# CFD Analysis of Single Cylinder Four Stroke Gas Fueled Engine for Prediction of Air Flow Rate during Suction Stroke

[1]Madhusudan Barot, [2]Prof.Abhishek Shah,[3]Prof. Mit Patel

[1]PG student, [2,3]Assistant Professor

[1] Department of Mechanical Engineering,

[1,2,3] Silver Oak College of Engineering and Technology, Ahmedabad

---

***Abstract— CFD analysis in internal combustion engine is aims to improve process parameter which improves performance of engine and in turns providing fuel economy. In first part of this thesis, the CFD analysis is carried out for single cylinder four stroke petrol engines to analyze the performance parameters in suction stroke and power stroke. The CAD geometric model is prepared in solid works software and imported to ANSYS software to perform CFD analysis in IC engine module. To predict the behavior of the engine during its working two type of analysis can be performed namely port flow simulation and combustion simulation. Port flow analysis is carried out to find the velocity magnitude of air during suction stroke. Maximum velocity of air reaches to value of 7.01 m/sec which ensure proper swirling better mixing with fuel to be injected. A result of port flow simulation indicates that valve lift affects velocity flow field inside the cylinder.***

*IndexTerms*— **IC Engine CFD, Swirl velocity, Port Flow Analysis**

---

## I. INTRODUCTION

CFD simulations can be a powerful engineering tool that can help you understand and the thermo-chemical processes that take place in the internal combustion engine without being forced to use the techniques of measurement, if any, complicated and costly. Some things (such as the flow field or temperature in the cylinder) can be difficult to measure in three-dimensional space and one-dimensional models do not take into account all geometric topologies, making CFD simulations an important complement. It can also be a valuable tool to run a lot of tests and research on an engine without having to run a test that can reduce development costs. CFD simulations can also serve as a quick and economical way to conduct studies of future engine designs and concepts before producing a prototype, further reducing development costs. CFD research, unfortunately, has not yet reached a state that fully describes all the processes that occur in the internal combustion engine. This is mainly due to the fact that it is a very complex mechanical device that includes many simultaneous interactive processes, thermal and chemical fluids. Because of the alternate nature of internal combustion engines, there is extreme deformation of the solution's dominance due to the movement valve piston, and the shape of the piston and cylinder head, which are crucial for engine design features, are often very complex. In addition to geometric complexity, there are also processes that simultaneously interact with thermo-fluid as; Unstable turbulent flow, heat transfer and injection mass, atomization, dispersion and vaporization of liquid fuel, ignition, combustion and consequent formation of harmful pollutants, to name a few. The sum of all this makes the entire internal combustion engine modeling one of the most difficult tasks in the field of CFD research.

The performance of a combustion engine depends on complex interactions between mechanical, fluid, chemical and electronic systems. However, the main challenge in design is the fluid-dynamic complex of turbulent flow flows with moving parts through the inlet / exhaust valves, piston and cylinder. The suction, injection, liquid vaporization, turbulent mixing, transport, chemical contaminants and overlap formation time scales are to be considered simultaneously. Computational Fluid Dynamics (CFD) has emerged as a useful tool to understand the dynamic engine design ICs of the fluid engine. This is because, unlike computational analytical, experimental, or less dimensional methods, multidimensional CFD modeling allows designers to simulate and visualize the dynamics of complex physical solving physics for mass transportation laws, amount of motion and energy in 3D geometric patterns For critical phenomena such as turbulence and fuel chemistry. The information provided by the CFD analysis helps guide the geometrical design of parts such as ports, valves and pistons; Because engine parameters such as timing and injection valve.

## II. LITERATURE REVIEWS

The analysis of internal combustion engine processes and its use for engine research and development has a long and steady history. Though new dimensions are being added to this field in terms of emission modeling, the use of engine simulation for performance and efficiency modeling, these activities were aimed at developing Realistic approximations to real engine processes such as intake, compression, expansion, exhaust etc. and More accurate methods for calculating the thermodynamic properties of the working fluids used in the engines.

