ISSN: 2366-1313

Computational Fluid Dynamics Analysis and Optimization of Engine Design Fuelling

N.SWARNA RAJ[1],D. RAMESH BABU[2], Dr. M. MASTANAIAH[3]

[1]M. Tech student, Thermal Engineering, Dept.of Mechanical Engineering, Dr.Samuel George Institute of Engineering and Technology, Markapur, Prakasam D.T, AP

[2]Associate Professor, Dept.of Mechanical Engineering, Dr.Samuel George Institute of Engineering and Technology, Markapur, Prakasam D.T, AP

[3]Professor,Dept.of Mechanical Engineering, Dr.Samuel George Institute of Engineering and Technology, Markapur, Prakasam D.T, AP

Abstract: *Computational Fluid Dynamics (CFD) is a device that can provide designers with accurate and valuable data, which can feed back into the design to extend a properly designed product. It is most notably used in the automotive industry for engine optimization and design, particularly for engine design. In CFD analysis, the basic equations describing the flowing fluid are solved using a grid representing an engine as 3D geometry and including all sub-models for fuel injection, turbulence, chemistry, and combustion. Engineers can visualize and examine the geometry in motion and its impact on fluid flows and the complex interactions of an engine with the help of CFD capabilities. Port slip analysis, cold flow evaluation, IC combustion simulation and full cycle simulation are the four scopes of CFD analysis of motor design, and each degree is more complex than the previous one. Diesel is used for combustion analysis in this task. However, turmeric gasoline is also used close to diesel for combustion assessment. Since 30% of conventional gasoline is better suited for our planet, the fuel of opportunity is used. The temperature and pressure limits were obtained using CFD equipment in this study.*

Keywords: *Computational Fluid Dynamics, Fuel Injector, Emission control, Fuel Flow Penetration.*

I. INTRODUCTION

The effect of injector nozzle geometry and operating pressure conditions, including initial pressure, ambient pressure and injection pressure, on the transient behaviour of fuel mist has been demonstrated through experiments. The flow information of the internal release nozzle and the actual nozzle were also investigated experimentally and numerically to clarify the effect of the internal release float nozzle on the external spray. For the development of injection

ISSN: 2366-1313

pressure, droplet sizes and velocities were obtained at maximum line pressures of 21 MPa and 105 MPa. The current pressure ignition engine must meet environmental and economic requirements. It must have high-performance fuel economy and low protection ratings and must allow operation in compliance with specified emissions standards. Since the combustion technology, pollution and noise emissions are mainly controlled by the fuel injection method; many attempts have been made to develop new and existing diesel gas injection structures. Some researchers have extensively investigated bore slip in diesel injector nozzles due to its potent effects on fuel mist and consequent spray combustion in diesel engines. From the first look at the cavity that goes along with the flow in small nozzles, he said that the holes create a large turbulence amplitude that leads to improved atomization jetting and the cavities tested in different nozzles with unique geometries.

The internal combustion engine (ICE) is another essential device used to generate mechanical power with the help of energy conversion from fuel. This takes the area with many complex tactics. Internal combustion engine (ICE) modelling is one of the highest levels of complexity and a challenging task. This is due to the additional parameters that play a significant role in fuel injection, flame spread, ignition procedure and the type of fluid produced by the instability. Therefore, the appearance and improvement of internal combustion engines need a profound description of what complex physical phenomena occur within the engine, especially with the development of generation. The flow function plays one of the essential functions within the machine's overall performance. Therefore, many researchers involved in the study of ICE, Heywood, said that air turbulence is better for combining air and fuel. Kuo predicted changes in fuel stress in the cylinder through a simple thermodynamic principle model. The main weakness of this model is that it no longer considers changes in the reaction of gasoline during combustion. Movva simulated fluid flow in an ICE combustion chamber using wave motion simulation, FORTRAN encoded, receiving pressure distribution and cadence in the extreme case of release. Princes studied bloodless drift in a twin-engine and used two-dimensional fluid time simulations. They obtained stress, timing, and temperature with crankshaft position with the help of evaluating sections at specific angles. Murasaki et al. investigated fluid properties in a GDI engine with numerical calculation using ANSYS and displaying sample flow and pressure during compression and force

