CFD ANALYSIS OF IN-CYLINDER FLOWS IN SINGLE CYLINDER SI ENGINE

Yogendra Aniya[1], Sharad Chaudhary[2]

[1]PG Scholar, [2]Professor
Mechanical Engineering Department Devi Ahilya Vishwavidyalaya, Indore,

***Abstract*: In today's scenario efficiency of engine and emission is major focus area due to the operating cost and stringent emission norms. In order to achieve present requirement of continuous improvement in existing system innovative design of engine and appropriate study of involved processes required. Many parameters involve in the assessment of Internal Combustion Engine emission, performance and effectiveness including geometrical parameters and operating conditions. In-cylinder flow is one of which we consider in our study and it is depend upon valve location, types and piston shape. Experimental techniques are difficult and not possible by considering different configurations due to complication, cost and required time.**

CFD present flexibility to study fluid flow inside of engine cylinder with different configuration by using boundary condition, this study explores the effects of piston bowl geometry on tumble flow. Based on the model validation, the in-cylinder flow and tumble formation are simulated and analyzed for different piston shapes. The results show that the bowl on the top of piston play huge role for tumble formation.

***Index Terms*: CFD (Computational fluid dynamics), Tumble Flow, In-cylinder Flow**

I. INTRODUCTION

In-cylinder flow play significant role for engine performance as well as exhaust emission. It is ensuring proper mixing and burning of fuel at maximum possible instance, depends upon geometrical parameter & operating condition of engine. Geometrical parameter influence more than operating parameter for performance of engine. Major geometrical parameters are stroke of engine, piston shape, compression ratio and valve geometry. Consideration of all at single instance is not possible due to many permutations an combination hence our focus is on modification of piston geometry by keeping other parameter constant.

(researchgate.net)

The piston is a component which slides from TDC to BDC inside the cylinder. Its purpose is to transfer impulse force of burnt gases and transfer power to crankshaft.
Objective of this study is

- Study of In-cylinder flow.
- Study of parameters affecting In-cylinder flow.
- Review of existing piston geometries.
- Study the effect of the piston geometry on in-cylinder flows using CFD.

II. LITERATURE REVIEW

This study consist computational fluid dynamics analysis perform on In-cylinder flow to find out charge velocity, velocity contour pattern for tumble and turbulence kinetic energy by modifying piston geometry.

- Understand In-cylinder flow and its types.
- Understand In-cylinder flow effect on engine.

In-cylinder flow

In-cylinder flow is representing fluid motion inside the engine cylinder and it is depend upon valve geometry, valve lifting, valve position and piston shape. Due to high initial velocity all in and out flow are turbulent in cylinder and It is essential to have well organized movement of air / air+ fuel within the combustion chamber for –

- Speedy evaporation of fuel
- To enhance air fuel mixing
- To increase combustion speed and
- To increase efficiency.

Turbulence in IC engines is mainly due to Swirl, Squish and Tumble

Swirl:

In-cylinder flow in which fluid rotation is parallel to axis of cylinder is defines swirl. It is achieved by ensuring tangential flow inside the cylinder with the help of intake manifold, valve ports and piston geometry. It is manly required for compression ignition engines with the help of intake system and play important role for complete combustion.

Squish:

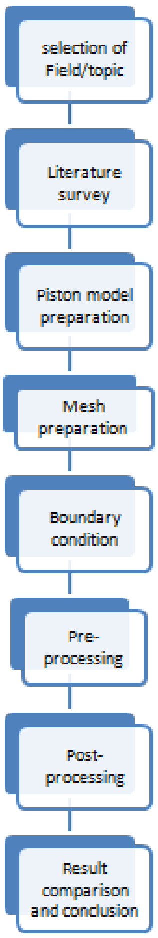
Air or air /fuel mixture movement in radial direction is defined as squish, and it is develop during compression stroke by piston movement towards TDC due to flow of charge inside the combustion bay by squeezing it out from between the piston and cylinder head during the end of compression stroke.