U Kongre and V. Sunnapwar [2] has have computer models of fluid dynamics and combustion engine experimental validation powered by direct ignition diesel. In their study, the development and use of sub-models for indirect injection combustion analysis has been considered. The experiments were performed in a single cylinder and a DI engine with full load conditions at constant speed of 1500 rpm. Combustion parameters, such as the speed of pressure increase, the heat release rate and cylinder pressure obtained from the experiment. Numerical modeling is solved considering the effect of turbulence. The turbulence model for the Model Theorem Group (RNG) model k-ε is used. Comparison of the simulation and experiment results was done in terms of pressure increase rate, cylinder release and heat release rate. It was concluded that computational fluid dynamics is a reliable tool for analyzing the combustion process of the IC engine .

K Pandey and B Roy [3] made the CFD of the suction valve for the SI Engine fuel injection port. The standard air-to-air ignition spark is about 60% fully loaded, but the actual full load thermal brake efficiency is about 32.60%, which is due to various losses. The main loss is the loss of combustion time is about 4.0% and occurs due to the final combustion load. It can be reduced by creating a greater turbulence magnitude that increases the turbulence level and the turbulence intensity of turns. In the combustion of lean combustion, high turbulence intensity is an important parameter in the propagation of the flame. Generally, drop flows and flow turbulence are configured in the vertical plane and the horizontal plane of the cylinder. In their study it was concluded that the surfaces near the suction valve at high speed with respect to the remote surfaces of the suction valves.

A Patil1 and L Navale [6] had made experimental verification with computational fluid dynamics analysis of four stroke single cylinder diesel engine for exhaust system. Research deals with the computational fluid dynamics and exhaust system designed, a compromise comparison made between brake thermal efficiency and back pressure. For computational fluid dynamics analysis, three exhaust diffuser systems with different angels were simulated using the appropriate fluid properties and boundary conditions with suitable assumptions. The model with limited backpressure was fabricated and experiments were carried out on four stroke single cylinder diesel engine test rig with rope brake dynamometer. The results show that increasing inlet cone angle leads to rise pressure of flow which decrease recirculation zones. CFD analysis for back pressure on engine shows good agreement with experimental work.

S Zanforlina and A Boretti [7] conducted a numerical analysis of the direct injection of methane into 250 cc of a single-cylinder petrol engine. A study was conducted to determine the potential of low pressure fuel injection systems below 20 bar for gas fuel combustion engines. The CFD analysis performed to assess the influence of valve valves and valves on jet characteristics, methane and air mixture, load distribution at ignition, injection pressure and valve and valve profiles in jet characteristics. The simulation was made for the desired engine to consider realistic contour condition as much as possible. You can also find which operating system and geometric details can withstand the complete blend of injection homogeneity. The study shows gaseous fuels difficulties to have satisfactory rolling loads that would be required to observe satisfactory performance on partial loads.

B Biradar1 and S Kumarappa [8] had investigated the analysis of volume fraction of air using clod flow simulation for four stroke single cylinder diesel engine. Diesel engine performance can increased by improving design of combustion chamber, inlet manifold, exhaust manifold and piston. Effect of piston configuration such as flat shaped piston, bowl and toroidal shaped piston was studied in cylinder flow. Batter mixing of air and fuel obtained by increasing swirl intensity. During compression with suitable piston design swirl velocity increased. Results show shat temperature and pressure generation is less in bowl and toroidal piston as compare to flat piston but generation of swirl is batter in toroidal and bowl shaped piston as compared to flat piston.

A Kolhe and R Shelke [9] has done combustion of CFD modeling engine powered by direct injection of biodiesel CI. Research shows the development of sub models for the analysis of the direct injection combustion engine for compression ignition Pongamia biodiesel-diesel fuel mixture pinnata powered. In the computational fluiddynamic modeling study, a complex combustion phenomenon was used in the diesel engine. With fully loaded condition at a constant speed of 1500 rpm, the experiment was performed in a single cylinder diesel engine. From experimental combustion parameters such as heat release rate and heat release rate has been obtained. When considering the effect of the numerical model turbulence it was solved by the CFD technique and the turbulence model k-ε Theory model renormalization (RNG) was used. Simulated results were obtained including the rate of increase in pressure, heat release rate and cylinder pressure. A good agreement between experimental data and modeling ensures the precision of numerical predictions. The rate of pressure increase, the rate of release of heat and the maximum cylinder pressure values shows a good agreement between measured data and experimental modeling.