ISSN: 2366-1313

stroke duration. Ravichandran et al. converted CI to SI using a dynamic grating approach with sixteen engine zeros numerically and experimentally, recording the temperature and pressure inside the combustion chamber. Abderrahimant et al. studied the intake manifold using CFD and confirmed the rotation ratio (SR) variable in the cylinder with an estimation of the crank angle of the single manifolds. This article represents a modern approach to 3D simulation and geometry modelling using the finite quantity technique. This paper aims to extend the simulation range to calculate the flow characteristic in the combustion chamber of a spark ignition engine with the help of a dynamic network approach to visualize the drift inside the combustion chamber with the help of using an ICE code with an engine speed of 2500 rpm. Velocity, stress and temperature profiles can be studied with respect to exclusive crank angles.

Today, analysis is becoming more important due to new designs and additional failures in the real world. Mathematical evaluation is appropriate in international real-time analysis to overcome these problems, and this method can complement real-time analysis. This approach is called CFD (Computational Fluid Dynamics). In this technique, we can observe the fluid flow, including (air, water, chemical compounds, etc.) The fluid flow in the frame can analyze the design, thermal resistance, conductivity, convection, combustion analysis, etc. In this fluid (fuel and liquid), flows are controlled by partial differential equations representing the conservation laws of mass, momentum and force. Computational Fluid Dynamics (CFD) is the art of replacing PDE systems with a set of algebraic equations that can be solved using virtual computing systems. In this document, the analysis will end on both turmeric and diesel by having several outlines and diagrams and points out the distinction between diesel and turmeric with the help of a comparison between them.

A CFD simulation can be a powerful engineering device that allows you to understand the thermochemical methods involved in an internal combustion engine without using complex and expensive dimensional methods, if any. Measuring some objects (including the slip field or the temperature inside the cylinder) in the 3D region can be difficult. One-dimensional models no longer represent all geometric structures, making CFD simulation a vital addition. It can also be a valuable tool for doing a great deal of testing and research on an engine without

ISSN: 2366-1313

having to do verification which can reduce development costs. CFD simulations can also be a short, cost-effective way to research future engine designs and concepts before producing a prototype, further reducing development costs. Unfortunately, CFD research has not yet reached a country that thoroughly describes all the processes inside the internal combustion engine. This is mainly because it is a complex mechanical device involving many simultaneous interactive, thermal and chemical fluid processes. Due to the reciprocating nature of internal combustion engines, there can be severe distortion in dominance response due to valve-piston movement and the shape of the piston head and cylinder, which can be vital to design capabilities. They are often very complex.

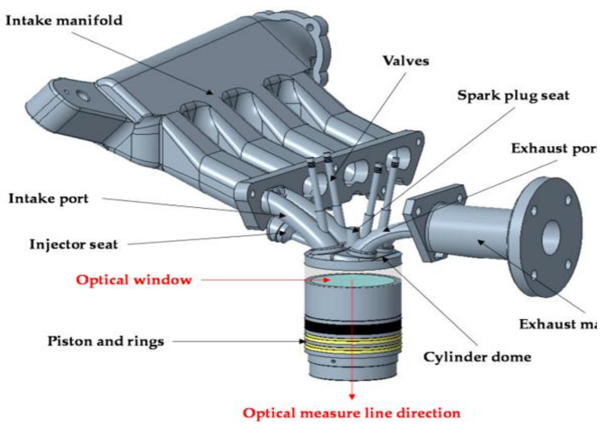


Fig.1 Optimal measures line directions

II. LITERATURE SURVEY

The analysis of internal combustion engine strategies and their use in engine research and improvement has an extensive and systematic record. Although new dimensions were added to this system in terms of emissions modelling, engine simulations for overall performance, and performance modelling, these activities were aimed at developing realistic approximations of actual engine actions along with fuel consumption. Compression, expansion, exhaust, and more correct techniques for calculating thermodynamic homes for operating fluids used within engines.

U Kongre and V. Sunnapwar [3] has portable models of fluid dynamics and experimental validation of a direct ignition diesel combustion engine. In his opinion, consideration has been given to developing and using sub-models for combustion analysis by indirect injection. Experiments were carried out on a single-cylinder and DI engine at full load conditions at a constant speed of 1500 rpm. Combustion parameters, including the rate of pressure rise, heat release charge, and cylinder pressure obtained from the test. Numerical modelling was solved, taking into account the effect of perturbation. The l perturbation model is used for the k-version of the Model Theorem Group (RNG). A comparison was made between

ISSN: 2366-1313

the simulation results and the experiment in terms of the tension armload, cylinder throwing and thermal extrusion load. It is concluded that computational fluid dynamics is a reliable tool for studying the combustion process of an IC engine.