Tumble:

Movement of air/fuel perpendicular axis of cylinder head is known as tumble flow and it is generated by squish motion of charge or piston head geometry to provide recess or piston bowl. Tumble flow help for faster flame propagation during power stroke due to high initial velocity of fluid.

III. METHODOLOGY

For CFD analysis, following methodology used to know velocity, velocity contour pattern & turbulence kinetic energy with respect to piston geometry



IV. ANALYSIS

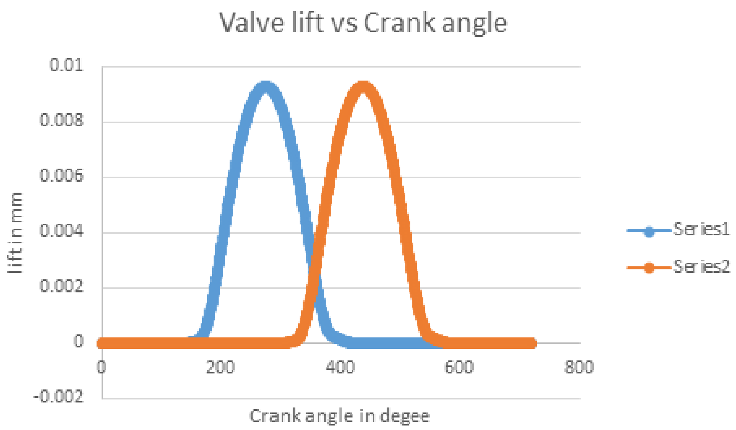
Following Different geometry and model used of analysis purpose.

Boundary conditions

The CFD is performed using moving mesh in fluent the velocity of moving mesh is calculated upon using parameter given below using Ansys automated approach.

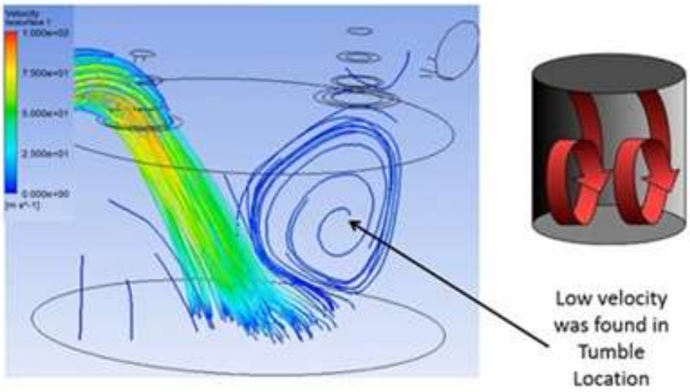
Speed	:	1800 RPM
Piston Diameter	:	48mm
Connecting Rod	:	200mm
Crank Radius	:	20 mm
Volume	:	100 CC
Stroke length	:	54 mm
Compression ratio	:	5

The viscous model is considered as standard k-epsilon.
The working fluid is considered as ideal gas.
A gradual valve lift is considered with respect to crank angle corresponding graph is shown below