## III. Modeling and Meshing

This deals with three dimensional modeling of engine from the measured dimension of splendor bike engine. Four stroke single cylinder spark ignition Splendor bike engine having compression ratio of 9 and displacement of 97.20 cc is considered in present work. First step is to model the engine by using CAD software from measured data of single cylinder four stroke petrol engine. Three dimensional modeling of the engine is done by using solidworks software. After that, meshing of the generated model is achieved in ANSYS mesh module. Finally, mesh generated model is imported to FLUENT software for simulation. The specifications of the engine and the modeled operating conditions are listed in Table 1.

**Table 1- Specification of Single Cylinder Four Stroke Petrol Engine**

| Engine Parameters | Value of Parameter |
|---|---|
| Type | 4-stroke |
| Cylinder arrangement | Single cylinder and $80^{o}$ inclined from vertical |
| Bore | 50.00 mm |
| Stroke | 49.50 mm |
| Displacement | 97.20 cc |
| Compression ratio | 9 : 1 |
| IVO (Inlet Valve Opening) | $4^{o}$ before TDC |
| IVC (Inlet Valve closing) | $24^{o}$ after TDC |
| EVO (Exhaust Valve Opening) | $26^{o}$ before BDC |
| EVC (Exhaust Valve closing) | $1.5^{o}$ after TDC |

**Computational Modeling: The** three dimensional model is done in solidworks software using the original geometry dimensions of the single cylinder four stroke petrol engine. The geometry formed includes the important details of the real engine. To simplify the meshing the geometry cleanup can be perform in ANSYS meshing module software. The simple geometry is meshed and specific zone names and types are assigned.

