

# In Cylinder Combustion Analysis of DI Diesel Using Experimental and CFD Approach

Vijayendra Patil<sup>#1</sup>, Aashish Agrawal<sup>#2</sup>

<sup>#1</sup>PG Student, Technocrats Institute of Technology, RGPV, Bhopal, MP, INDIA

<sup>#2</sup>Asso.Professor, Technocrats Institute of Technology, RGPV, Bhopal, MP, INDIA

**Abstract**— Combustion is one of the most complicated process in IC Engines, in DI diesel engine fuel is injected into combustion chamber with pressure more than 200 bar. Combustion causes rise in temperature in combustion chamber due to high temperature various parts of engine are subjected to thermal stresses which may cause failure of these components. In order to avoid failure it is necessary to make a study of combustion so that preventive measures can be taken. Experimental techniques are limited for visualisation of combustion and hence are costly. CFD offers the opportunity to carry out repetitive parameter studies with clearly defined boundary conditions in order to investigate various configurations.

The objective of this work is to calculate the combustion properties of four stroke (DI) diesel engine by numerical approach using CFD package. Experimental analysis will be carried out on a four stroke variable compression Diesel engine for validation with numerical solutions. The simulation starts at 317 ° CA i.e before fuel injection which is at 337° CA. The combustion process terminates near about at 50° CA after injection hence, the simulation will end at near about 387° CA. The combustion Parameters will be studied at these Crank Angles.

**Keywords**— IC Engine Combustion, CFD Analysis of Engine, Direct injection Diesel Engine Combustion Analysis.

## I. INTRODUCTION

Combustion is one of the complicated process in engines in which certain elements of fuels like carbon and hydrogen combine with oxygen liberating heat energy and causing increase in temperature of gases. The conditions necessary for combustion are the presence of combustible mixture and some means of imitating the process. Basically, the combustion in engines occur towards the end of compression stroke. In Direct Injection (DI) engines fuel at pressure more than 200 bar is directly injected into the combustion chamber. In High Speed Direct Injection Diesel engines the combustion is critical in the Cylinder at the end of the compression stroke when piston is near TDC [1]. Nowadays Manufacturers and Research are looking for cutting edge solutions for low emission, high combustion efficiency and low fuel consumption[2]. IC engines involves complex fluid dynamics interaction between air flow, fuel injection, moving geometries and combustion[3]. Due to high temperature gradients in engines during combustion various engine parts are subjected to thermal stresses which may cause seizure and damage to vital engine components[4].

Quite a few Experimental works in visualizing In cylinder flows have been conducted to measure velocity fields by using hot wire anemometry or Laser Doppler Velocimetry. It is really hard task to perform because the measurements of in cylinder flows in the engine are characterized by highly complex three dimensionality, turbulence and unsteadiness[5]. Due to extreme conditions inside a typical IC engine experimental techniques are limited. From last few decades a computer simulation method has emerged rapidly to solve such extreme conditions known as CFD computational fluid dynamics. CFD offer the opportunity to carry out repetitive parameter studies with clearly defined boundary conditions in order to investigate various configurations[6].

## II. PROBLEM DEFINATION

The objective of this work is to calculate the combustion properties of four stroke (DI) diesel engine by numerical approach using CFD package ANSYS Fluent. The most widely used CFD package and well known simulation software for variety of engineering applications is Fluent. Experimental analysis will be carried out on a four stroke variable compression Diesel engine for validation with numerical solutions. During Combustion Both the Inlet and Exhaust valves are closed hence, a 3D model is designed without valves. A centrally located fuel injector with a 30° sector model is considered which indicates the combustion chamber of the engine. To reduce the computation time only 30° sector is meshed. The combustion parameters such as pressure, volume, temperature and Heat Release Rate affects the combustion process and operation of the engine. The simulation starts at 317 ° CA i.e before fuel injection which is at 337° CA. The combustion process terminates near about at 50° CA after injection hence, the simulation will end at near about 387° CA. The combustion Parameters will be studied at these Crank Angles.