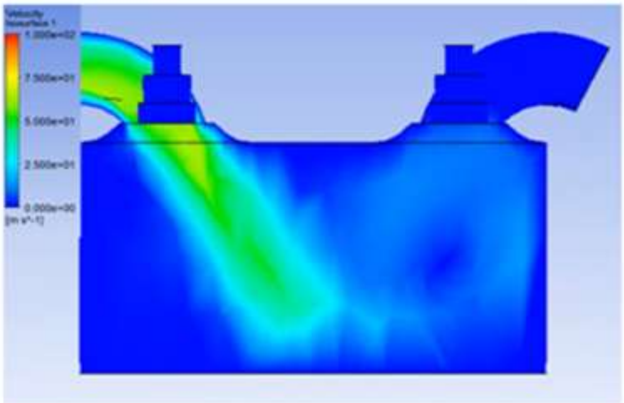


Design-1 Flat head piston

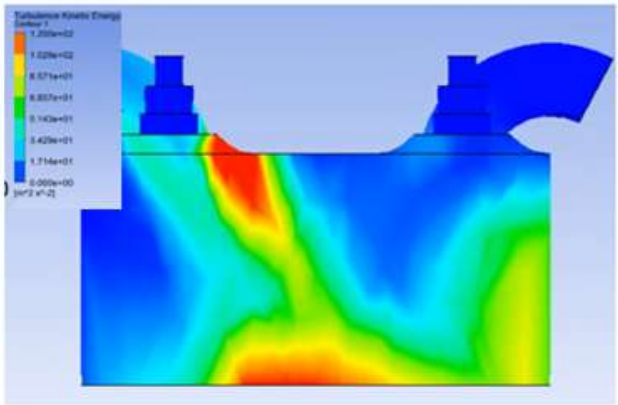
Analysis velocity stramline



Analysis velocity

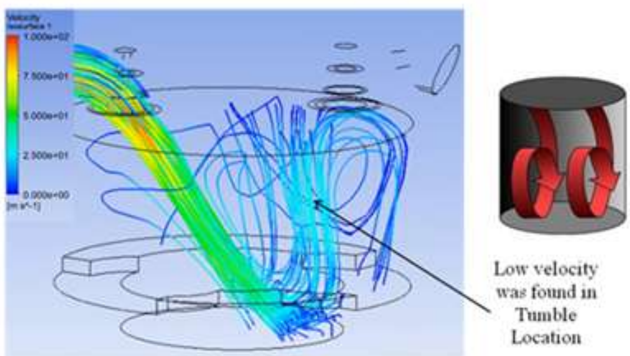


Analysis Turbulence kinetic energy

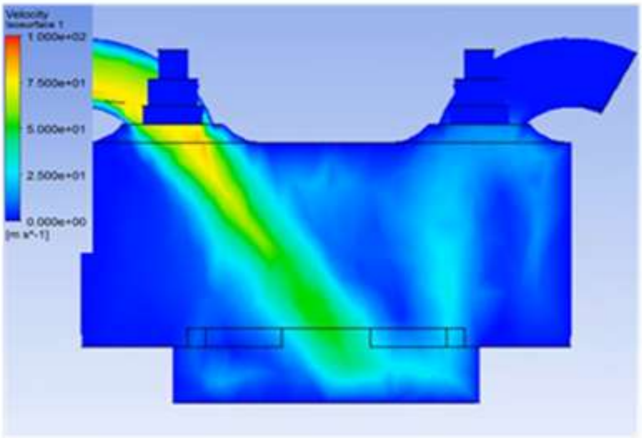


Design-2 Cross Flow piston

