



IJRASET

International Journal For Research in
Applied Science and Engineering Technology



INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 Issue: VIII Month of publication: August 2017

DOI: <http://doi.org/10.22214/ijraset.2017.8326>

www.ijraset.com

Call:  08813907089

E-mail ID: ijraset@gmail.com

3D In-Cylinder Combustion Simulation Comparison Between with Squish and without Squish Area

B Kurmi Naidu¹, A Srinivasa Rao², A Satish Kumar³

¹M. Tech student, ²Assoc. Professor, ³Asst. Professor, AITAM Tekkali, Andhra Pradesh, INDIA

Abstract: Fluid flow dynamics inside an engine combustion cylinder permits 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. The flow phantasm can create a 3D geometry and also the numerical analysis by using CFD results. By doing like this, we have a chance to compare different designs and this comparison helps to determine the best suited optimal designs. The engine used for this combustion simulation is a simple straight valve engine with two intake and two exhaust valves. The piston moment can be visualized through the animation and velocity magnitude curves are 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. The graphs indicate that the piston with squish area can maintain high velocities and produce maximum temperatures and gives maximum energy.

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

I. INTRODUCTION

It is very difficult to getting better results in the internal combustion engines. Because it includes so much of losses and we need to calculate these results by experimentation. Very compact, light, powerful, and flexible engines are needed for the next generation, though it produces less pollution and using less fuel. Fluid dynamics of turbulent reacting flows with moving parts through the intake/exhaust manifolds, valves, cylinder and piston is the main challenge in design. Present all the engineers are looking for the best method for improving the working of the product using various approaches which will reduce cost of research and give the desired results. We may consider the time taken for of the intake air flow, fuel injection, liquid fuel vaporization, turbulent mixing of fuel and air and pollutant formation.

To overcome all these problems Computational Fluid Dynamics (CFD) has appeared as one of the best useful tool to understand the fluid dynamics of IC Engines for design purposes. From the last few decades, CFD is such a technique which has developed in research of different systems finding its applications in entire engineering discipline. In this present paper, I describe the use of CFD method for simulation and analysis of internal combustion engine by using IC Engine tool. Using CFD methods we can get required results with no cost of experimentation and within less time. I think ANSYS/FLUENT is the best software for such a simulation forgetting accurate results and due to better meshing capabilities for sophisticated problems.

The analysis prescribed by CFD helps to guide the geometry design of parts, such as ports, valves, and pistons; likewise engine parameters like valve timing and fuel injection system. The flow phenomena can be visualized on 3D geometry Using CFD results and also the numerical results can be calculated inside the engine. These results are used for comparing different engine designs and quantify the different trade-offs like soot vs NOx, swirl vs tumble and impact on turbulence production, combustion efficiency vs pollutant formation. These are all helps to determine optimal designs.

The CFD analyses performed can be classified into three types by based on the increasing order of their complexity out of which the first analysis is Port Flow Analysis. In which Quantification of flow rate, swirl and tumble, with static engine geometry at different locations during the engine cycle. The second analysis is Cold Flow Analysis. In this we can perform different actions like the Engine cycle with moving geometry, air flow, and no fuel injection or reactions. And the last analysis is In-Cylinder Combustion Simulation. In this analysis we can consider the 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.

II. COMBUSTION SIMULATION IN-CYLINDER ANALYSIS

To solve in-cylinder (IC) problems in ANSYS FLUENT, we have mainly two approaches. They are 1) hybrid approach and 2) layering approach. The layering approach is used for engines which having vertical valves like diesel engines and the hybrid approach is used for engines which are having canted valves mostly for spark ignited (SI) engines. There are mainly 3 stages for Internal Combustion engine simulation. The first stage is to decompose the geometry into different zones and sub nodes. The second stage is meshing. The decomposed geometry is used to apply mesh for different motion strategies to different regions in a single simulation. The third stage is to set up the engine case in ANSYS FLUENT with the help of a setup journal. Here we can perform a transient IC simulation. Importing the engine geometry is the starting for combustion simulation. The imported geometry is then divided into smaller volumes by using the decomposition tool. This decomposition performs every volume in the engine to be meshed separately. The main function of the Decomposition is to divide a volume into sub-volumes and then the sub-volumes are meshed properly. Every volume will be meshed into hex or tetra elements, depending upon the approach. Before the geometry is decomposed you should maintain the pistons at TDC (top dead center) and valves at closed position. When the piston is placed at TDC, less volume is remained.

A. 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. Air in this area attempting to rotate around a cyclone due to that there creates a "swirl" the air attempting to move into a low pressure area is not able to because of strong centrifugal accelerations. The air is pumped into the combustion chamber during the intake cycle and it passes through the space between the valve and the valve seat. This jet creates the angular momentum, known as swirl and tumble, to the fresh charge. When the piston moves up towards the top during the compression stroke, most of the energy contained in the tumble of the jet is converted to turbulence due to that the available space in the vertical direction is reduced. The swirl will become stronger as the air is squished out to the side. If there is a narrow region between the piston and the cylinder head, the air may be squished from the sides of the cylinder into the combustion chamber, by converting the swirl energy into turbulence.

To enhance turbulent levels, one of the best methods is to create a swirling vortex in the cylinder during compression stroke. This will increase the turbulence rate in the compression stroke.

B. Effect of Swirl Ratio

The swirl ratio is defined as the angular velocity of a solid body rotating flow which has equal angular momentum to the actual flow divided by the crank shaft angular rotational speed. 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 measure of intensity of each circulation center is nothing but a swirl ratio. Hence, the larger the swirl ratio, the stronger the circulation center, the stronger the pressure fall at the center.

