# 3D IN-CYLINDER COMBUSTION SIMULATION STUDIES IN AN IC ENGINE USING LAYERING APPROACH THROUGH CFD

[1]**B Kurmi Naidu** [2]**A Satish Kumar**

[1]PG student, AITAM Tekkali, Andhra Pradesh, INDIA
[2]Asst. Professor AITAM Tekkali, Andhra Pradesh, INDIA

**Abstract:** Fluid flow dynamics inside an engine combustion cylinder enables a better cylinder combustion, engine performance and efficiency. The In-Cylinder model of the software ANSYS Fluent is used in this paper to simulate 3D air motion with fuel combustion. Using CFD results, the flow phenomena can be visualized on 3D geometry and analyzed numerically. This allows you to compare different designs and which it helps to determine optimal designs. The engine used for this combustion simulation is a simple straight valve engine with two intake and two exhaust valves. Dynamic motion was visualize and velocity magnitude is plotted for crank angle starting from 570 to 833. The engine is simulated for only working stroke/combustion stroke. The text file is written in working directory containing swirl, x-tumble, y-tumble and moment of inertia as a function of CA. This tool automatically creates animations for mesh on cut plane, temperature on ice cut plane and velocity magnitude on ice cut plane. These created counters and graphs can show the performance of the engine used.

**Keywords:** CFD, In-Cylinder analysis, combustion Simulation, Swirl, IC Engine Analysis

## 1. INTRODUCTION

The design and manufacture of Internal Combustion (IC) Engines for getting better results includes so much difficulty. The next generation of engines needs to be compact, light, powerful, and flexible, yet produce less pollution and use less fuel. The central challenge in design is the complex fluid dynamics of turbulent reacting flows with moving parts through the intake/exhaust manifolds, valves, cylinder and piston.

Now a day's engineers are looking for new methodologies for improving the functionality of the product using various approaches which will reduce cost of research and give the desired results. We may consider the time scales of the intake air flow, fuel injection, liquid vaporization, turbulent mixing, species transport, chemistry, and pollutant formation. Computational Fluid Dynamics (CFD) has emerged as a useful tool in understanding the fluid dynamics of IC Engines for design purposes. CFD is such a technique which has emerged from last few decades as an important part in research of various systems finding its applications in near about every engineering discipline. This paper describes the use of CFD method for simulation and analysis of internal combustion engine combustion simulation process, which is tedious and complicated by experimentation. Using CFD methods we can get optimum results with no cost of experimentation and in less time. ANSYS/FLUENT is the CAE package which can be used for such simulations for accurate results and due to better meshing capabilities for sophisticated problems.

Insight provided by CFD analysis helps guide the geometry design of parts, such as ports, valves, and pistons; as well as engine parameters like valve timing and fuel injection. Using CFD results, the flow phenomena can be visualized on 3D geometry and analyzed numerically, providing tremendous insight into the complex interactions that occur inside the engine. This allows you to compare different designs and quantify the trade-offs such as soot vs NOx, swirl vs tumble and impact on turbulence production, combustion efficiency vs pollutant formation, which helps determine optimal designs.

In increasing order of complexity, the CFD analyses performed can be classified into Port Flow Analysis: Quantification of flow rate, swirl and tumble, with static engine geometry at different locations during the engine cycle. Cold Flow Analysis: Engine cycle with moving geometry, air flow, and no fuel injection or reactions. In-Cylinder Combustion Simulation: Power and exhaust strokes with fuel injection, ignition, reactions, and pollutant prediction on moving geometry. Full Cycle Simulation: Simulation of the entire engine cycle with air flow, fuel injection, combustion, and reactions.

# 2. COMBUSTION SIMULATION IN-CYLINDER ANALYSIS

Two approaches are employed in ANSYS FLUENT to solve in-cylinder (IC) problems, namely, hybrid approach and layering approach. The layering approach is used for engines with vertical valves like most diesel engines, while the hybrid approach is typically used for engines with canted valves like most spark ignited (SI) engines. There are three stages for IC simulation. Decompose the geometry into different zones and mesh them properly. By breaking up the model into different zones, it is possible to apply different mesh motion strategies to different regions in a single simulation. Set up the engine case in ANSYS FLUENT with the help of a setup journal. Perform a transient IC simulation. The simulation of the engine starts by import engine geometry. The imported geometry is divided into smaller volumes before meshing. This enables each volume to be meshed separately. Decomposition partitions a volume into sub-volumes and then the sub-volumes are meshed individually. Each volume will be meshed into hex or tetra elements, depending upon the approach. There are certain mesh topology requirements for valves and pistons. You should have pistons at TDC (top dead center) and valves at closed position before the geometry is decomposed. With the piston at TDC, the volume is the smallest.