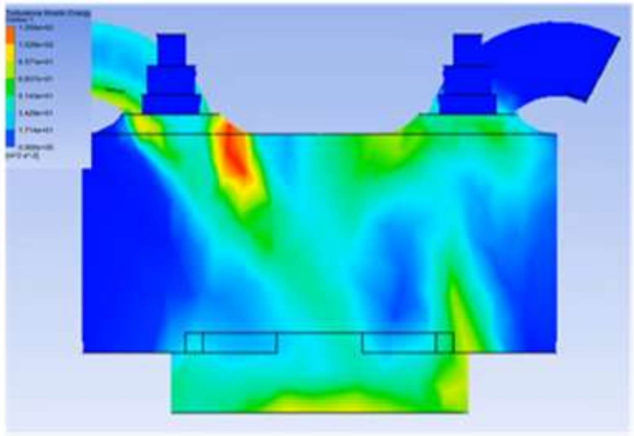
Analysis velocity stramline



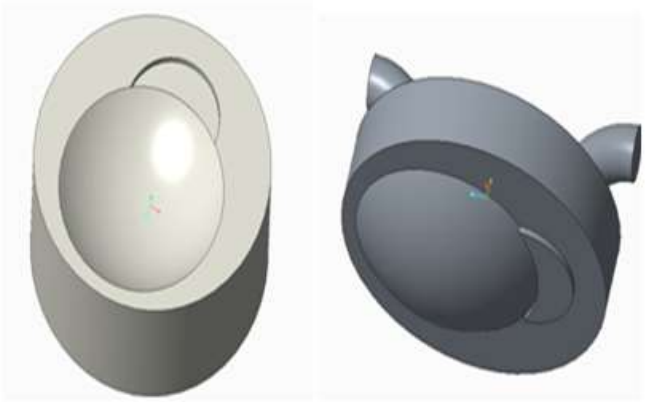
Analysis velocity



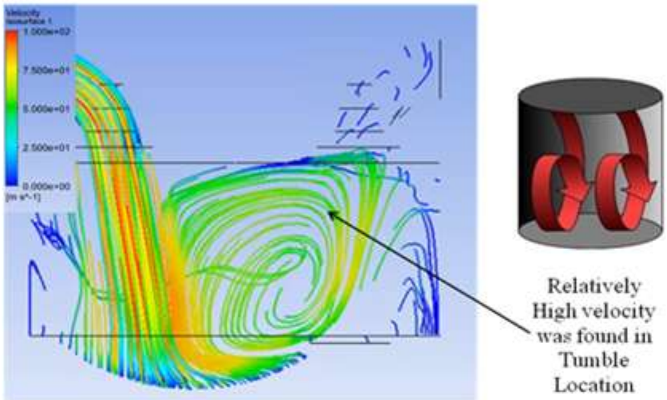
Analysis Turbulence kinetic energy



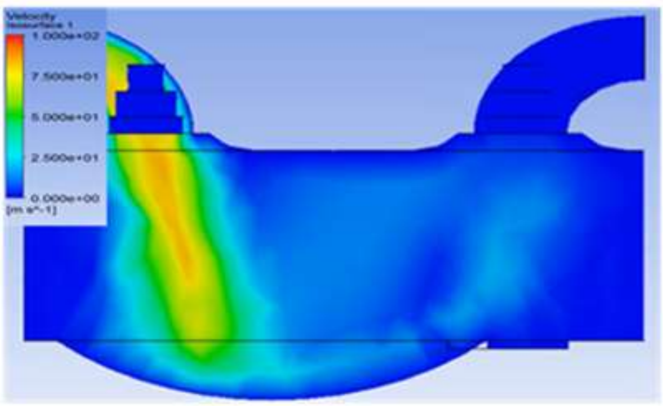
Design-3 Dome piston (offset)