CFD Approach plays a very vital role in Research and analysis from recent years. As this technique has become a part of every research field[7]. In Internal combustion Engines CFD may be useful to study one of the most complicated process of combustion in Engines. By combustion analysis and simulation the plots of crank angle vs pressure, crank angle vs temperature and pressure vs volume can be obtained which can be validate with the experimental results thus conforming the accurate of results obtained by using CFD.

This method will be very advantageous to study process of combustion at various time steps during operation of engine by simulations thus reducing the experimental time and calculations. The high initial cost of experimental setup can be avoided as the results are obtained directly on computer screen within some time. The engine studied in this paper is a single cylinder direct injection (DI) diesel engine. The engine consists of one inlet valve and one exhaust valve with a centrally located fuel injector. CFD analysis of the combustion process in this engine is carried out by numerical method using a CFD package FLUENT. The engine geometry consists of inlet manifold, exhaust manifold a fuel injector and piston bowl fig.1. During combustion in engines both the inlet and exhaust valves are closed and the combustion takes place above the piston usually known as combustion chamber. Hence, in order to reduce the computational time a simplified model having 30° sector is consider for analysis. The model consists of a centrally located fuel injector and piston bowl which forms the combustion chamber fig 1. Various fluid zones and interfaces are provided for different regions of a model to enhance the results and for providing accurate interactions of boundary conditions. A dynamic mesh is carried out to simulate the suction and compression process up to the point of injection. After injection the monitors are applied for getting results of combustion parameters such as pressure, temperature and volume



Fig. 1 A 30° sector model for analysis

### III. EXPERIMENTAL METHODOLOGY

The setup consists of single cylinder, four stroke, VCR (Variable Compression Ratio) Diesel engine connected to eddy current type dynamometer for loading. The compression ratio can be changed without stopping the engine and without altering the combustion chamber geometry by specially designed tilting cylinder block arrangement. Setup is provided with necessary instruments for combustion pressure and crank-angle measurements. These signals are interfaced to computer through engine indicator for Pθ–PV diagrams. Provision is also made for interfacing airflow, fuel flow, temperatures and load measurement. The set up has stand-alone panel box consisting of air box, two fuel tanks for dual fuel test, manometer, fuel measuring unit, transmitters for air and fuel flow measurements, process indicator and engine

indicator. Rotameters are provided for cooling water and calorimeter water flow measurement.



Fig. 2 Experimental Setup of VCR Engine.

The valve timing of the engine is as below:-  
 Inlet valve closes 35.5° after BDC. At 215.5° CA.  
 Fuel injection starts 23° before TDC. At 337° CA.  
 Exhaust valve opens 35.5° before BDC.  
 From the valve timing diagram  
 Table 1 shows the engine specifications.

TABLE I  
 TECHNICAL SPECIFICATIONS OF THE ENGINE.

PARAMETER	VALUES
Make	Kirloskar Oil Engines
Type	Four stroke, water cooled
No. of Cylinders	1
Bore	87.5
Stroke	110
Connecting Rod Length	234
Cubic Capacity	661
Compression ratio	17.5 : 1
Compression Ratio Range	12 to 18
Maximum Speed	2000
Power at 1500 RPM	3.5 KW

A Computer interface is used for displaying results of Pressure vs Crank angle, Volume vs Crank angle, Pressure vs Volume, Temperature vs Crank angle, Net heat release rate vs crank angle etc. Engine Soft is a software which is provided by the manufacturer of Experimental setup o visualize the various parameters at every Crank angle. Hence, finally this results will be compared with numerical Results for the accuracy and stability of numerical simulations.