K Pandey and B Roy [4] made the intake valve CFD of the gasoline injection port of the SI engine. The air-to-air spark plug at full charge is about 60%, but the actual thermal brake efficiency at full load is set to be 32.60% due to various losses. The primary failure is the lack of combustion time which is about 4.0% and is caused by the final combustion charge. It can be reduced by increasing the importance of turbulence, the degree of turbulence, and the depth of turbulence in turns. In non-smooth combustion, the depth of high turbulence is a critical parameter in flame spread. Droplet flows and flow turbulence are generally formed within the vertical and horizontal planes of the cylinder. Their study concluded that surfaces near the suction valve at high speed, like covers far from the suction valves.

A Patil1 and L Navale [5] performed an experimental validation with computational fluid dynamics evaluation of an exhaust system single-cylinder four-stroke diesel engine. Research advances with computational fluid dynamics and a tailored exhaust system, a trade-off between thermal brake performance and lower back fatigue. For computational fluid dynamics evaluation, three exhaust distributor systems with unusual angles were simulated using appropriate fluid housings and boundary conditions with proper assumptions. The model was built with restricted back pressure, and trials were carried out on a four-stroke, single-cylinder diesel engine with a rope brake dynamometer. The results show that the increased perspective of the inlet cone results in upward drift pressure, which reduces the recirculation areas. The CFD evaluation of the back pressure in the engine indicates a desirable agreement with the experimental paints.

S Zanforlina and A Boretti [6] performed a numerical evaluation of direct methane injection in a 250-cc single-cylinder petrol engine. A study was conducted to determine the capacity of low-pressure fuel injection systems below 20 bar for gas combustion engines. A CFD evaluation was completed to evaluate the effect of valves and fuses on jetting properties, methane-air mixture, load distribution at ignition, injection voltage, and fuse and fuse profiles on jetting properties. The simulation was performed so that the preferred engine combines the most

ISSN: 2366-1313

sensitive surround mode possible. You can also learn the working device and engineering information to handle the complete injection homogenization mixture. Testing indicates gaseous fuel problems to obtain the outstanding rotor hundreds required to demonstrate the best overall performance in partial hundreds.

B Biradar1 and S Kumarappa [7] investigated air volume fraction evaluation using mass explosion simulation of a four-stroke single-cylinder diesel engine. The performance of a diesel engine can be increased by optimizing the design of the combustion chamber, intake manifold, exhaust manifold and piston. The effect of the piston configuration with flat, bowl, and ring-shaped pistons on cylinder float was studied. A mass mixture of air and gasoline is received by increasing vortex density during compression with a suitable piston design, and the vortex speed doubles. The results show that the generation of temperature and pressure is much less in the annular and annular piston than in the flat piston. However, the vortex squeezing is more robust in the annular and bowl-shaped piston than in the flat piston.

A Kolhe and R Shelke [8] investigated the combustion of a CFD modelling engine powered by direct injection of CI biodiesel. The paper shows sub-model optimization of a direct injection combustion engine analysis of Pongamia compression ignition powered by a mixture of biodiesel and diesel pinnata gas. The computational fluid dynamics modelling study used a complex combustion phenomenon inside a diesel engine. Under full load conditions at an average speed of 1,500 rpm, the test was carried out on a single-cylinder diesel engine. The experimental combustion coefficients, including heat release and heat release rate, were obtained. It considers the effect of the perturbation on the numerical model. It was solved by the CFD approach and used the rematch perturbation (RNG) model to model the ok-theory. Simulation effects were obtained, including stress growth rate, heat release rate and cylinder pressure. A good match between experimental and modelling information ensures the accuracy of the numerical predictions. The stress growth rate, heat release rate and maximum cylinder pressure values show striking agreement between the measured data and the experimental modelling.