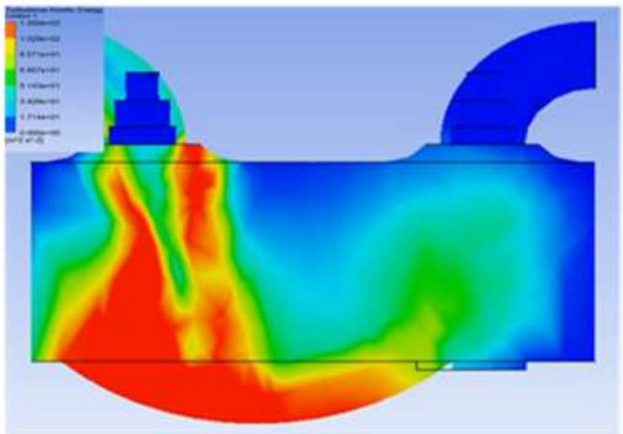
Analysis velocity stramline



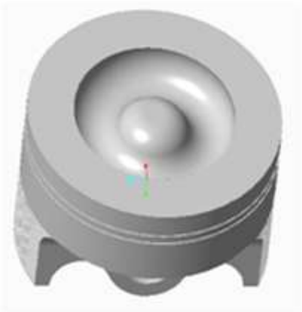
Analysis velocity

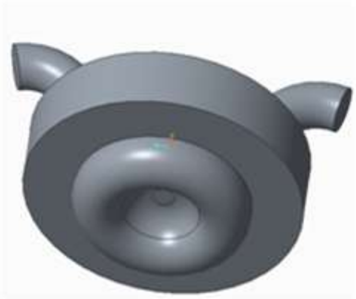


Analysis Turbulence kinetic energy

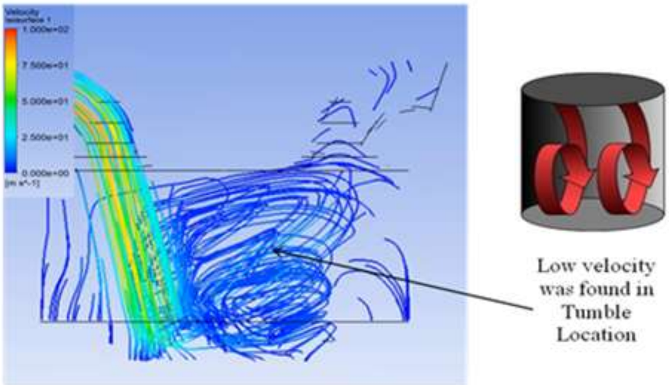


Design-4 Circular cavity piston

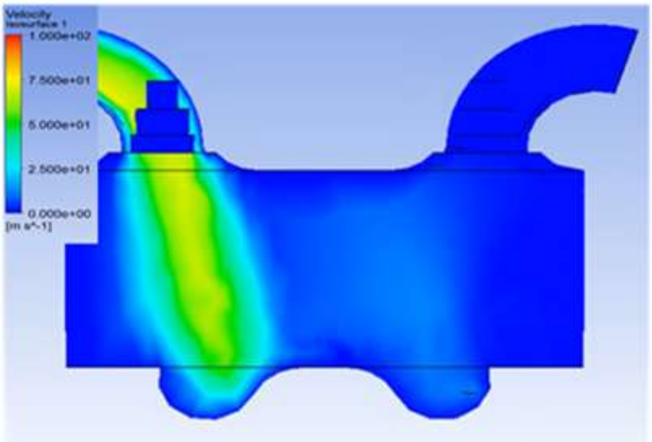




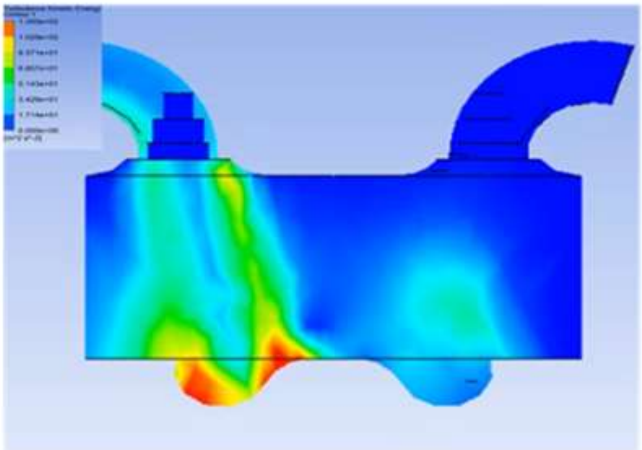
Analysis velocity stramline



Analysis velocity



Analysis Turbulence kinetic energy



V. RESULT & CONCLUSION

	Piston shape			
	Design-1	Design -2	Design -3	Design -4
Velocity (M/s)	65	71	95	76
Turbulence kinetic energy	111	116	124	120

Design 1

In this case flat head piston is considered to know velocity and turbulence kinetic energy for spark ignition engine. Flat head piston geometry is taken from benchmarking of single cylinder spark ignition engine. CFD analysis performed and we got above values mention of charge flow velocity, velocity streamline for tumble generation and turbulence kinetic energy.