### IV. NUMERICAL METHODOLOGY

In order to study the process of combustion a finite volume commercial flow program has been used to solve the

discretized Navier Stokes equation. The standard  $k-\epsilon$  turbulence model for high Reynolds's numbers with wall function is used[8]. The  $k-\epsilon$  model for solving Navier stokes equation employees the eddy viscosity concept. Since  $k$  and  $\epsilon$  both are produced destroyed and transported by flow, one obtains partial differential equations governing their values in the floe domain[9]. The code is based on pressure-correction method using PISO algorithm. The first order upwind differencing scheme is used in the momentum and turbulence equations[10]. The governing equations for the gas flow and the combustion process are Mass equations, Momentum equations and Energy equations. Finite volume method has been used to solve the discretized partial differential equations governing the mean fluid motion. The discretization of space in the appearance of cell numbers and time in the form of time stepping are set which provide a reliable indication of the suitability of the discretization settings. The time discretization is necessary to obtain the best compromise between the convergence requirements and the computing time i.e. the smaller time steps were set for initiating simulations when the crank angle reached the fuel injection timing and the TDC position[11]. A species transport model with discrete phase has been used for non premixed combustion with non adiabatic equilibrium phase model. Injection is define at specific crank angle at which the fuel injection starts and stops. The fuel temperature, pressure and mass flow rate is defined during injection. The fuel injection parameters like location of injector, size of injector, injection temperature and pressure, mass flow rate etc. are having significant effect in diesel combustion modeling The droplet model of injection with injection location is specified. The surface monitor is defined to find Pressure, Temperature and volume from  $20^\circ$  before injection to end of combustion. As discuss earlier in Table 1 the fuel injection starts at  $23^\circ$  before TDC means at  $337^\circ$  CA and the combustion ends at  $397^\circ$  CA hence simulation starts at  $317^\circ$  CA and ends at  $397^\circ$  CA. Iso surface is created to plot contours of temperature and pressure for the combustion simulation.

*A. Grid Independence Study*

A grid generation program has been used to create hexahederal grid. The Hexahederal cells have been adopted for mesh generation because they provide better accuracy and stability as compared to Tetrahedral cells[6]. The main purpose of using hexahederal element is its requirements of moving mesh. Dynamic mesh is carried out for this problem in order to study the In-cylinder process going on inside the engine cylinder. Hence, dynamic mesh starts at  $0^\circ$  CA up to  $317^\circ$  CA which is the starting point of our simulation. Four mesh cases are considered for this problem Case I, Case II, Case III and Case IV. Table 2 shows the description for all the above Cases considered for Grid independence study for CFD Simulations.

TABLE 2  
DESCRIPTION OF GRID CASES.

CASE	FACE SIZING	NO. OF NODES	NO. OF ELEMENTS
Case I	10 mm	905	327
Case II	5 mm	2830	1391
Case III	2 mm	15130	8413
Case IV	1 mm	60797	35154

