Angle. | 70 | degree |
| | Decomposition Crank Angle | 570 | degree |
| | Compression Ratio | 17.83 | - |

The mesh information in IC Setup course mesh type used with reference size 0.947 and minimum mesh size of 0.19mm and maximum mesh size 0.474mm with inflation layer 3 and other parameters and boundary condition parameters used are summarized in table 2 and table 3 below respectively.

**Table 2. Mesh Information for ICE**

| Domain | Nodes | Elements |
|---|---|---|
| ice fluid chamber bottom | 466165 | 446292 |
| ice fluid chamber top | 14020 | 10143 |
| ice fluid piston | 70305 | 63543 |
| All Domains | 550490 | 550490 |

**Table 3. Boundary Conditions**

| Type | **Zones** | Values |
|---|---|---|
| wall | ice-sector-top-faces | Temperature (k) 602 |
| wall | ice-piston | Temperature (k) 645 |
| wall | ice-cyl-chamber-top | Temperature (k) 567 |
| wall | ice-cyl-chamber-bottom | Temperature (k) 567 |
| wall | ice-cyl-piston | Temperature (k) 567 |

The mass fraction values for species under fuel and oxidizer stream are summarize in table 4 below.

**Table 4. Mass fraction values for species under fuel and oxidizer stream**

| No | Species | Fuel | Oxidizer |
|---|---|---|---|
| 1 | $nc_7h_{16}$ | 1 | 0 |
| 2 | $o_2$ | 0 | 0.2336 |
| 3 | $co_2$ | 0 | 0.00046 |
| 4 | $h_2o$ | 0 | 5e-7 |
| 5 | co | 0 | 0.7658 |

## 3. RESULTS AND DISCUSSION

Distribution of pressure and temperature at various crank angle degrees are shown in figure compression stroke for flat piston. The increment of time step is taken 0.5 crank angle degree. Piston starts from bottom dead center about 541degrees and the maximum pressure reaches at 716 degrees. The pressure increases from 542 crank angle degree to 716 crank angle degree and reaches its maximum value at 716 crank angle degrees. The Maximum pressure at the end of compression is 63.69Kpa and temperature is 1200K.

### 3.1. Velocity Vector Contours

The mesh information shown in figure 1b in IC Setup course mesh type used with reference size 0.947 and minimum mesh size of 0.19mm and maximum mesh size 0.474mm with inflation layer 3. The sector combustion geometry decomposition modelled on Catia is shown in figure 1a below.

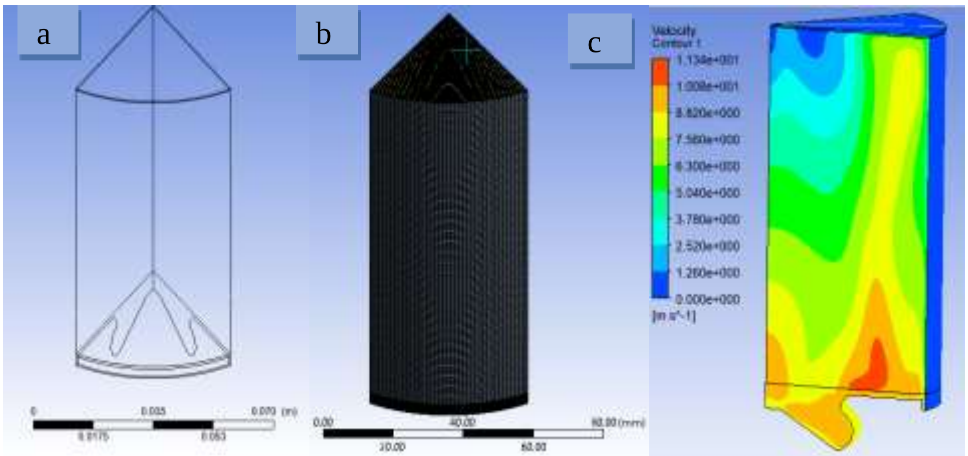


**Figure 1. a) geometry of piston sector, b) mesh of piston sector, c) Velocity Vector Contours**

Results of the analysis were plotted as the contours of velocity magnitude of fluid in the 4-cycles of 4 Stroke IC engine at various time steps and crank angle to check the flow. Figure 1c shows the velocity contour at a crank angle 550 degrees and 850 degrees.