Design 2

In this case complete system including valve position, boundary conditions are kept same as Design 1 except piston head geometry modification which is based on cross flow valve position used in ford engines. CFD analysis performed and we got above values mention of charge flow velocity, velocity streamline for tumble generation and turbulence kinetic energy.

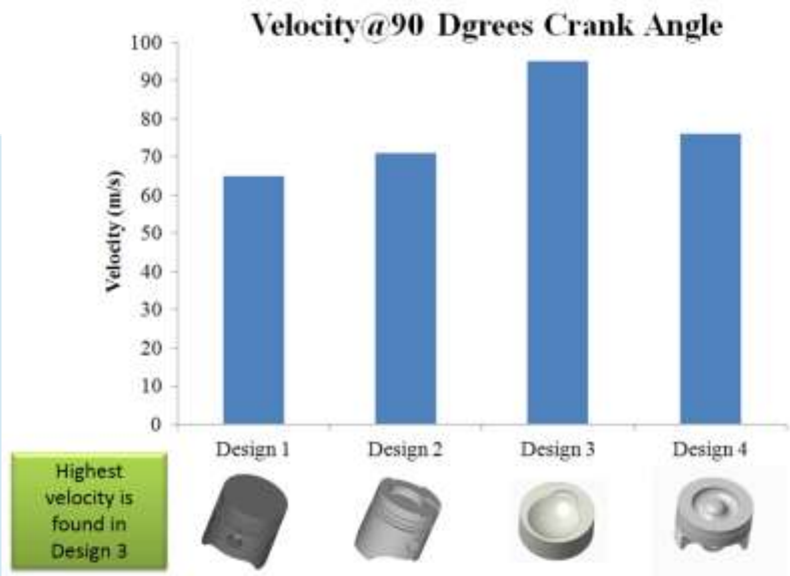
Design 3

In this case piston crown is further modified and offset bowl created on piston top surface CFD analysis performed and we got above values mention of charge flow velocity, velocity streamline for tumble generation and turbulence kinetic energy.

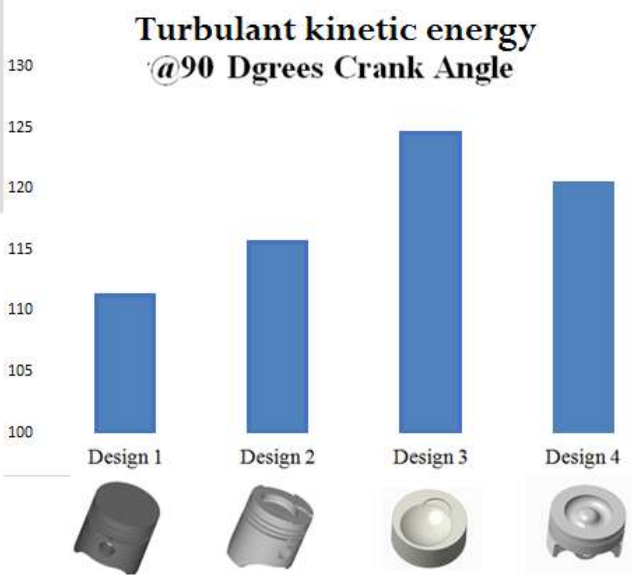
Design 4

In this case Circular cavity type shape head piston is CFD analysis performed and we got above values mention of charge flow velocity, velocity streamline for tumble generation and turbulence kinetic energy.