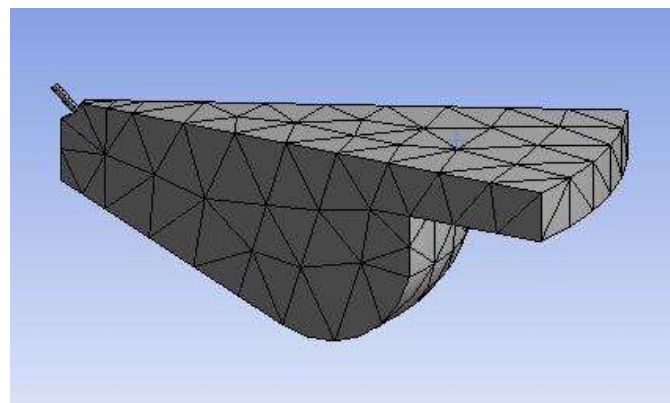


Fig. 3 Mesh Case I

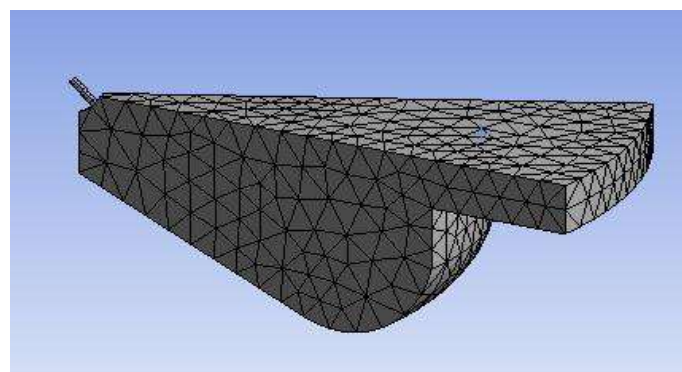


Fig. 4 Mesh Case II

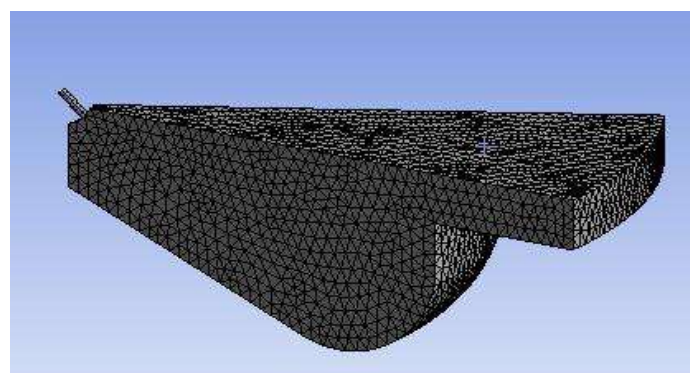


Fig. 5 Mesh Case III



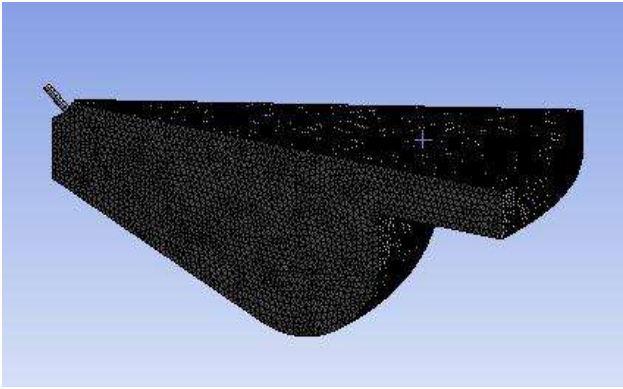


Fig. 6 Mesh Case IV

Calculations were performed using Four Cases from which Case III and Case IV found results which were in good agreement with the experimental results. The experimental data is available for the same case hence, first the Pressure and Temperature is compared with all the four Cases shown in Fig. 7 and Fig. 8. The Case III is taken as the reference grid and the further calculations are performed using Case III. As Case III and Case IV provides same results but number of Nodes and Elements for Case IV are greater than Case III. so using Case III as reference grid will reduce the computational time and the probability of accuracy will increase.

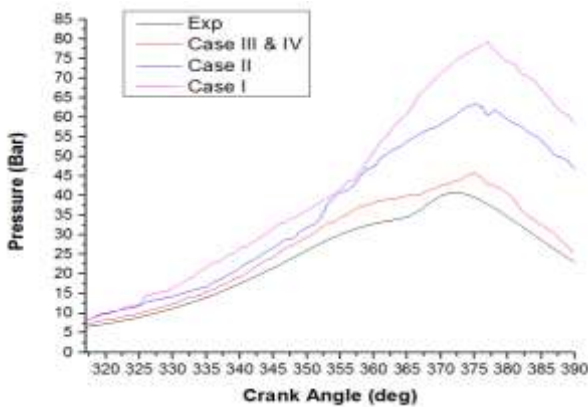


Fig. 7 Pressure comparison for grid independence study.

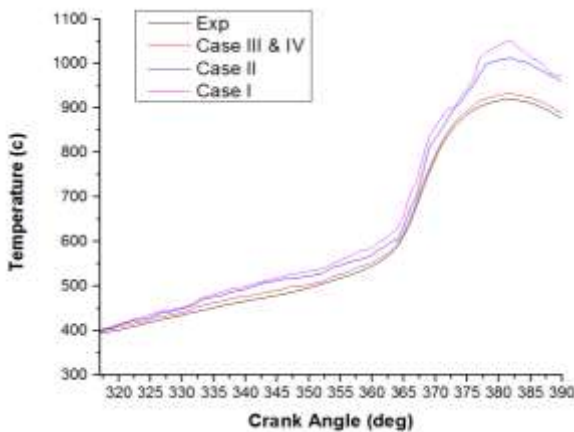


Fig. 8 Pressure comparison for grid independence study.