III. METHODOLOGY

CFD Tool

Internal combustion engine design and manufacturing are under tremendous pressure to improve. The age of engines

ISSN: 2366-1313

demands that they be smooth, reliable, robust, flexible and efficient. Innovative engine designs may be required to meet these needs. The likelihood that different engine designs will perform as they should is critically essential and critical, as internal combustion engines involve complex fluid dynamic interactions between air slip, gasoline injection, moving parts, and combustion. Using the CFD results, the slip phenomenon can be visualized in 3D geometry and analyzed numerically, providing fantastic insight into the complex interactions that occur in the actuator. CFD simulations are part of the design system in automotive engineering, primarily with current-generation drive-ups.

Modelling Geometry

The engineering modelling method was achieved by selecting the actual model of the Hyundai four-stroke four-stroke spark-ignition engine type, as shown in the parent. Then the engineering version of the engine was generated through SolidWorks. As shown in the original file, a single-cylinder engine was implemented to reduce simulation. More data on engine specifications are indexed in the table.

Table.1simulation parameters

Bore	71 mm
Stroke	60mm
Connecting rod length	126 mm
Compression ratio	8.2
Intake valve diameter	32 mm
Maximum intake valve lift	6.9 mm at 104 deg. ATDC
Intake valve opening	41 deg. BTDC
Intake valve closing	84 deg. ABDC
Exhaust valve diameter	26mm
Maximum valve lift	9.6 mm at 64 deg. ABDC
Exhaust valve opening	66 deg. BBDC
Exhaust valve closing	16 deg. ATDC

Fig.2 one cylinder of engine

This provides 3D modelling of the engine from the measured dimensions of the camel bike engine. In the present work, a single-cylinder, four-stroke, spark-ignition Splendor motorcycle engine was considered, with a compression ratio of 9 and a displacement of 97.20cc. The first step is to model the engine using CAD

ISSN: 2366-1313

software from the measured data of the single-cylinder four-stroke gasoline engine. 3D modelling of the machine was completed with the help of Solidworks software. Next, the generated model is interlaced in the ANSYS grid module. Finally, the version developed by the network is imported into the FLUENT simulation software.

Computational modelling:

The 3D version was completed in Solidworks using the unique geometry of the single-cylinder four-stroke petrol engine. The formed geometry consists of the basic information of the actual engine. Engineering cleaning can be performed in the ANSYS network module software to simplify networking. Simple geometry is nested, and unique tag and segment names are assigned.

Mesh Generation:

Because of the rather complex geometry of the top-and-piston sky engine, especially considering the standard valve and valve interaction, networking software technology had to take care of interconnecting all the paperwork patterns of the cell. It is also important that the software can handle unique cell phone segments, each containing specific cell types, within the same population. Since the Mummi approach was used in these boards, there has also been a demand for the easy and fast creation of new grids based on an initial grid. ANSYS software can import the most popular CAD geometry recording formats and convert them to vertices, edges, faces, and sizes. Gambito can routinely or manually create triangles/rectangles (2D) or hexagons, tetrahedrons, and wedge prisms (3-D). It will be noted that each volume can only include one cell shape. Therefore it is necessary to divide the geometry into unique sub-volumes if the entire geometry engine will consist of multiple cell types. This allows for a fast, completely automatic generation of the modern network, which means that the initial network is done manually.

Port Flow Analysis To assess port flow, the geometries of the ports, valves and cylinders are "frozen" at key points throughout the engine operating cycle and using computational fluid slip through the nozzles can be analyzed. You can limit the amount of airflow into the engine, and turbulence, reduce cylinder phases and turbulence. Phenomena, including separation, jetting, throttling, wall effect, intersection, and secondary actions, can be visualized and analyzed through computational fluid dynamics. Products can be received as snapshots of fluid dynamics throughout the engine which can

ISSN: 2366-1313

be used to adjust the orifice geometry to provide the preferred airflow cycle path. Simulation validation using the actual geometry configuration, vortex slip dimension, cadence, and turbulence phases can be performed using techniques including laser Doppler. The results cannot benefit from dynamic events consisting of air pressure, inflation due to piston movement, turbulence production, turbulence, and plunges in the simulated flow range. It is well known that performing buoyancy evaluation on a single factor is a straightforward form of static geometry well-suited to the program's workflow and computational fluid dynamics. The configuration can be performed using the gate, valve, and cylinder geometry in a given area, constructing a network, determining the mass float or voltage drop for compressive slip and disturbance pattern, and calculating the results. The perturbation models used are entirely based on calculating the perturbation effect. Disturbance with flow interactions is a necessary partition and network optimization in the vicinity of vital parietal layers using hypertrophy or boundary. Experimental data provide validation criteria to extend good practice for model preparation and accuracy. However, the variety of basic positions will increase, and thus the type of issues will increase, and the complexity of the problem will increase. Setting a large variety of instances with the same static and wave bridge settings is time-consuming, with the potential for error.