Mathematically, the effective angular speed of in cylinder air motion is divided with the engine speed can give the swirl and tumble ratio. Here the effect angular speed is the ratio of the angular momentum to the angular moment of inertia.

III. CFD METHODOLOGIES

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.

The comparison of model designs for the combustion chamber with squish area and the combustion chamber without squish area are as shown in below fig. The having this squish area increases the area of combustion chamber. By increasing the combustion area we can get the maximum result with high temperatures and due to this we can get high velocities for piston movements.

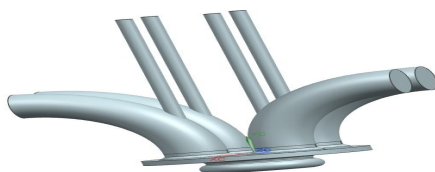


Fig -1 Basic Geometry with Squish



Fig -2 Geometry without Squish

The analysis can be carried on both the designs with the same boundary conditions and the same specified conditions. The boundary conditions for the analyses can be shown in the table 1.

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

A. Case Description

The engine design used in this paper for combustion simulation is four stroke single cylinder diesel engines with two inlets and two exhaust straight valves. It is an in-cylinder engine having piston and cylinder in line. The model is prepared with solid works software. The engine modeled for this study is imported in ANSYS/Fluent (3D). To set up necessary motions for valves and pistons along with solution parameters case set up is used. Along with solution parameter the journal file creates the required motions for valves and piston. 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. Before using the IC engine solver settings and the boundary conditions the mesh must have the correct decomposition and name

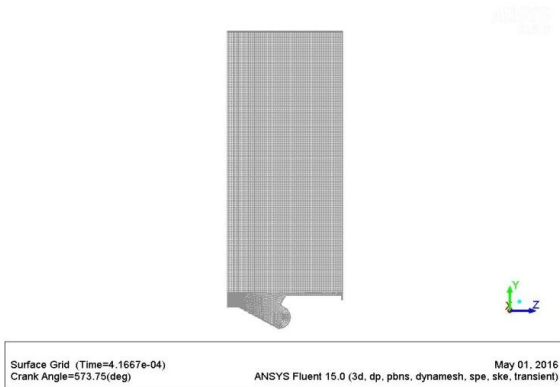


Fig-3: The Meshing with squish

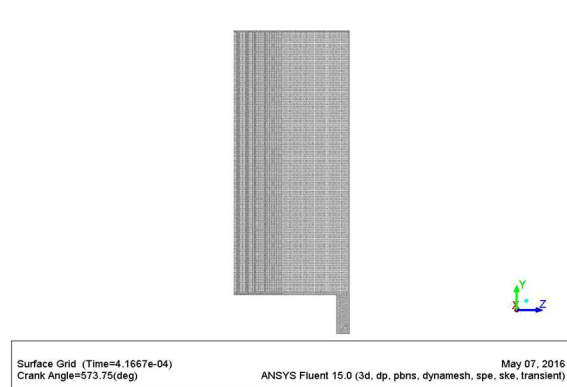


Fig-4: The Meshing without squish

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

IV. RESULTS

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

CA means - Crank Angle.

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

Sa means - Swirl axis.

Tx means - Tumble x-axis.

Ty means - Tumble y-axis.

Isa means - Moment of Inertia of the fluid mass about swirl axis.

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

It means - 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.