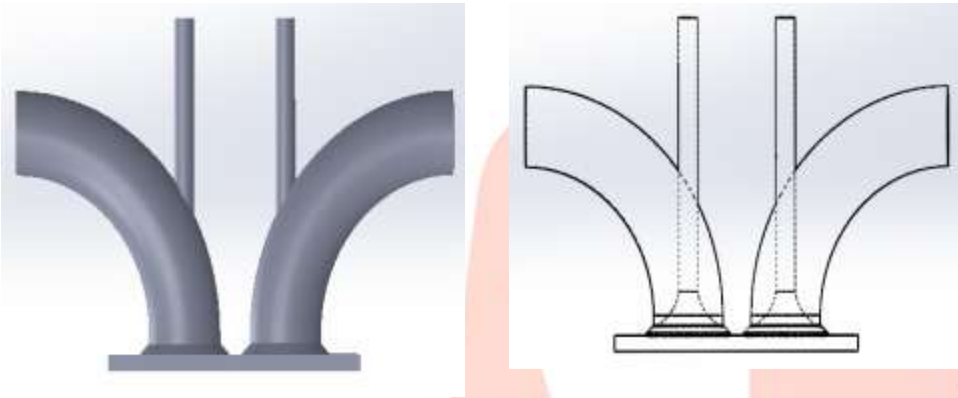

Fig. 1- CAD Model Generated in Solid Works

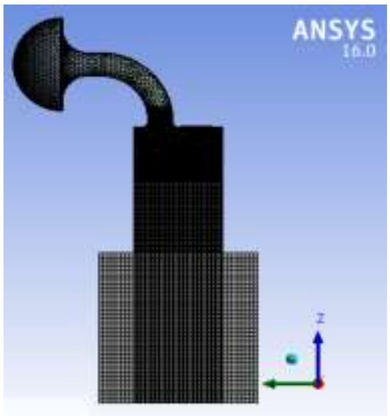


Fig. 2- Mesh model

**Mesh Generation:** Because of the rather complex geometry of the head and the piston sky engine, especially considering the valve interaction and the stock valve, the generation of mesh software needed to handle meshes of all types of cellular forms. It is also necessary that the software can handle different cell segments, each containing different cell types, within the same network. Since the Mummi approach used in this work there was also a request to make new meshes easily and quickly on the basis of a first initial mesh. ANSYS software can import most common CAD geometry file formats and become vertices, edges, faces, and volumes. Gambetto is able to automatically or manually create triangular rectangles / (2D) or hexadricos, tetraedric and wedge prism (3D). Note that each volume can contain only one type of cell form and, therefore, it is necessary to divide the geometry into different sub-volumes if all the geometric engine should be made up of different types of cells. This allows a very fast automatic creation of new mesh, which means that an initial mesh occurs manually.

## IV. PORT FLOW ANALYSIS

For flow analysis port, the geometry of ports, valves and cylinders is "frozen" at critical points during the engine operating cycle and using fluid computational fluid flow can be analyzed through the openings. You can determine the volume of airflow in the engine, the turbulence and drop the cylinder and turbulence levels. Phenomena such as separation, jet, choke valve, impact wall and binding and secondary movements can be visualized and analyzed through computational fluid dynamics. The results can be obtained as snapshots of fluid dynamics along the motor that can be used to modify the orifice geometry to produce the desired airflow cycle behavior. Validating the simulation can be done using the actual geometry mounted counterflow flow measurement, velocity and turbulence levels using techniques such as Laser Doppler. The results are unable to capture dynamic events such as compression and expansion of the air due to piston motion and the production of turbulence turbulence and the fall in simulation flow light. In general, performing flow analysis at a single point is relatively simple thanks to static geometry, which fits well with workflow and software computational fluid dynamics. Initiation can be done with the geometry of the door, valve and cylinder in a specific location, creating a mesh, specify the mass flow or drop of pressure for the compressible flow and turbulence pattern and calculate the results. Used turbulence models are based on the calculation of the turbulence effect. Turbulent flow interactions are critical walls, the mesh refinement in the region near the necessary parietal layers using inflation or limit. Experimental data provides validation criteria to develop best practices for model configuration and precision. However, the number of critical positions and thus the number of cases increases, the complexity of the problem increases. Setting a large number of instances with identical static and flow jumper settings takes a long time, with the possibility of error.

## V. RESULT AND DISCUSSION

**Velocity Profile on Cut Plane** : Simulation of single cylinder four stroke spark ignition engine for port flow analysis has been carried out in ANSYS IC Engine module for given cylinder head and intake port over varying valve lift of 2 mm, 4 mm, 6 mm and 8 mm. Computational result for the specified SI engine at various locations along the length of the engine cylinder is shown in counter plot for cut plane and swirl plane which is 15 mm below TDC.