## 2.1 TURBULENCE DUE TO SWIRL

In this paper we can choose an engine which is having a squish area at the top of the piston to create swirl. The "swirl" occurs because air attempting to rotate friction less around a cyclone that has low end musicale dimensions is really in cyclostrophic, as opposed to geostrophic, balance. In essence, air attempting to move into a low pressure area is not able to because of strong centrifugal accelerations.

When the air is pumped into the combustion chamber during the intake cycle, it passes through the gap between the valve and the valve seat. This jet imparts angular momentum, known as swirl and tumble, to the fresh charge. When the piston travels back up towards the top during the compression stroke, most of the energy contained in the tumble (or angular momentum orthogonal to the swirl axis) of the jet is converted to turbulence as the available space in the vertical direction is reduced significantly. The swirl will become stronger as the air is squeezed out to the side. If there is a narrow region between the piston and the cylinder head, the air may be squeezed (or "squished") from the sides of the cylinder into the combustion chamber, converting the energy in the swirl into turbulence. Creating a swirling vortex in the cylinder has been recognized as a way of enhancing turbulence levels during compression stroke since the early days of IC engines. This will increase the turbulence late in the compression stroke.

## 2.2 EFFECT OF SWIRL RATIO

The swirl ratio actually is a ratio of the circulation around the periphery of the vortex to the updraft strength, and, as such, is a measure of the amount of mass rotating around the circulation center at a given level. Large swirl ratios imply increasing amount of mass, and if the vertical velocities are not strong enough to "evacuate" that mass from a given level, then the circulation breaks down into multiple circulation centers. The swirl ratio itself is also a measure of the intensity of each circulation center. Hence, the larger the swirl ratio, the stronger the circulation center, the stronger the pressure fall at the center.

Mathematically, the swirl and tumble ratios (in a non dimensional form) are calculated by the effective angular speed of in cylinder air motion divided by the engine speed and the effect angular speed is the ratio of the angular momentum to the angular inertia of moment.

# 3 CFD METHODOLOGY

The CAD model in native format can be imported to ANSYS/Fluent. After importing the CAD file the engine geometry is meshed into different volumes separate mesh is used for separate volumes in the engine. The engine model can be as shown below.
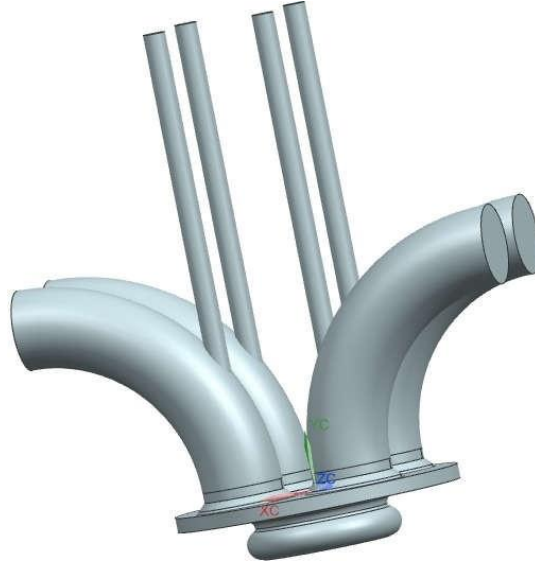


**Fig -1 Basic Geometry**

The boundary conditions for the analyses can be shown in the table1.