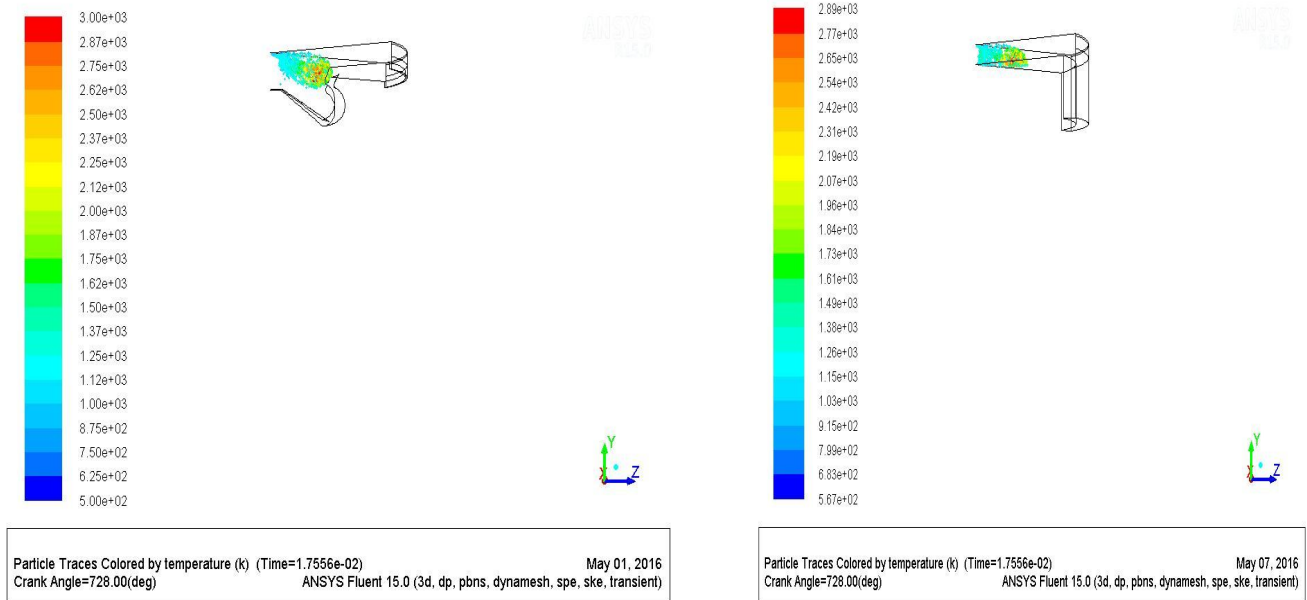
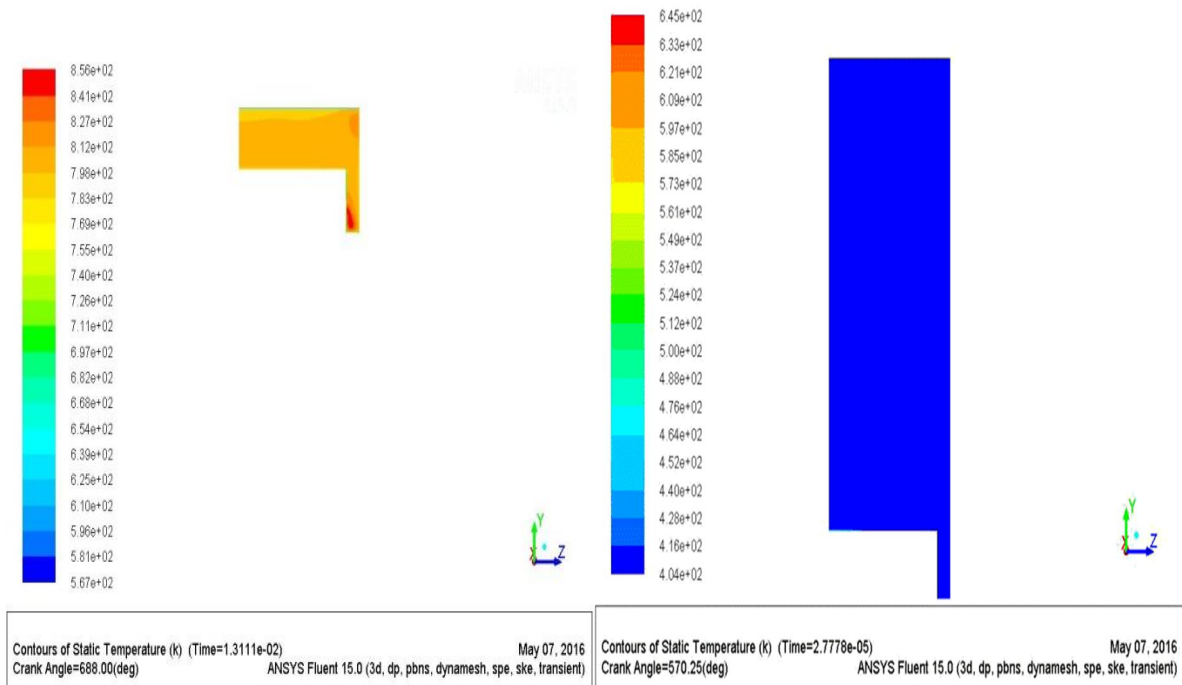


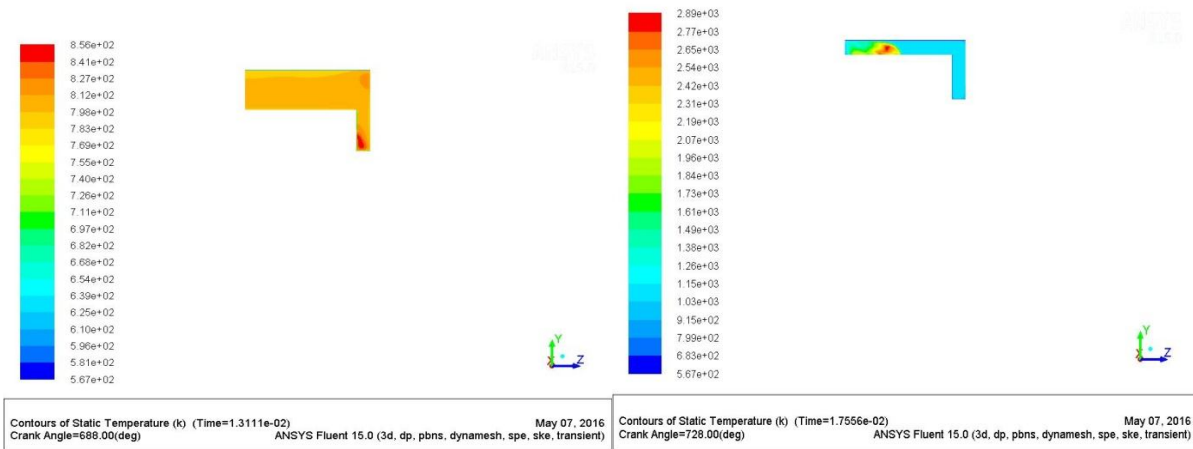
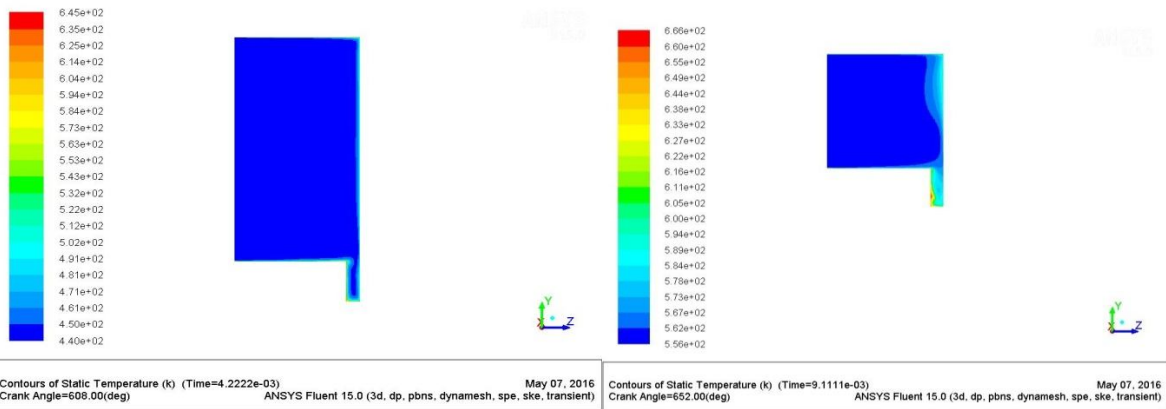
Fig-5 particle traces by colored by temperature at CA 728.00 (deg), Fig-6 particle traces by colored by temperature at CA 728.00 (deg)