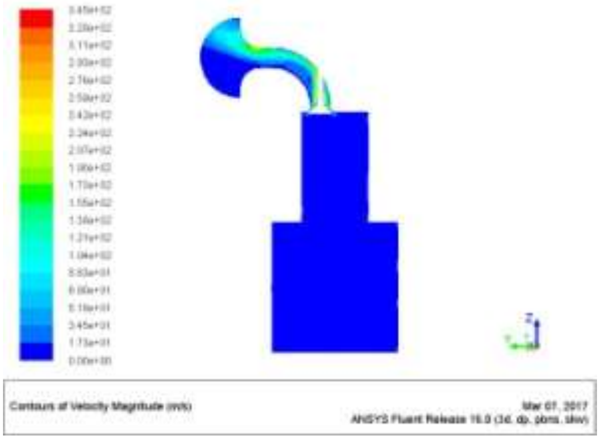


Fig.3 - Velocity Magnitude on Cut Plane Valve Lift 2 mm

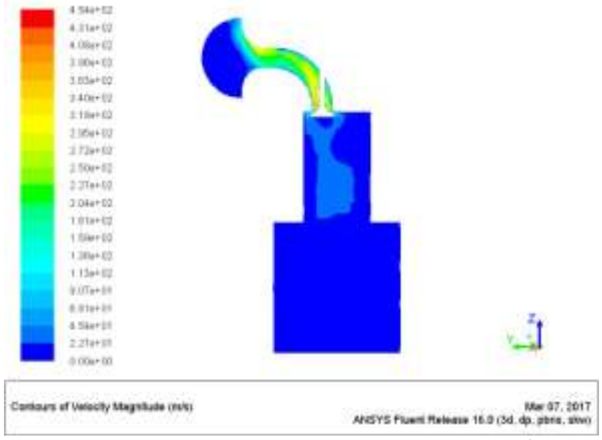


Fig.4- Velocity Magnitude on Cut Plane Valve Lift 4 mm

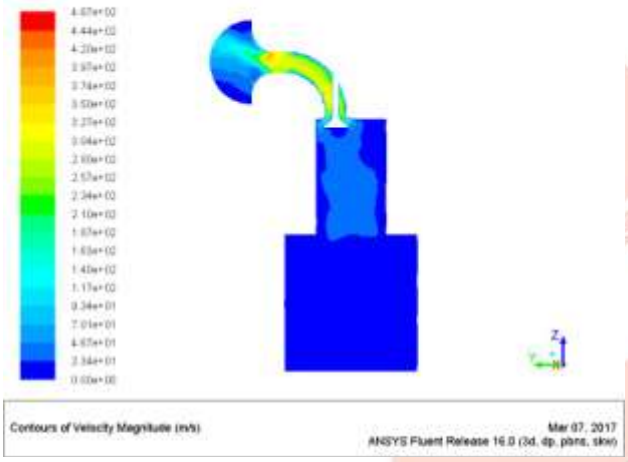


Fig.5- Velocity Magnitude on Cut Plane Valve Lift 6 mm

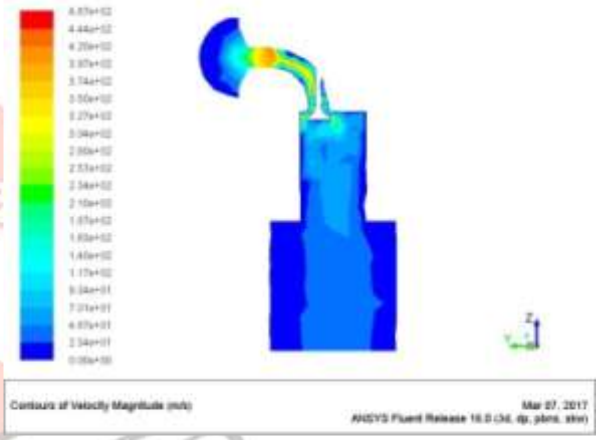


Fig.6- Velocity Magnitude on Cut Plane Valve Lift 8 mm

When inlet valve is closed no air is enters into cylinder so velocity of air in the cylinder is zero and as soon as inlet valve opens air is started drawn into cylinder. Figure shows the cylinder position when valve lift is 2 mm, 4 mm, 6 mm and 8 mm. From the above simulated result the maximum velocity of air is obtained as 7.01 m/sec when the inlet valve is fully opened during the suction stroke.

**Velocity magnitude on swirl plane:** For present simulation Velocity magnitude on swirl plane which is located 15 mm below TDC is obtained when valve lift is 2 mm, 4 mm, 6 mm and 8 mm.