**Table-1 Boundary Conditions**

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

## 3.1 CASE DESCRIPTION

The engine in this paper is four stroke single cylinder petrol engine with straight valves two inlet and two exhaust valves. It is a in-cylinder engine having piston and cylinder in in line. The model is prepared with cad software meshing is carried out. The case file is imported in ANSYS/Fluent (3D). This paper considers a 3D symmetric geometry of a IC engine cylinder configuration. Case setup is performed using a scheme file that automatically sets up necessary motions for valves and pistons along with solution parameters found to be best suit for the in-cylinder simulation. The journal file automatically sets up the necessary motions for valves and piston, along with solution parameter which suit the IC simulation best. Tetrahedral mesh is used in the upper combustion chamber and ports to facilitate the setup. We are using sector decomposition option to simplify the solution. The mesh must have the correct decomposition and names before using the journal to automatically set up the case.
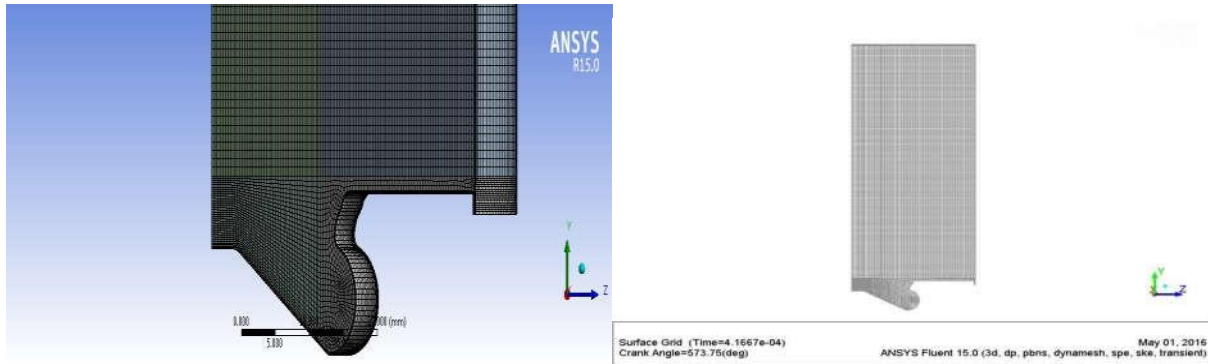
**Fig–2: The Mesh for the Engine Model**

**Table-2 Engine System Inputs**

| S.No | Parameter | value |
|------|-----------|-------|
| 1 | Engine speed(rev/min) | 1500 |
| 2 | Crank Radius (mm) | 55 |
| 3 | Piston pin offset/wrench (mm) | 0 |
| 4 | Connecting Rod Length (mm) | 165 |

# 4 RESULTS

The various parameters used for the calculation of the engine performance are given below.

Where,

CA = Crank Angle.

L = Angular momentum vector of fluid mass contained in selected cell zones with respect to swirl center.

sa = Swirl axis.

tx = Tumble x-axis.

ty = Tumble y-axis.

Isa = Moment of Inertia of the fluid mass about swirl axis.

Itx = Moment of Inertia of the fluid mass about tumble x- axis.

Ity = Moment of Inertia of the fluid mass about tumble y- axis.

The below figure shows the temperature variations of a particle moved from the spark plug. The tracing can be carried through different colors.
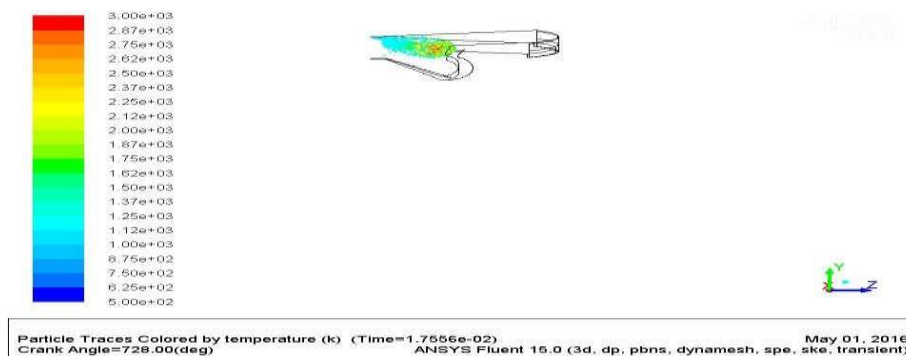