A. 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.

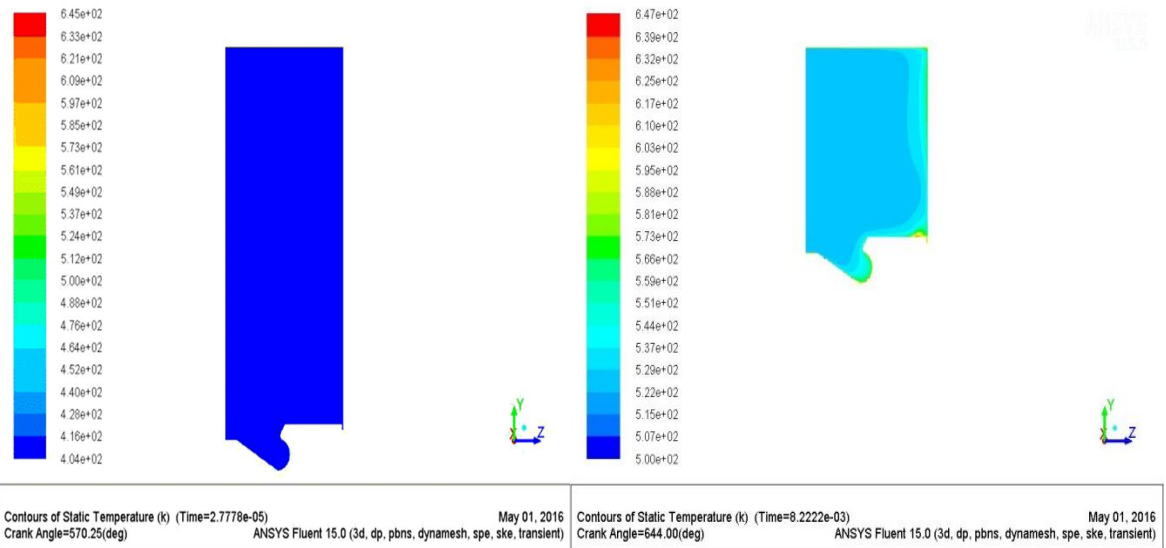
Counters of static temperature of the design without squish area