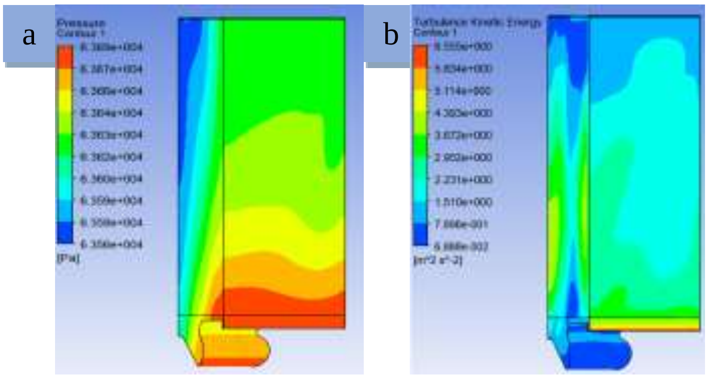


**Figure 2. a) Pressure and b) turbulence kinetic energy Contours**

Results of the analysis were plotted as the contours of pressure magnitude of fluid in the 4-cycles of 4 Stroke IC engine figure 2a shows the maximum pressure of 63690Pa and maximum turbulence kinetic energy of -6.555$m^2/s^2$.

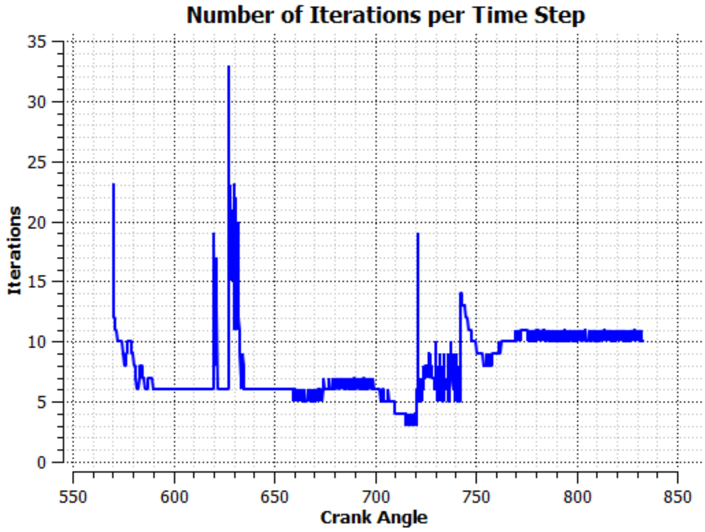


**Figure 3.Number of Iterations per Time Step vs crank angle**

Figure 3 shows number of iteration vs crank angle within each time step. The iteration for intake stroke and compression stroke is done by using time step (0.001) for complete the cycle. More iteration is occurred dung suction stroke is completed- at crank angle 769.8 degrees.

3.2. Swirl ratio Vs Crank Angle

From figure 4 here the swirl ratio is higher at $570^0$ crank angle and $760^0$ crank angles. The initial mixing is having a very high swirl ratio which is considered to be good for engine this may be due to piston shape which is giving better air movement in the cylinder.