From the above plots of Pressure (Bar) and Temperature (c) VS Crank Angle it is found that Case III & Case IV results are optimum. So finally, Case III is considered for further analysis.

### B. Time Step Study

As explained earlier that the time discretization is necessary to obtain the best compromise between the convergence requirements so time step plays a very important role in solving combustion problems. Hence, time step is varied from 1, 0.5 and 0.25 of which time step of 0.25 is found optimum for combustion analysis[12]. The volume monitors are obtained at various zones in the form of output data with respect to flow time and time step. The number of iterations required for solving the combustion problems depends upon the time step and the CA to run. The computational time required for such simulations is more as they involved Dynamic mesh and Transient analysis. A CFD combustion simulation has been carried out for different time steps and compared with the experimental results of engine such as Pressure and Temperature from which the optimum time step is find out which gives the results which are in good agreement with the experimental results. Fig 9 and Fig 10 shows the graphical comparison of different time steps for Pressure and Temperature at different Crank Angles.

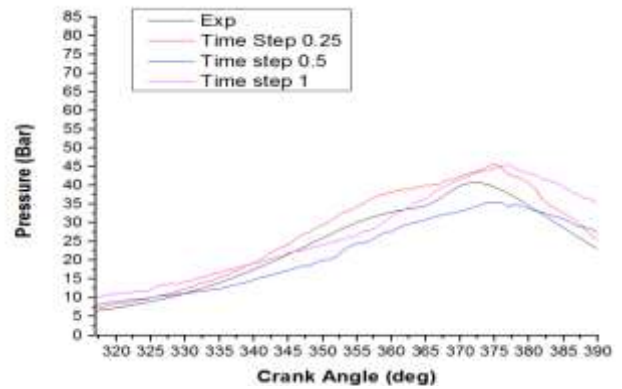


Fig. 9 Pressure for different Time steps.

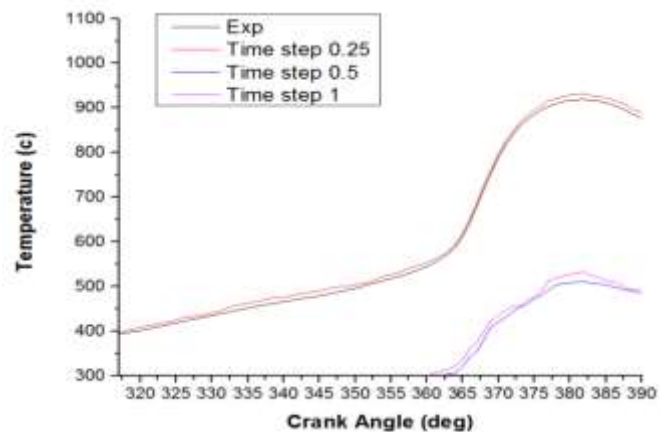


Fig. 10 Temperature for different Time steps.

V. RESULTS AND DISCUSSIONS

The Combustion analysis is carried out on a engine by Experimental and CFD methods. First Experimental Study is carried out which gives various charts for Pressure, Temperature, heat release rate and volume at different crank angles. CFD analysis is carried out by selecting proper grid and time step by grid independence study and time step study. Proper grid and time step is selected according to the optimum methods and finally using numerical modelling the simulation of engine is carried out to get insight of various parameters. Fig 11 to fig 13 shows the comparisons of experimental and CFD results.