**Fig-3 particle traces by colored by temperature at CA 728.00 (deg)**

## COUNTERS OF STATIC TEMPERATURE AT DIFFERENT CRANK ANGLES

The below counters shows the different temperatures at different crank angles. We can plot the counters at the crank angles like 570.25, 644.00, 664.00, 700.00, 712.00, and 728.00. Which is all these crank angles in the combustion stroke only.
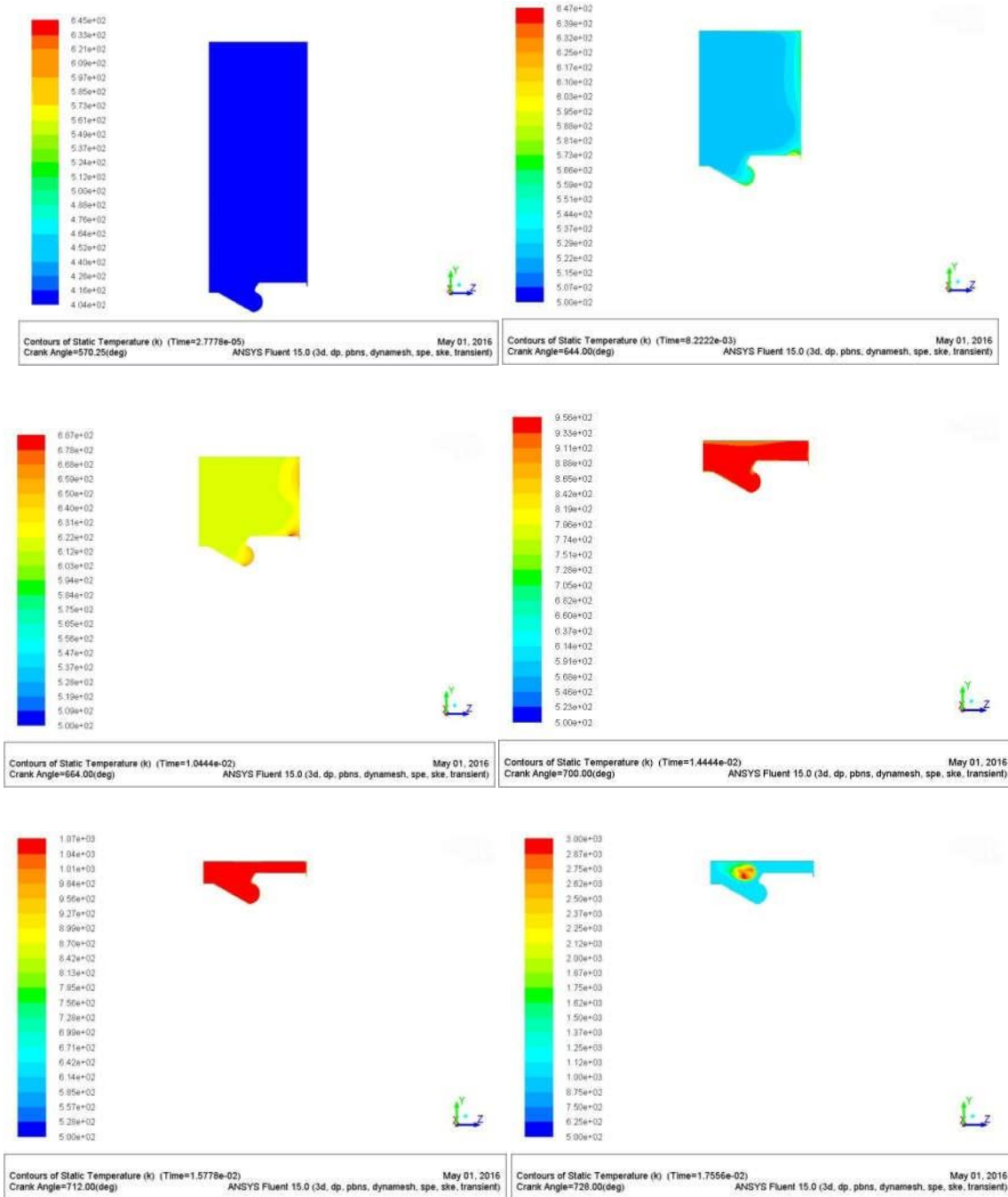