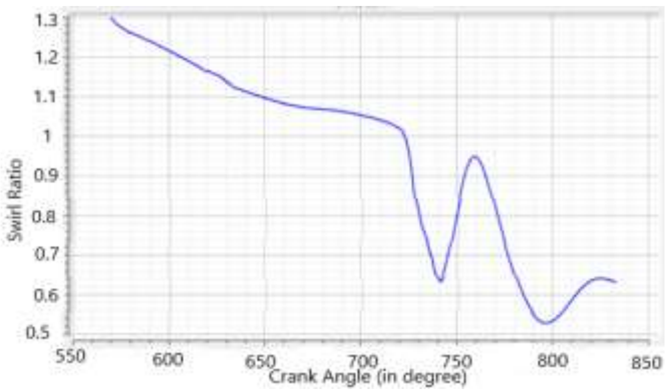


**Figure 4. Swirl ratio Vs Crank angle**.

**3.3. Tumble Ratio Vs Crank Angle**

From Fig. 5 and 6, here the negative or positive magnitude of tumble ratio indicates the direction of the overall in-cylinder tumble flows at a given plane as clockwise or counter clockwise respectively. The reasons for this could be (i) change in the overall tumble flow pattern due to low pressure and bifurcation zones, (ii) change in piston speed with crank angle degree, and (iii) change in the direction of the piston movement during suction and compression strokes. At $630^0$ crank angle we have achieved the maximum tumble ratio.

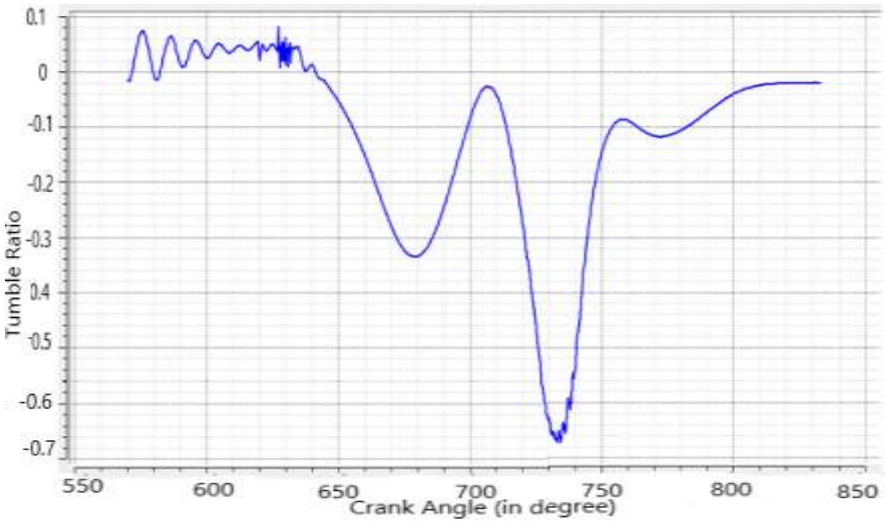


**Figure 5. Tumble Ratio Vs Crank Angle**

### 3.4. Cross Tumble Ratio Vs Crank Angle

At $708^0$ crank angle turbulent kinetic energy is maximum, this may be due to the positioning of piston bowl in centre which gives better guidance for the jet entering the cylinder and after that slowly the turbulent kinetic energy decreases as the piston is moving towards TDC.