B. Counters of Static Temperatures at Different Crank Angles for a Cylinder without Squish Area

Counters of static temperature of the design without squish area



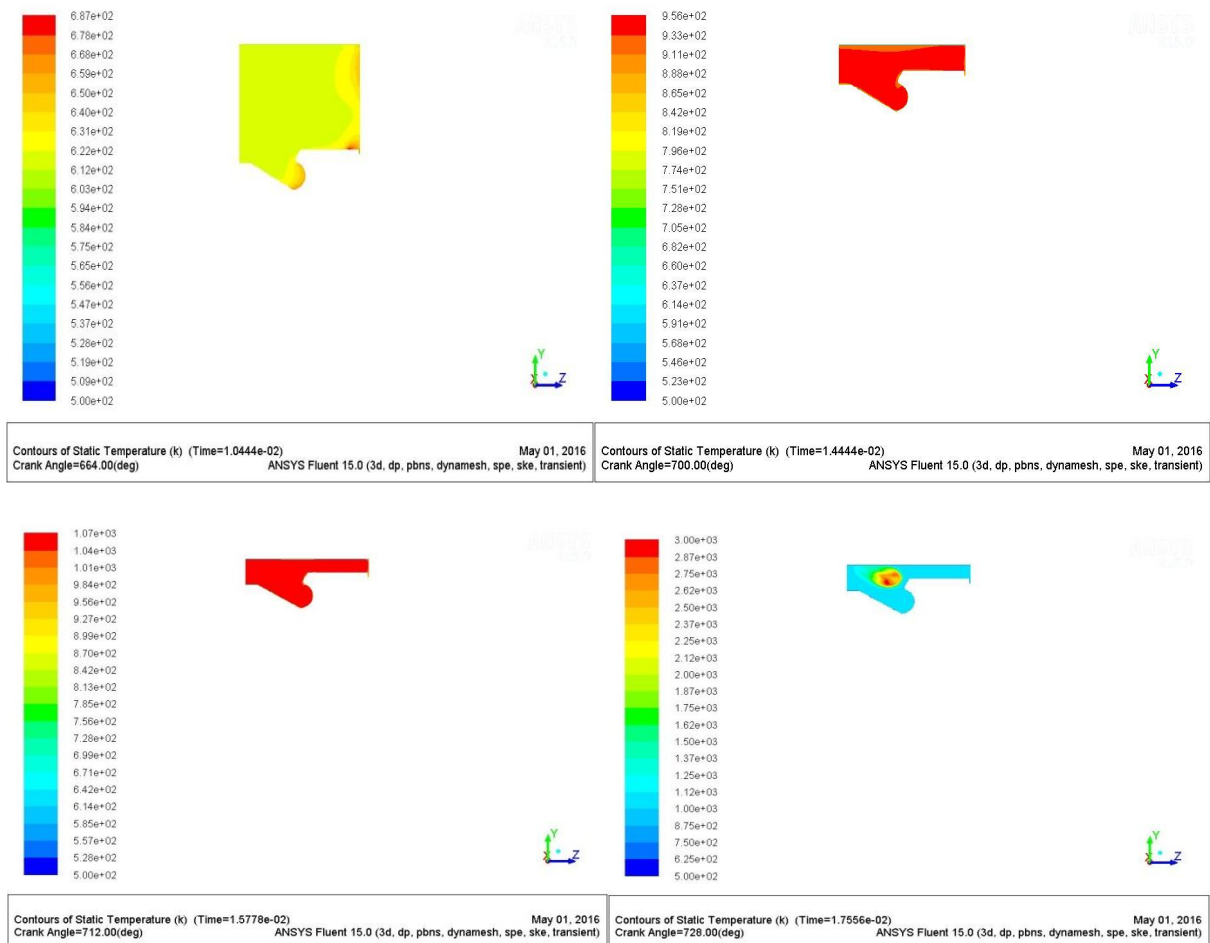
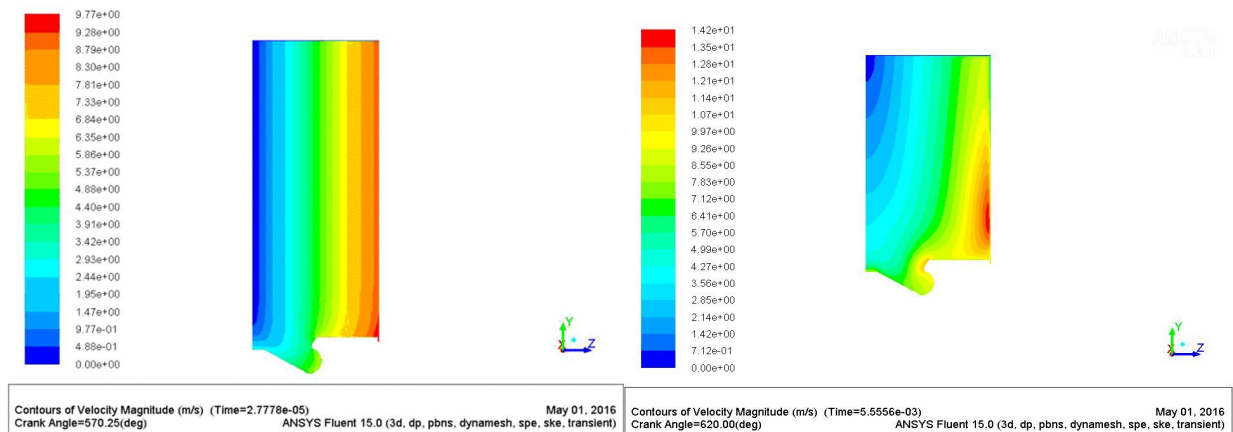


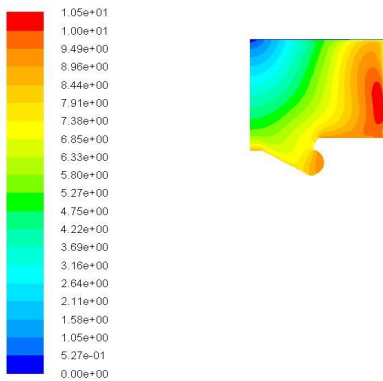
Fig-4 Counters of Static Temperatures at Different Crank Angles

By comparing above two diagrams at the crank angle 555 degrees, the combustion chamber with squish area having high static temperature compare to the combustion chamber without squish area. Remaining all the diagrams are with same results i.e. high static temperatures.

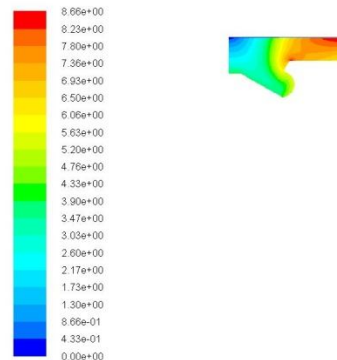
C. Counters of Velocity Magnitude at different Crank Angles

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

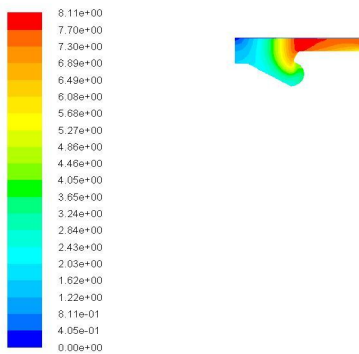




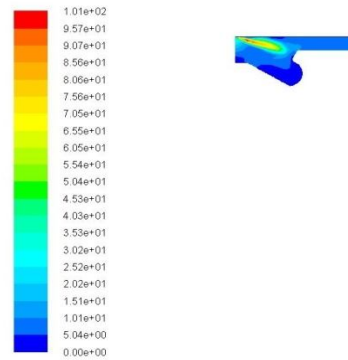
Contours of Velocity Magnitude (m/s) (Time=1.0444e-02) May 01, 2016
Crank Angle=664.00(deg) ANSYS Fluent 15.0 (3d, dp, pbns, dynamesh, spe, ske, transient)