**Fig-4 Counters of Static Temperatures at Different Crank Angles**

**COUNTERS OF VELOCITY MAGNITUDE AT DIFFERENT CRANK ANGLES**
The below counters shows the velocities of particles at different crank angles. We can plot the counters at the crank angles like 570.25, 644.00, 664.00, 700.00, 712.00, and 728.00
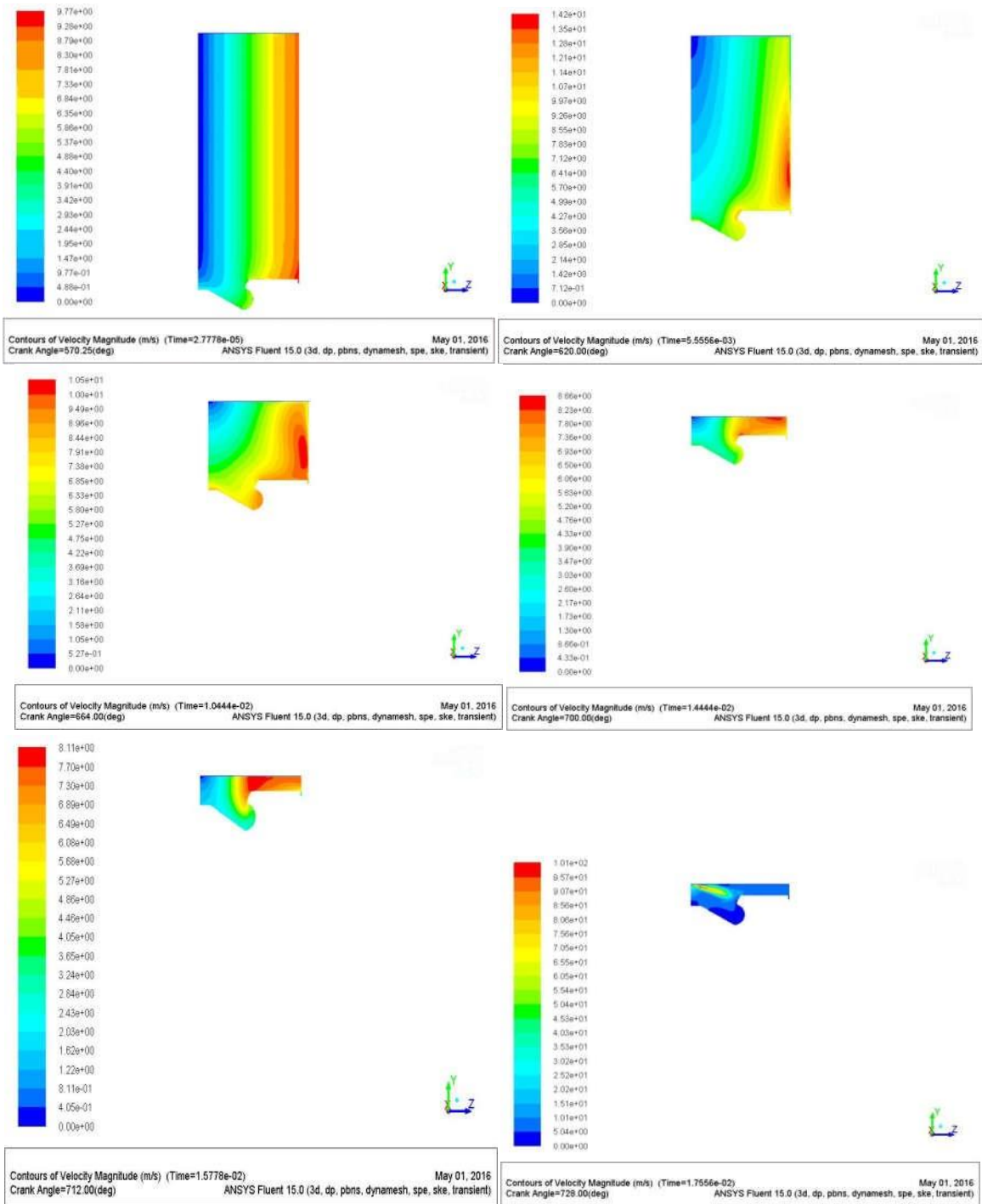


**Fig-5 counters of velocity magnitude at different crank angles**
We can also know the performance of an engine model by using following graphs. Swirl ratio vs crank angle graph shows how the ratio at different crank angles.
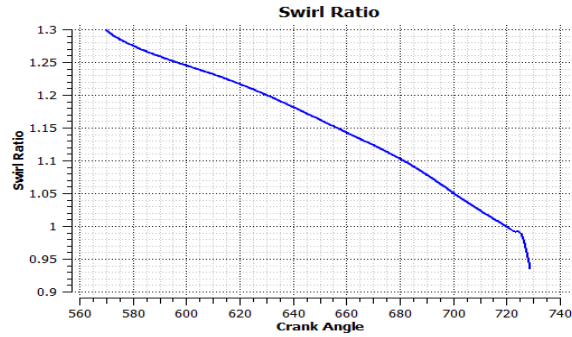
**Chart-1 swirl ratio vs crank angle**