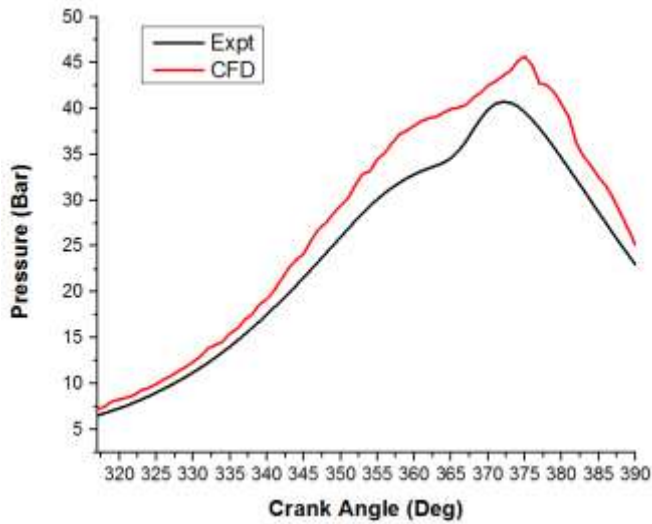


Fig. 11 Comparison of Experimental and CFD Pressure.

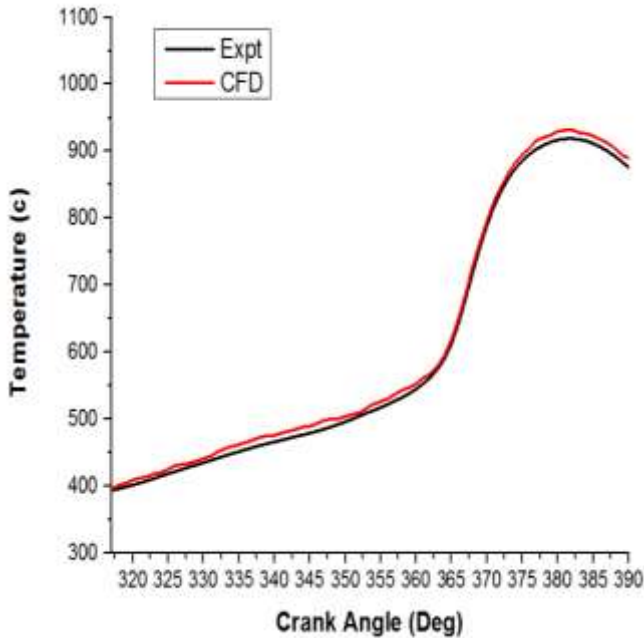


Fig. 12 Comparison of Experimental and CFD Temperature

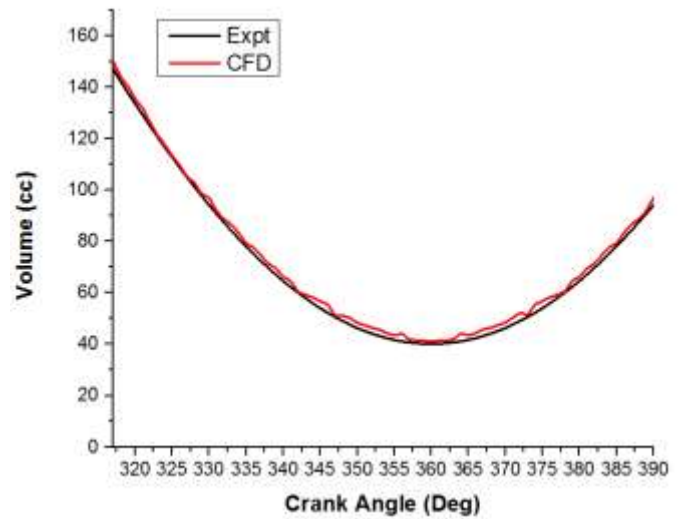


Fig. 13 Comparison of Experimental and CFD Volume.