Numerical Test Plan

Three unique engine operating conditions were considered in this study:

• Engine speed at full power, 7,500 rpm, 100% throttle start (full load). It is helpful to evaluate this condition because it is short for the body to inject and then homogenize the gas-air aggregates.

• Engine speed at full power is 7,500 rpm at 30% throttle open (half load). In this case, the flow discipline in the cylinder is much weaker than during a full charge, and the aggregate formation in the combustion chamber and primary chamber can worsen.

• Low revs, 2,500 rpm, at 10% throttle open (low load). This situation is essential for two-stroke engines because the incoming air stream is insufficient to remove the exhaust gases inside the cylinder.

As previously described, two exceptional injection techniques are simulated: cylinder head positioning (CH) and switch port positioning (TP). The locations of both injectors were tested for the three

ISSN: 2366-1313

engine drivers. Each simulation run took 72 hours on a 24 HPC compute node CPU.

Since the HPDI device applied in a 300 cm3 petamotor has been previously evaluated by some authors, the injector used in this work is the same as that used in [7,8]. The mass of gasoline injected into the cycle step is now forced to acquire common equivalent positions both in the cylinder and in the preparation chamber. The choice of onset of injection (SOI) was mainly based on previous simulations of LPDI and HPDI. At 7500 rpm and full load, the amount of fuel to be injected as a function of the cycle is close to the maximum rate and the actual time available for vaporization and homogenization of the fuel droplets is small; For this reason, SOI must be pre-configured with Top Dead Center (TDC) recognition. On the other hand, this improvement should be limited if you think fast circuit problems are to be avoided. At medium and occasional hundreds, the risk of a rapid gas circuit is less applicable because a small amount of gasoline is injected; However, advanced injection is still essential because the progressive slip discipline is unable to enhance gas homogenization. In all cases, a quality compromise was detected by adopting SOI close to BDC. Summarizes the test plan for simulating two waiting rooms.

IV. RESULTS AND DISCUSSIONS

The numerical analysis has been performed to determine the flow within the engine, so the simulation provided stress statistics for flow without blood 4 times, as shown in figure.3. The result indicates that the pressure is a vacuum on the intake stroke because the intake valve begins to open 360 degrees, the piston acts from TDC downward, increasing the combustion chamber volume, and then the tension is reduced. This seems evident at 390 AC. Then as the piston rises from BDC to TDC, the volume decreases and thus, the tension increases, after which it increases step by step on the second stroke and peaks on the third stroke at 720CA while the piston is at TDC; It gets close to 16 bar because the combustion chamber volume is so small, and on top of that, the EVCs are closed at the moment. During the exhaust stroke, the code shows a lower stress cost corresponding to the expansion at the end of the third stroke as the piston moves toward the BDC, the exhaust rig sliding.