Contours of Velocity Magnitude (m/s) (Time=1.4444e-02) May 01, 2016
Crank Angle=700.00(deg) ANSYS Fluent 15.0 (3d, dp, pbns, dynamesh, spe, ske, transient)



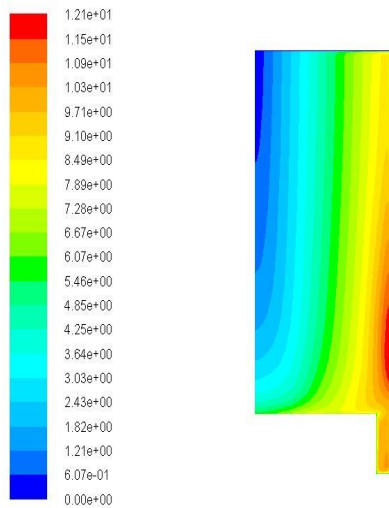
Contours of Velocity Magnitude (m/s) (Time=1.5778e-02) May 01, 2016
Crank Angle=712.00(deg) ANSYS Fluent 15.0 (3d, dp, pbns, dynamesh, spe, ske, transient)



Contours of Velocity Magnitude (m/s) (Time=1.7656e-02) May 01, 2016
Crank Angle=728.00(deg) ANSYS Fluent 15.0 (3d, dp, pbns, dynamesh, spe, ske, transient)



D. Counters of Velocity Magnitude for the Design Without Squish Area

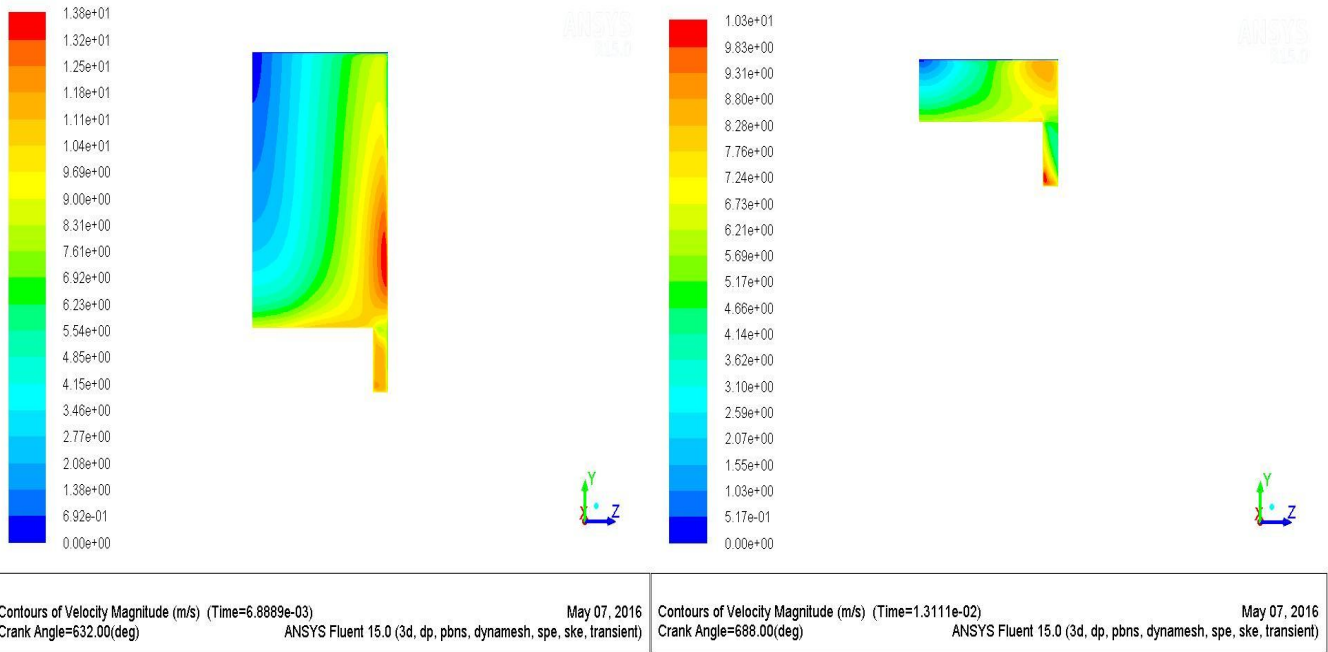


Contours of Velocity Magnitude (m/s) (Time=2.8889e-03) May 07, 2016
Crank Angle=596.00(deg) ANSYS Fluent 15.0 (3d, dp, pbns, dynamesh, spe, ske, transient)