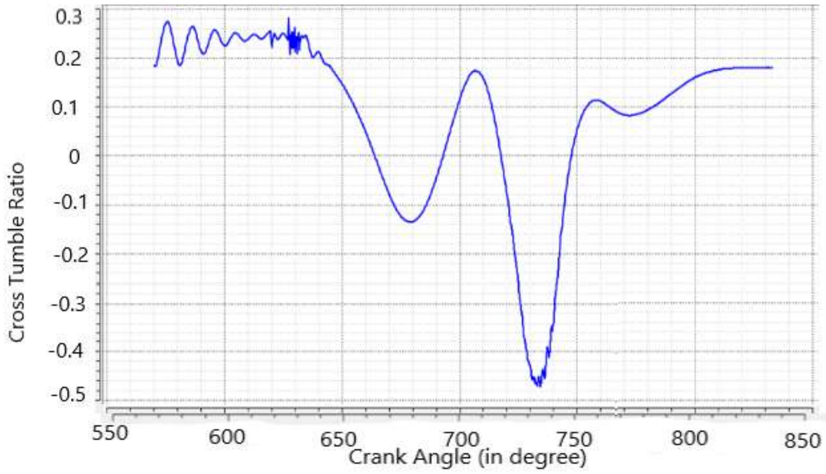


**Figure 6. Cross Tumble Ratio Vs Crank Angle**

### 3.5. Apparent Heat Release Rate

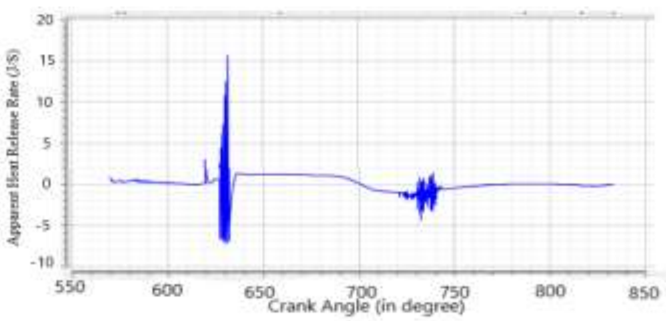


**Figure 7. Apparent Heat Release Rate Vs Crank angle**

Figure 7 shows variation of apparent heat release rate for various crank angle degree during intake and compression strokes an engine speed of 1800 rpm. From the figures, it can be observed that the peak apparent heat release rate reachs at crank angle of 625 degree.

### 3.6. Penetration Length of Injection

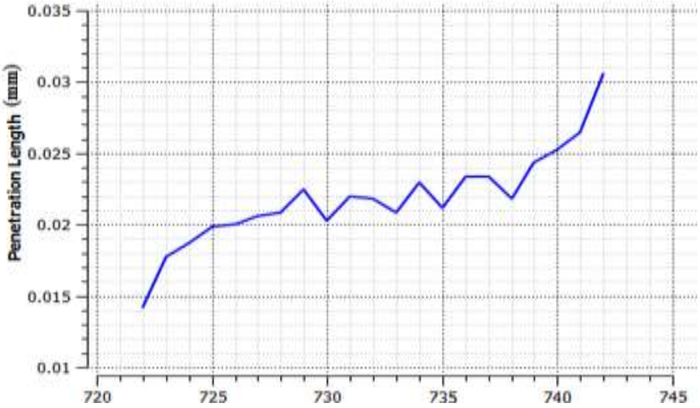


**Figure 8.Penetration length of injection Vs Crank angle.**

Figure 8 shows variation of penetration length of injection for various crank angle degree during intake and compression strokes an engine speed of 1800 rpm. From the figures, it can be observed that overall variation of penetration length of injection with crank angle 722 degree to 742 degree is considered 0.014mm and 0.31mm respectively.

### 4. CONCLUSIONS

From this investigation, it is observed that CFD computational study is important to understand in sector combustion structure during suction and compression stroke on a single cylinder DI Diesel engine with different piston flat geometry. The simulation is carried out using and dynamic motion and velocity magnitude is plotted for crank angle starting from 570 to 833. Among the four strokes, compression and suction (intake) strokes are studied under sector combustion simulation. Turbulent kinetic energy which is the parameter used to quantify the turbulence is analysed and result bring forward the fact that turbulence developed during intake stroke and drop during compression stroke. Turbulence starts building up during the initial stages of the cycle and decays to a minimum at the end. This process keeps getting repeated time and again during the engine performance cycle resulting in useful power. CFD can be used as a useful tool to fix the parameters related to engine performance as well as CFD sector combustion analysis complements testing and experimentation by reducing total effort and cost required for experimentation and data acquisition.

### 5. ACKNOWLEDGEMENT

This research work is sponsored by the Dire Dawa University under Ethiopian ministry higher education institute. The authors would like to thank the Adama science and technology university for their support in a form of facilitating PhD scholar for the student.