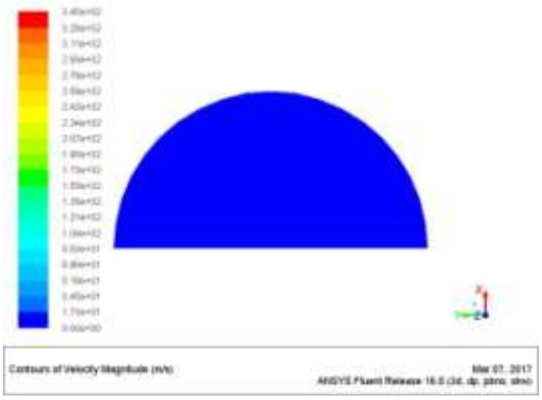

Fig.7- Velocity Magnitude on swirl plane (15 mm below TDC Valve Lift 2 mm)

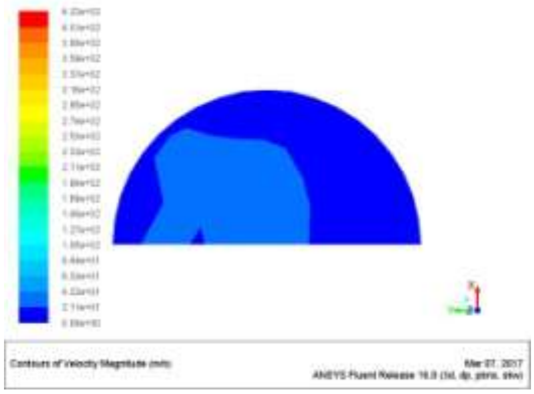

Fig.8- Velocity Magnitude on swirl plane (15 mm below TDC Valve Lift 4 mm)

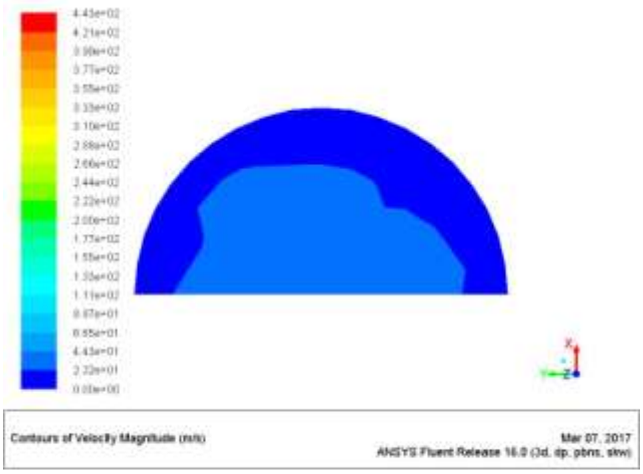

Fig.9- Velocity Magnitude on swirl plane (15 mm below TDC Valve Lift 6 mm)

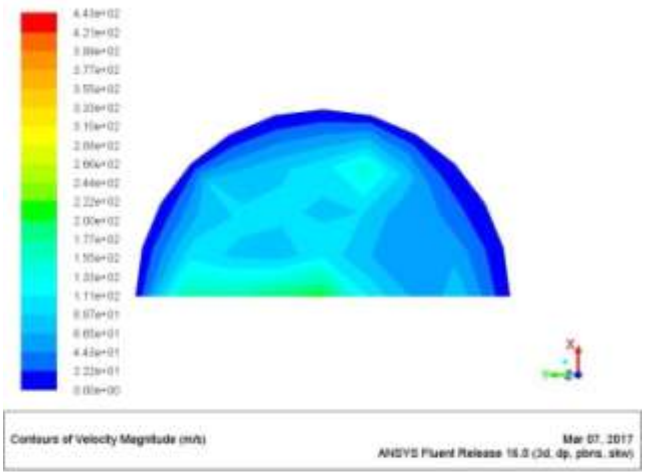

Fig.10- Velocity Magnitude on swirl plane (15 mm below TDC Valve Lift 8 mm)

From the results of the computation analysis carried out with poppet intake valve, for the specified SI engine it is seen that the surface at 15 mm from TDC for 2 mm, 4 mm, 6 mm and 8 mm valve lift. Results shows that higher velocity is obtained at surface which is closer to the valve as compared to the surface at away from the valve from TDC which is at higher distance from the intake valve.

**Mass Flow Rate:** Amount of air entered into the cylinder during suction stroke simulated and obtained as shown in Table 5.1 for varying valve lift 2 mm, 4 mm, and 6 mm.