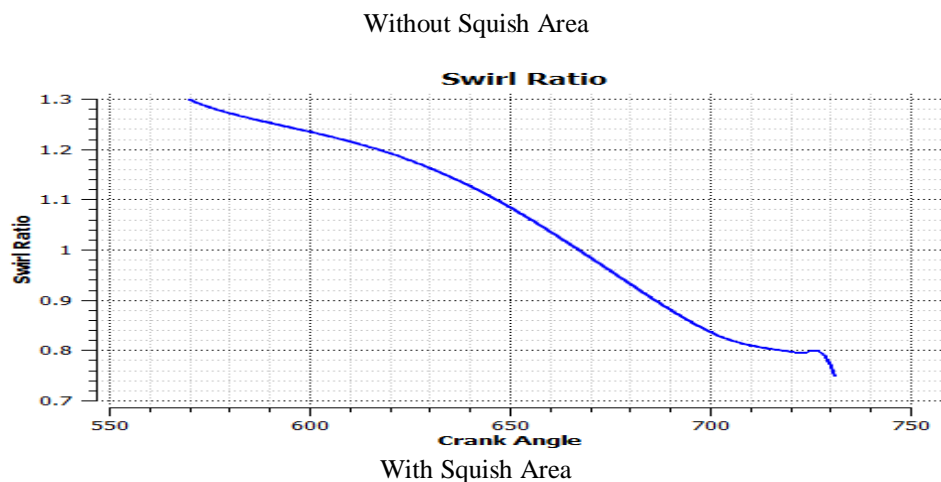
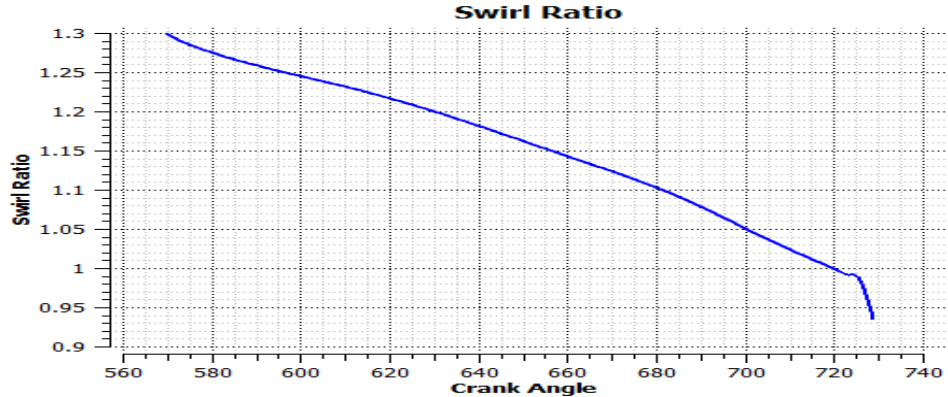
Contours of Velocity Magnitude (m/s) (Time=1.5778e-02) May 07, 2016
Crank Angle=712.00(deg) ANSYS Fluent 15.0 (3d, dp, pbns, dynamesh, spe, ske, transient)



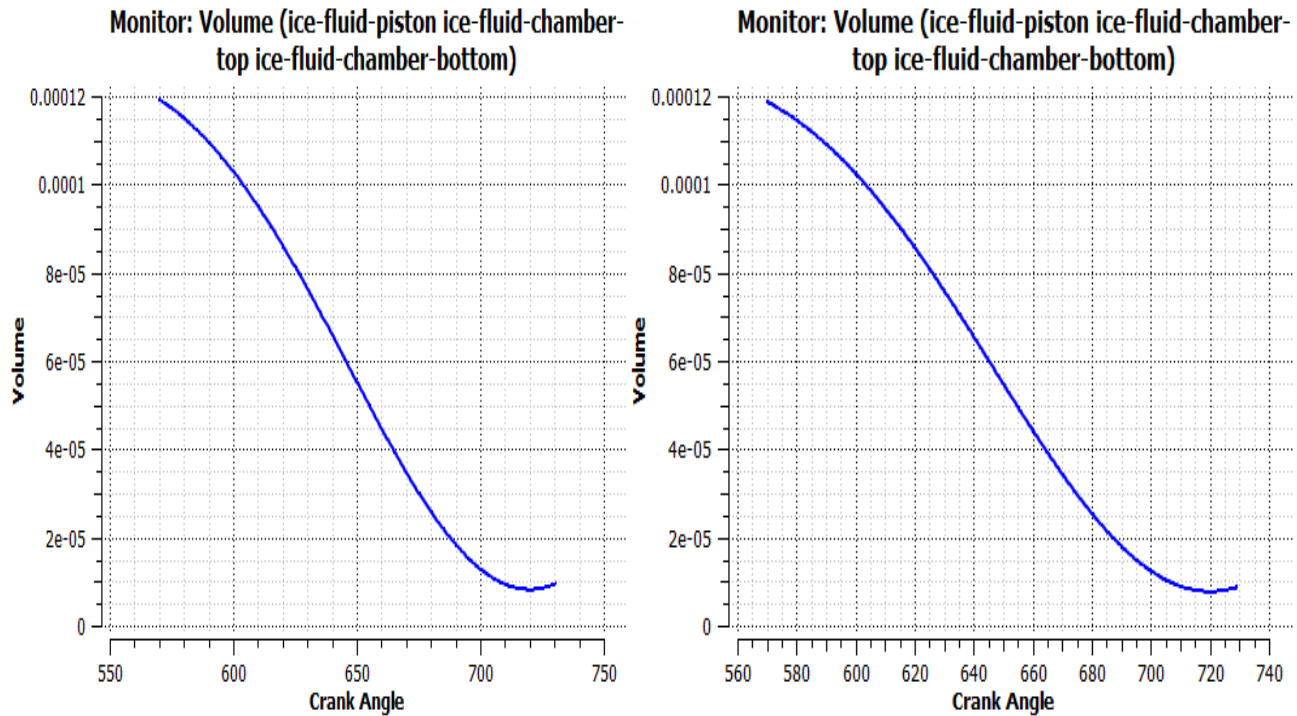


E. Counters of Velocity Magnitude at different Crank Angles for a Cylinder Without Squish Area

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.



1) *Chart-1 Swirl Ratio Vs Crank Angle*: The combustion chamber with squish area having high swirl ratio compare to the combustion chamber without squish area. Creating this area burnt the remaining gases in the combustion chamber.