### REFERENCES

1. Bérd, P., Béard, P., Mokaddem, K., & Baritaud, T. J. S. t. (1998). Measurement and modeling of the flow-field in a DI diesel engine: effects of piston bowl shape and engine speed. 1583-1595.
2. Dhyani, V., Kumar, D., Ganji, P. R., & Raju, V. J. F. (2014). Numerical Experiment of CI engine combustion using Converge Software. *2014*.
3. El-Adawy, M., Heikal, M., Aziz, A. R. A., Siddiqui, M., & Wahhab, H. A. A. J. A. E. J. (2017). Experimental study on an IC engine in-cylinder flow using different steady-state flow benches. *56*(4), 727-736.
4. Erdil, A., & Kodal, A. J. P. o. t. I. o. M. E., Part C: Journal of Mechanical Engineering Science. (2007). A comparative study of turbulent velocity fields in an internal combustion engine with shrouded valve and flat/bowl piston configurations. *221*(12), 1597-1607.
5. Gugulothu, S., & Reddy, K. J. J. A. F. M. (2016). CFD simulation of in-cylinder flow on different piston bowl geometries in a DI diesel engine. *9*(3), 1147-1155.
6. Karunakar, K., Reddy, C. V. B., Basha, D. J., & Sivaramakrishnaiah, M. J. I. R. J. E. T. (2016). Design & analysis on a diesel engine by implementing tangential grooves on piston for combustion improvement through CFD. *3*(6), 646-652.
7. Krishna, B. M., Bijucherian, A., Mallikarjuna, J. J. I. J. o. E., & Sciences, A. (2010). Effect of intake manifold inclination on intake valve flow characteristics of a single cylinder engine using particle image velocimetry. *6*(2).
8. Kurniawan, W. H., Abdullah, S., & Shamsudeen, A. J. J. o. a. S. (2007). A computational fluid dynamics study of cold-flow analysis for mixture preparation in a motored four-stroke direct injection engine. *7*(19), 2710-2724.
9. Londhekar, A., Shelke, S., Patil, N., Rajput, R., & Phadkale2019, K. (2019). A REVIEW ON: EFFECT OF NANO-FUEL ADDITIVES ON DIESEL

ENGINE. *Journal of Emerging Technologies and Innovative Research (JETIR)*, *6*(3).

10. Millo, F., Piano, A., Peiretti Paradisi, B., Marzano, M. R., Bianco, A., & Pesce, F. C. J. E. (2020). Development and Assessment of an Integrated 1D-3D CFD Codes Coupling Methodology for Diesel Engine Combustion Simulation and Optimization. *13*(7), 1612.
11. Pathak Yogesh, R., Deore Kailas, D., Patil Vijayendra, M. J. I. J. o. R. i. E., & Technology. (2014). In cylinder cold flow CFD simulation of IC engine using hybrid approach. *3*, 16-21.
12. Raju, V., & Rao, S. S. J. P. E. (2015). Effect of fuel injection pressure and spray cone angle in DI diesel engine using CONVERGETM CFD Code. *127*, 295-300.
13. Rao, G. P., Dhyani, V., Kumar, D., Raju, V., & Rao, S. S. J. W. J. o. E. (2016). Investigating optimal operating parameters of DI diesel engine: a CFD approach using CONVERGETM.
14. Shafie, N. M., Said, M. M. J. J. o. E. S., & Technology. (2017). Cold flow analysis on internal combustion engine with different piston bowl configurations. *12*(4), 1048-1066.
15. Stone, C., & Ladommatos, N. J. S. t. (1992). The measurement and analysis of swirl in steady flow. 1674-1690.
16. Wei, S., Wang, F., Leng, X., Liu, X., Ji, K. J. E. C., & Management. (2013). Numerical analysis on the effect of swirl ratios on swirl chamber combustion system of DI diesel engines. *75*, 184-190.
17. Zahid, M., & Syed, K. S. (2021). Investigation of Pollutants Formation in a Diesel Engine Using Numerical Simulation.