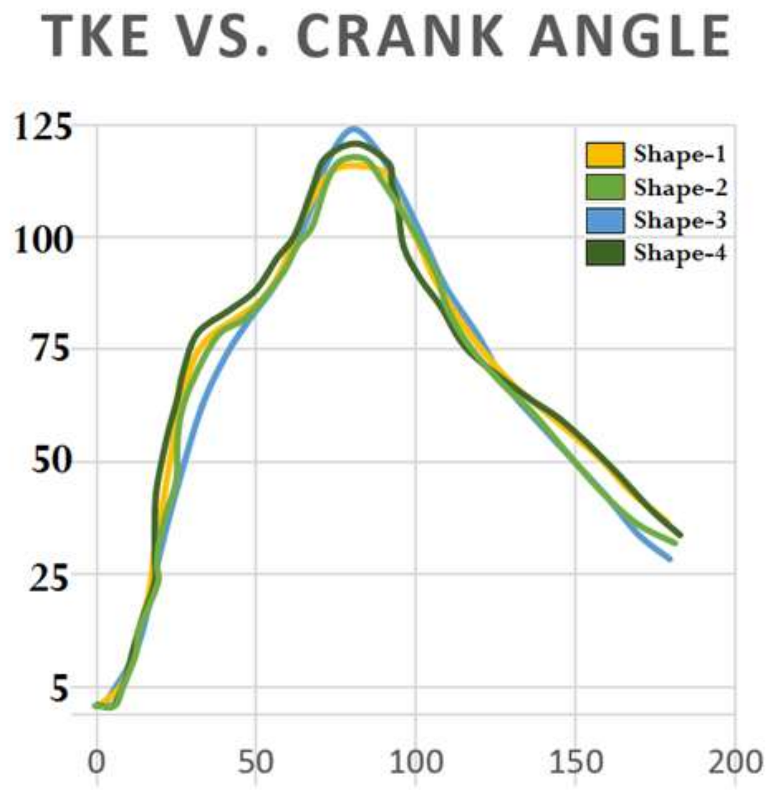
Bar Graph plotted between velocity values of different piston shapes to get effective comparison –



Bar Graph plotted between TKE values of different piston shapes to got effective comparison –



Graph plotted between Turbulence kinetic energy values of piston shapes with respect to different crank angle to got effective comparison –



Complete combustion of fuel inside the cylinder has a influence on the emission and performance of the engine. In-cylinder flow plays important role for the fast and complete combustion of the fuel. In-cylinder flow measure in term of flow velocity and turbulence kinetic energy and its different types is swirl. Squish and tumble.

For a single cylinder spark ignition engine tumble flow is necessary for proper burning of fuel and from above analysis and comparison piston design 3 which had offset bowl head piston having a better flow of charge 90-degree crank angle as the curved shape doesn't allow much loss of kinetic energy of charge.

The design 3 piston shape plays a huge role in cylinder flow as we can see above graph for velocity. We had taken the velocity in 90-degree crank angle as at this instance the velocity of charge is maximum because of inductance in inflow of charge

REFERENCES

[1] Marilia Gabriela J. Vaz, Felipe Grossi L. Amorim, Jean Helder M. Ribeiro
Numerical Analysis of the Piston Crown Geometry Influence on the Tumble and Squish in a Single Cylinder Engine
©Copyright 2014SAE International ISSN 0148-7191

[2] Bandi.Ramanjulu, Adissu Fulli, D.Jegan Raj, Abera Endesha Bekele
Performance Analysis of IC Engine Based on Swirl Induction by Using CFD
International Journal of Advanced Research in Science, Engineering and Technology
ISSN: 2350-0328

[3] Congbo Yin, Zhendong Zhang, Yuedong Sun, Tao Sun & Renjie Zhang
Effect of the piston top contour on the tumble flow and combustion features of a GDI engine with a CMCV: a CFD study
Engineering Applications of Computational Fluid Mechanics ISSN: 1994-2060

[4] R. Thundil Karuppa Raj and R. Manimaran
Effect of Swirl in a Constant Speed DI Diesel Engine using Computational Fluid Dynamics *CFD Letters*

[5] John L. Lumley
FLOW IN THE CYLINDER
Engines – an introduction. Cambridge University Press, 1999

[6] Heywood, J.B
Internal combustion engine fundamentals.USA: McGraw-hill Ltd.

[7] Richard Stone
Introduction to internal combustion engines