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.

Apparent Heat Release Rate on (ice-fluid-chamber-bottom ice-fluid-chamber-top ice-fluid-piston)

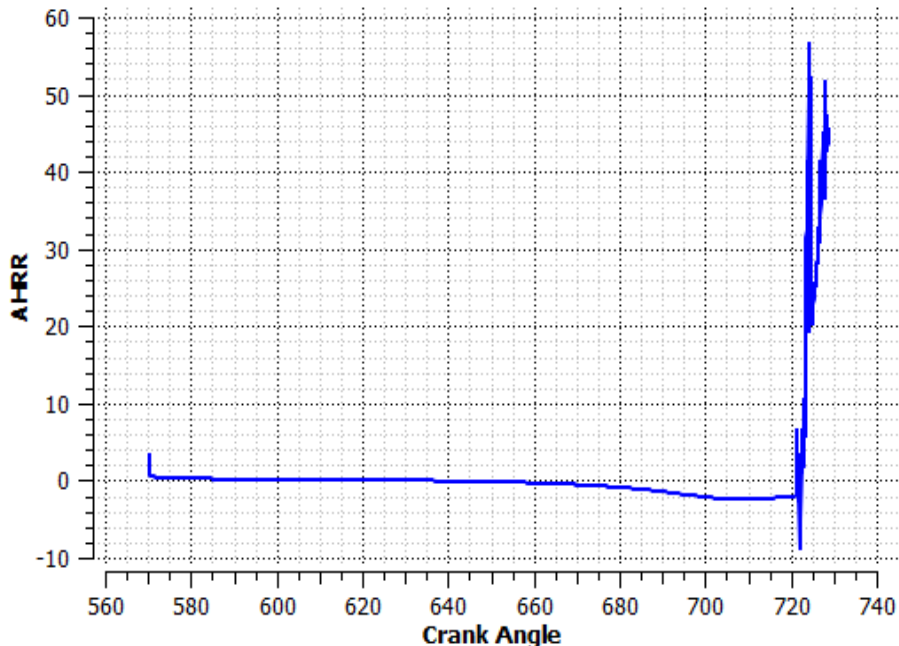


Chart-2 apparent heat release rate vs crank angle

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

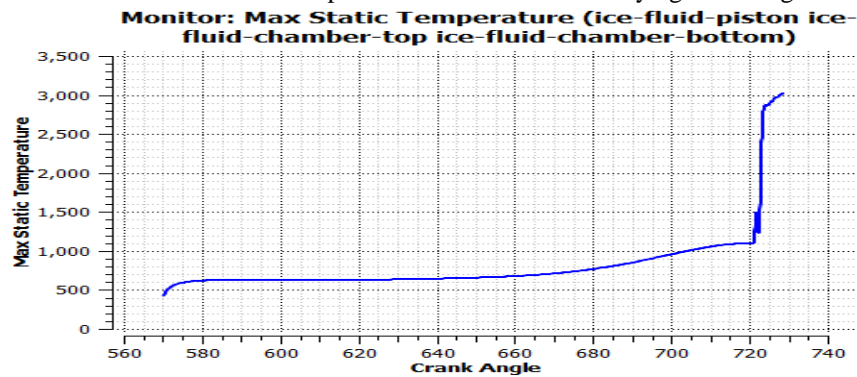


Chart-3 max static temperature vs crank angle

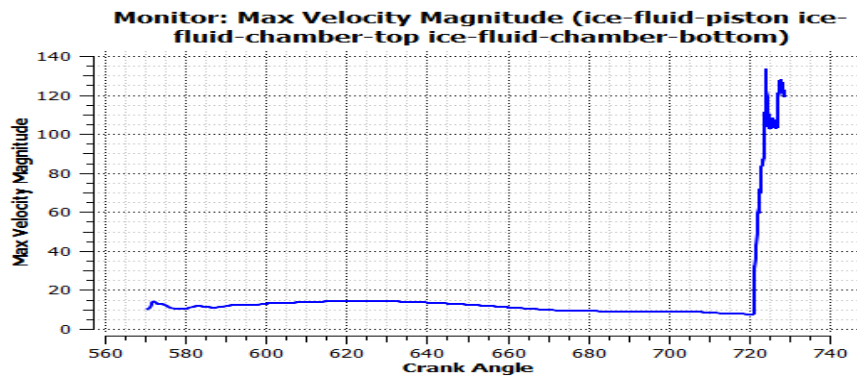


Chart-4 max velocity magnitude vs crank angle

V. CONCLUSION

The combustion simulation is carried by using a geometry having a squish area at the top of the piston. The piston moment can be visualized through the animation and the velocity magnitude curves are plotted for different crank angles starting from 570 to 833. The designed single cylinder engine is simulated for a 60° sector angle cylinder cycle. The In cylinder combustion simulation data file is displaying swirl and tumble for zones of fluid-ch and fluid-piston-layer. The text file for swirl, x-tumble, y-tumble and moment of inertia as a function of crank angle is written in the working directory. The graphs indicates that the cylinder with squish area gives the maximum heat which gives the maximum power. This process can be repeated for different input designs and compare the results with one another up to getting the best design for our requirement.

REFERENCES

- [1] PathakYogesh R, Deore Kailas D, PatilVijayendra 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., internal combustion engines 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 University Chennai.
- [7] DivyanshuPurohit, Pragma Mishra, VishwanathBanskar , " 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)



10.22214/IJRASET



45.98



IMPACT FACTOR:
7.129



IMPACT FACTOR:
7.429



INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089  (24*7 Support on Whatsapp)