ISSN: 2366-1313

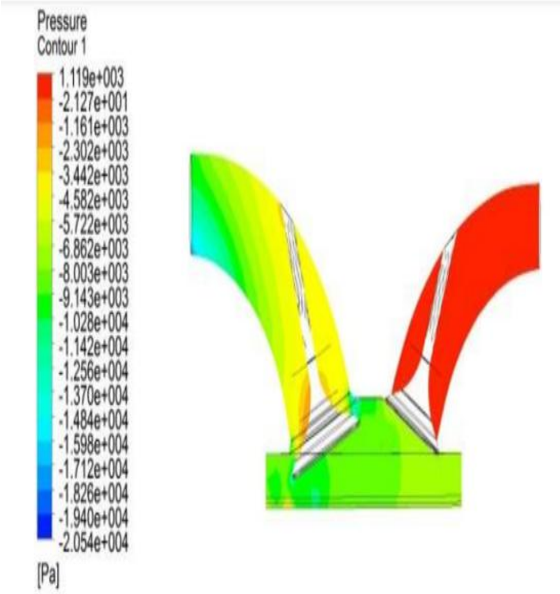


Fig.3 Pressure at 390 deg. Crank angle

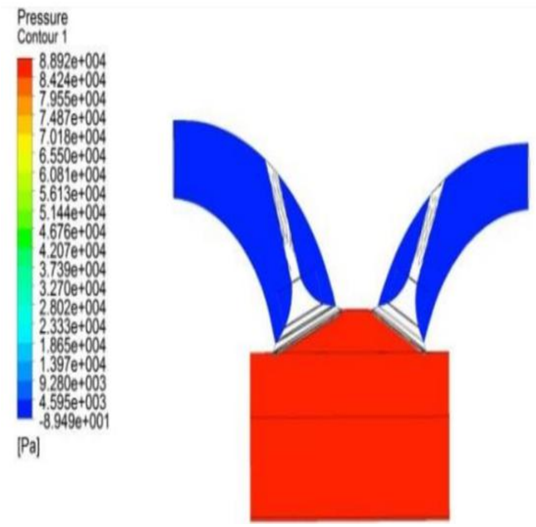


Fig.4 Pressure at 630 deg. Crank angle

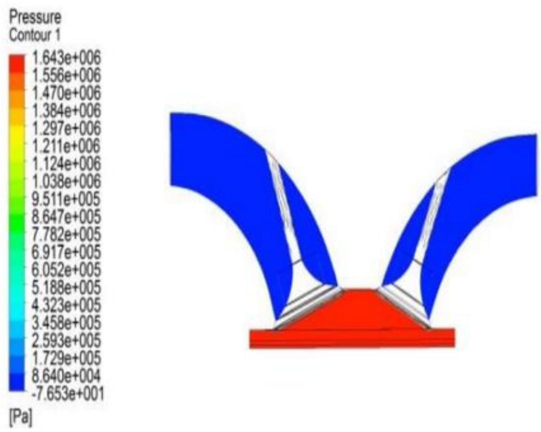


Fig.5 Pressure at 720 deg. Crank angle

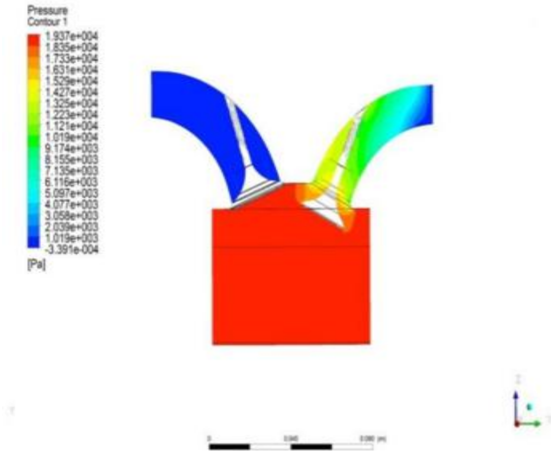
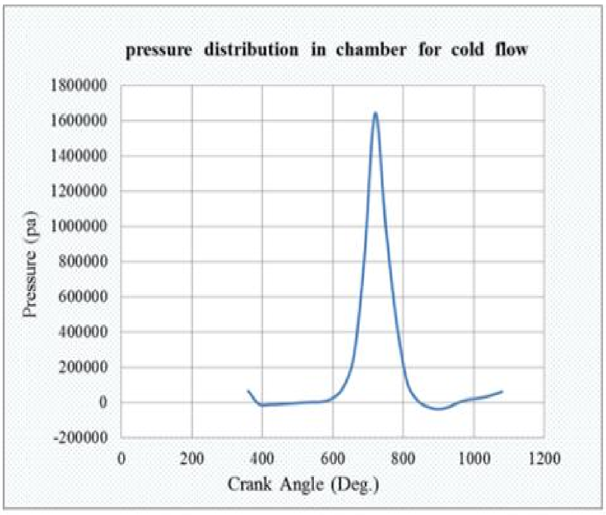


Fig.6 Pressure at 990 deg. Crank angle

Table.2 Pressure distribution in chamber for cold flow



In conjunction with pressure behavior during the engine operation, the temperature takes a similar behavior to the pressure, shows that the temperature is reduced in the intake port because of the pressure vacuums in the intake port at suction stroke at 390,420,450 CA. The maximum temperature at 720 CA is 705K and then reduces to approximately 411K at 900 CA where exhaust valve is partially opened