We can easily identify where the maximum amount of heat can be released. The maximum amount of heat can be released in between the crank angles 720-730.
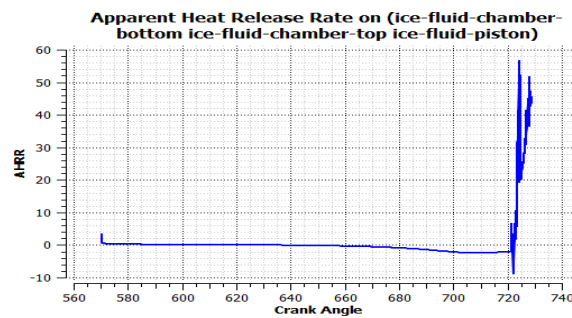


**Chart-2 apparent heat release rate vs crank angle**

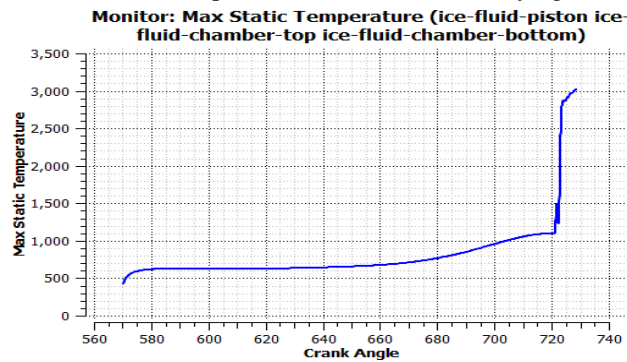The below graphs shows the variations of static temperature and velocities at varying crank angles.
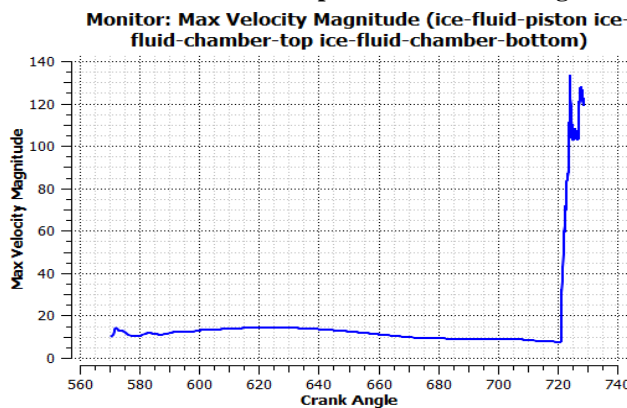


**Chart-3 max static temperature vs crank angle**



**Chart-4 max velocity magnitude vs crank angle**

# 5  CONCLUSION

The simulation is carried out using parameter and journal files which is a geometry having a squish area at the top of the piston. Dynamic motion was visualize and velocity magnitude is plotted for crank angle starting from 570 to 833. The engine is simulated for full cylinder cycle. The In cylinder data file is write displaying swirl and tumble for zones of fluid-ch and fluid-piston-layer. The text file is written in working directory containing swirl, x-tumble, y-tumble and moment of inertia as a function of CA. 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.

## *REFERENCES*

[1] Pathak Yogesh R, Deore Kailas D, Patil Vijayendra M , " In        Cylinder Cold Flow CFD Simulation Of IC Engine Using Hybrid Approach" IJRET: International Journal of Research in Engineering and Technology; Volume: 03 Special Issue: 08 | NCAME-2014

[2] A Lakshman, C P Karthikeyan and R Padmanabhan "3D In-cylinder Cold Flow Simulation Studies in an IC Engine using CFD" International journal of research in mechanical engineering volume1, july-september, 2013

[3] Feilong Liu, Gehan A. J. Amaratunga and Nick Collings and Ahmed Soliman "An Experimental Study on Engine Dynamics Model Based In-Cylinder Pressure Estimation" SAE International 2012-01-0896 published 04/16/2012.

[4] Mathur M.L. and Sharma R.P., DhanpatRai publication.

[5] ANSYS user's guide, ANSYS, Inc. Southpointe, Canonsburg, PA 15317, 2012.

[6] Lakshman A, Karthikeyan CP and Davidson Jebaseelan (2012). CFD studies on In-cylinder air motion during different strokes of an IC Engine. SET Conference 2012 VIT Univerity Chennai.

[7] DivyanshuPurohit, Pragya Mishra, Vishwanath Banskar," Flow Simulation of an I.C. Engine in FLUENT, ANSYS 14.0" , International Conference On Emerging Trends in Mechanical and Electrical Engineering (ICETMEE- 13th-14th March 2014)

[8] Ganeshan. V Advanced I.C. Engines 3rd ed, McGraw- Hill: 200