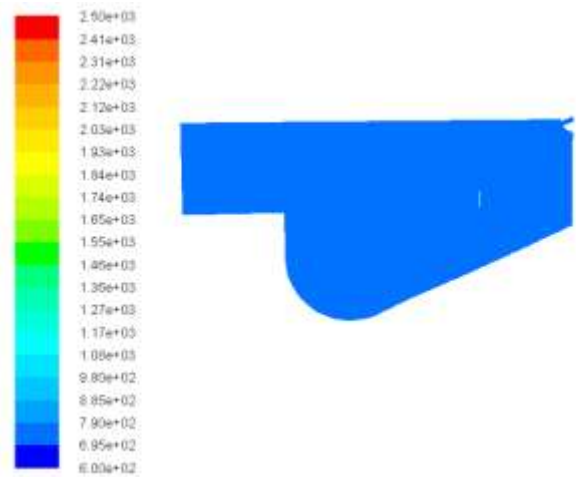


Fig. 14 Contours of Temperature at 327° CA

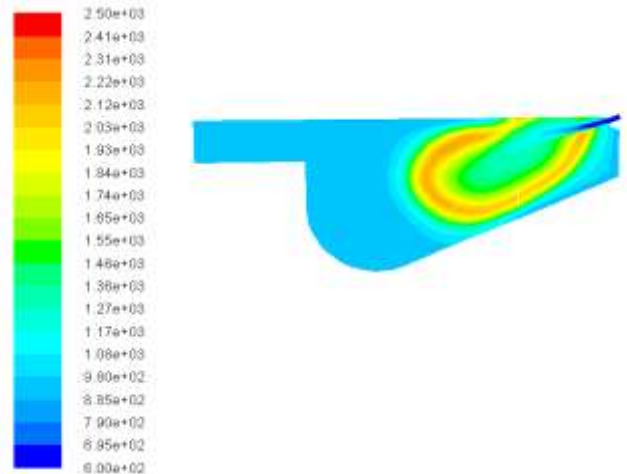


Fig. 15 Contours of Temperature at 342° CA

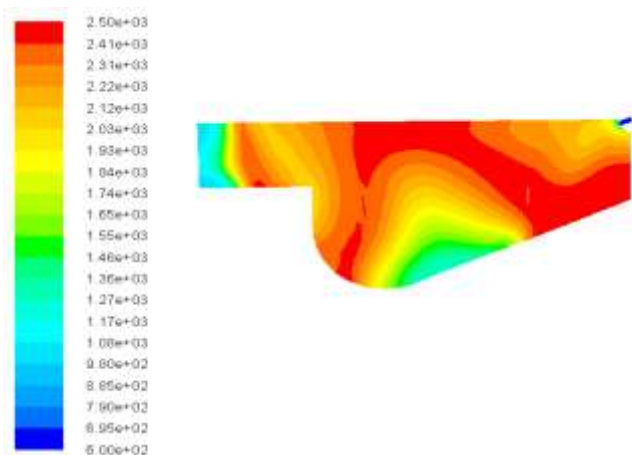


Fig. 16 Contours of Temperature at 387° CA

Fig 14 to Fig 15 shows temperature distribution in combustion chamber at various crank angles, which is not possible in experimental setup.

## VI. CONCLUSIONS

A CFD combustion Simulation has been carried out for a four stroke single cylinder DI engine. Due to complexity of problem only a 30° sector of a model is modelled in a 3D software. Model is defined with various Zones to define the boundary conditions and interfaces, Dynamic mesh is carried out In-cylinder simulation. Grid Independence Study is carried out to find optimum grid to reduce computational time and improve accuracy of results for these four cases are considered from which case III is found to be feasible. Three cases of time step are considered for Time step study, from which optimum time step of 0.25 provides good results is selected. The various solver methods of a commercial CFD package is used for solving the problem. Validation of Results is done with the good quality mesh results by comparing it with Experimental data obtained. The plots of Pressure, Temperature and Volume are plotted against Crank angle. The contours of Pressure at various Crank angles are visualized for simulation and insight to In-cylinder combustion processes which is not possible practically. The plots shows that CFD results are in good agreement with Experimental results with a maximum difference of 15%. In general, this study shows that In-cylinder CFD simulation and study yield accurate results that allow improving the knowledge of combustion in DI diesel engines. The work presented in this paper insights the simulation process inside the engine cylinder, CFD has become modern tool to study the complicated process of combustion in engines, Instead of using highly cost experimental setup this technique can be adopted for combustion analysis of DI Diesel engines, which could play a very important role in engine design, research and development.