Table 2- Mass Flow Rate of air

| Sr No | Valve Lift (mm) | Mass Flow Rate (Kg/sec) |
|---|---|---|
| 1 | 0 | 0 |
| 2 | 2 | 0.012801 |
| 3 | 4 | 0.029903 |
| 4 | 6 | 0.039427 |

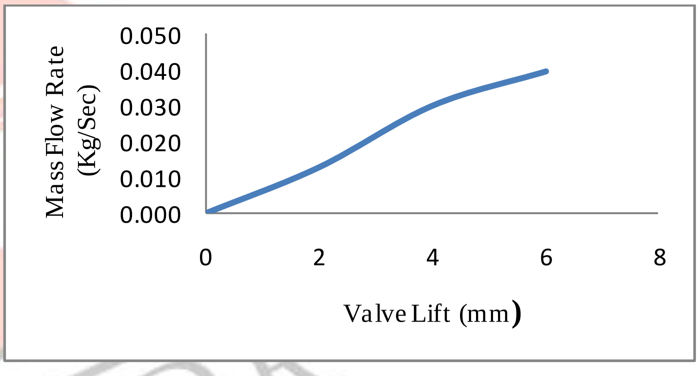

Fig.11 - Mass Flow Rate (kg/sec) vs Valve Lift (mm)

The graph shows that with the increase in valve lift from 2 mm, 4 mm and 6 mm the mass flow rate of air is increases to 0.012801 kg/sec, 0.029903 kg/sec and 0.039427 kg/sec. Hence mass flow rate is increases with increase in valve lift.
From this study the following it can be concluded that the surface which is closer to the poppet intake valve shows higher velocity at various locations compared to the surfaces which are at higher distance from the intake valve i.e. the intensity of swirl decreases along the stroke length of the engine cylinder.

## VI. CONCLUSION

After performing port flow simulation and combustion simulation of single cylinder four stroke petrol engine using CFD following remarks can drawn as a conclusion of the present study.

- During suction stroke maximum swirl velocity of air is obtained as 7.01 m/sec when the inlet valve is fully opened.
- The surface closest to the intake valve pin has a higher speed in the various locations than surfaces that are further away from the suction valve.
- Mass flow rate of air is increases with increase in valve lift.

## 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 Sunnapwar V"CFD Modeling and Experimental Validation of Combustion in Direct Ignition 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] Hiregoudar Yerrennagoudaru, "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] Basanagouda Biradar, "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.

[11] Patil V and Agrawal A, "In Cylinder Combustion Analysis of DI Diesel Using Experimental and CFD Approach", International Journal of Engineering Trends and Technology (IJETT) – Volume 14 Number 5 – Aug 2014.

[12] Patil UD, "Cylinder Head Intake Port Design & In-Cylinder Air-flow Patterns, Streamlines formations, Swirl Generation Analysis to Evaluate Performance & Emissions" International Journal of Engineering Research & Technology (IJERT) - Vol. 2 Issue 9, September – 2013.

[13] Sabale SK and Sanap SB, "Design and Analysis of Intake Port of Diesel Engine for Target Value of Swirl" American Journal of Mechanical Engineering, 2013, Vol. 1, No. 5, 138-142,

[14] Himanth Kumar and Jayashankar N, "Port Flow Simulation Of An Ic Engine" International Journal Of Innovations In Engineering Research And Technology Volume 2, Issue 9, Sep.-2015.

[15] Priyadarsini I, "Flow Analysis of Intake Manifold using Computational Fluid Dynamics" International Journal of Engineering and Advanced Research Technology, Volume-2, Issue-1, January 2016.

[16] Workshop manual of HERO HONDA splendor. Service Department, Hero Honda Motors LTD.

[17] Geok HH and Mohamad TI, "Experimental Investigation of Performance and Emissions of a Sequential Port Injection Compressed Natural Gas Converted Engine" SAE International 2009.

[18] Tripathi A and Panchal P, "Turbulent Flame Speed Prediction For S.I. Engine Using Methane As Fuel", International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622, Vol. 3, Issue 4, Jul-Aug 2013, pp.248-254