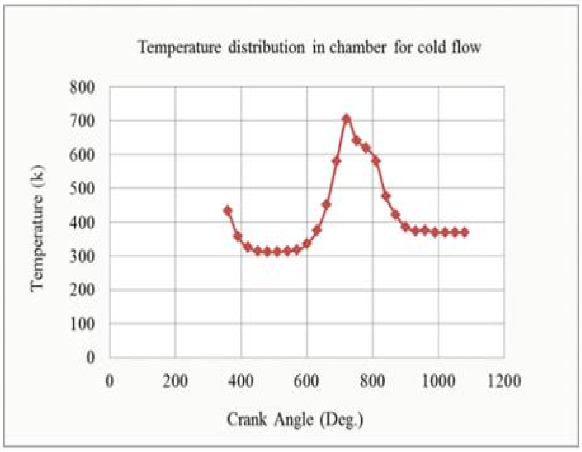


Fig.7 In –cylinder temperature trend of cold

Table.3 Final result

Pressure			Temperature			velocity		
	min	max		Min	max		Min	max
720 CA	- 1.728e	7.719e	390 CA	- 3.108e	1.646e	720 CA	0.000e	4.510e
900 CA	- 1.917e	1.427e	720 CA	3.000e	3.782e	750 CA	0.000e	2.510e
1020 CA	- 3.780e	1.890e	900 CA	2.891e	5.082e	780 CA	0.000e	1.744e

V. CONCLUSION

Using CFD ICE CODE, investigated the buoyancy characteristics of the SI engine. As a result, the following conclusions were drawn:

- Internal combustion engine optimization can enjoy the approved combustion simulation modelling method using the dynamic network method and ANSYS ICE CODE.
- The standard OK model equations have created an acceptable version of the kinetic wave motion simulation.
- These plates determine predicted directions within the cylinder with flow characteristics.
- The crank angle was used to plot temperature, pressure and velocity.
- This simplified CFD simulation no longer consists of reactions caused by engine component friction losses. Hence, the temperature and voltage values from the ignition simulation were slightly better than the experimental results.

REFERENCES

[1] Wendy and Abdullah, "Numerical analysis of the combustion process in a four-stroke compressed natural gas engine with direct injection system" Journal of Mechanical Science and Technology 22 (2008) 1937 -1944 Springer.

[2] Umakant and SunnapwarV"CFD Modeling and Experimental Validation of

Combustion in Direct Ig nition Engine Fueled with Diesel" International Journal Of Applied Engineering Research, Dindigul Volume 1, No 3, 2010.

[3] Pandey K.M and &Bidesh Roy, "CFD Analysis of Intake Valve for Port Petrol Injection SI Engine", Global Journal of Researches in Engineering Mechanical and Mechanics Engineering 2012.

[4] HiregoudarYerrennagoudaru, "Effect of Inlet Air Swirl On Four Stroke Single Cylinder Diesel Engine Performance" International Journal of Recent Development in Engineering and Technology 2014.

[5] Gaikwad D, "Experimental validation of combustion with CFD modeling in single cylinder four stroke CI engine fueled with biodiesel " Journal of Multidisciplinary Engineering Science and Technology November - 2014

[6] Patil A and Navale LG, "Experimental Verification and CFD Analysis of Single Cylinder Four Strokes C.I. Engine Exhaust System " international journal of science, spirituality, business and technology, vol. 3, no. 1, dec 2014

[7] Stefania Z, "Numerical analysis of methane direct injection in a single-cylinder250 cm3 spark ignition engine " 69th Conference of the Italian Thermal Engineering Association, ATI 2014 Science Direct.

[8] BasanagoudaBiradar, "Cold flow analysis of a single cylinder four stroke direct injection CI engine and analysis of volume fraction of air using CFD technique" International Research Journal of Engineering and Technology 2015

[9] Kolhe AV, "Combustion Modeling with CFD in Direct Injection CI Engine Fuelled with Biodiesel" Jordan Journal of Mechanical and Industrial Engineering 2015

[10] Gurram A and Veronika K, "Simulation of Combustion in Spark Ignition Engine " Journal of Basic and Applied Engineering Research 2015.