## REFERENCES

- [1] F. Payri, J. Benajes, X. Margot and A. Gil, "CFD modeling of the in-cylinder flow in direct-injection Diesel engines", ELSEVIER, Science Direct, Computers & Fluids 33 (2004) 995–1021.
- [2] G. Sucharitha and A. Kumaraswamy, "Analysis on Three Dimensional Flow of Direct-injection Diesel Engine for Different Piston Configuration using CFD", Indian Journal of Science and Technology, Vol 6 (5S), Print ISSN: 0974-6846, Online ISSN: 0974-5645, May 2013.
- [3] Priscilla and Meena. P, 'A Comprehensive Study On In-Cylinder Ic Engine Due To Swirl Flow', International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 2 Issue 7, July – 2013.
- [4] Fritz Bedford, Xiao Hu, and Ulrich Schmidt, "In Cylinder Combustion Modeling and Validation using Fluent".
- [5] Wendy Hardyono Kurniawan, Shahrir Abdullah and Azhari Shamsuden, "A Computational Fluid Dynamics Study of Cold – Flow Analysis For mixture Preparation In a Motored Four Stroke Direct injection Engine", Journal of Applied Sciences 7 (19); 2710-2724. ISSN 1812-5654, 2007.
- [6] Abhilash M Bharadwaj, K Madhu, Seemanthini J, Vismay K G, Aravind T and Anand M Shivapuji, "Study of Swirl and Tumble Motion using CFD", International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME), ISSN : 2319 – 3182, Volume-1, Issue-2, 2012.
- [7] Maher M Abou Al-Sood Dr, Yousef M Abdel-Rahim Prof, Mahmoud A Ahmed, "Rapid thermodynamic simulation model for optimum performance of a 4-stroke, direct injection, variable compression ratio diesel engine", International Journal of Energy and Environmental Engineering, Springer Open, ISSN 2251-6832, 2012.
- [8] S Gavudhama Karunanidhi, Melvinraj C R, Sarath Das K P and G Subba Rao, "CFD Studies of Combustion in Diesel Engine", International Journal of Engineering Research and Applications (IJERA), ISSN: 2248-9622, Vol. 3, Issue 4, Jul-Aug 2013, pp.827-830.
- [9] S. M. Jameel Basha, P. Issac Prasad and K. Rajagopal, "SIMULATION OF IN-CYLINDER PROCESSES IN A DI DIESEL ENGINE WITH VARIOUS INJECTION TIMINGS", ARPN Journal of Engineering and Applied Sciences, ISSN 1819-6608, VOL. 4, NO. 1, FEBRUARY 2009.
- [10] A. HATZIAPOSTOLOU and G. RAPTIS, "CFD modeling of the in-cylinder flow in a variable compression", Proc. of the 9th WSEAS Int. Conf. on Mathematical and Computational Methods in Science and Engineering, Trinidad and Tobago, November 5-7, 2007.
- [11] Shahrir Abdullah, Wendy Hardyono Kurniawan and Azhari Shamsudeen, "Numerical Analysis of the Combustion Process in a Compressed Natural Gas Direct Injection Engine", Journal of Applied Fluid Mechanics, ISSN 1735-3645, Vol. 1, No. 2, pp. 65-86, 2008.
- [12] Patil Vijayendra Maharu and Aashish Agrawal, "OPTIMIZATION OF TIME STEP AND CFD STUDY OF COMBUSTION IN DI DIESEL ENGINE", IJRET: International Journal of Research in Engineering and Technology, eISSN: 2319-1163 | pISSN: 2321-7308, Vol- 03.
- [13] V.Ganesan, "Internal Combustion Engines." Second edition, Tata Mcgraw Hill ISBN 0-07-049457-6.
- [14] Apex Innovations, VCR Engine User Manual.
- [15] ANSYS Fluent User Manual.