IJETST- Vol.||02||Issue||06||Pages 2557-2564||June||ISSN 2348-9480 2015

Open access Journal International Journal of Emerging Trends in Science and Technology

Simulation of In-Cylinder Partially Premixed Combustion Mode

Authors

Sharath S[1], Dr. G. V. Naveen Prakash[2]

[1]M-Tech student, Department of Mechanical Engineering, Vidyavardhaka College of Engineering, Mysuru 570002, Karnataka, India

Email: *sharath2705@gmail.com*

[2]Professor, Department of Mechanical Engineering, Vidyavardhaka College of Engineering, Mysuru 570002, Karnataka, India

Email: *gvnp@vvce.ac.in*

Abstract

An SI engine gets the charge as a mixture of air and fuel. The quality of the mixture plays a very important factor that decides the quality of combustion, efficiency of the engine, particulate and acoustic emission. Hence it is important to arrive at the right quality of fuel and method of induction of charge. This study focuses on comparison of premixed and partially premixed combustion modes. Various parameters are studied and compared using the Ansys Fluent software. Contours such as progress variable, total pressure, total temperature and turbulent flame speed are obtained at various crank angles for premixed and partially premixed modes. By this simulation it is concluded that partially premixed mode is a better viewpoint in engine design.

Keywords:*partially, premixed, combustion, in-cylinder, turbulent, flame*

1. INTRODUCTION

Engine researchers are continuously working on maximizing the power produced from fuel combustion. In-cylinder fluid dynamics is one of the variables that affect the internal combustion in an engine. [1] The processes of preparation and conveyance of fuel mixture in the engine is predominantly governed by the in-cylinder charge motion. [2] The production of high turbulence flame intensity is one of the most important factors for stabilizing the ignition process and fast propagation of flame, especially in the case of lean-burn combustion. Turbulence inside the cylinder is high during the intake and then decreases as the flow rate slows near the bottom dead center (BDC).

This turbulence is enhanced by the expansion of engine cylinder during the combustion process.[3] At the spark event, the mixture is assumed to be homogeneous. The simulation starts at intake valve close (IVC) and ends at exhaust valve open (EVO). So, there are no valves involved. A pure layering approach can be used on a 2D axisymmetric geometry. The In-cylinder fluid motion is one of the major factors that control the fuel-air mixing and combustion process in SI engines. The in-cylinder motion may be characterized as a combination of swirl, sideways-tumble and normal-tumble. [4]

Anurag Mani Tripathi et al [5] have presented the combustion modelling of a single cylinder 4 stroke spark ignition engine with a compression ratio 9.2 and displacement of 124.7cc using premixed combustion model of FLUENT software package with methane gas as fuel. They have plotted the results of turbulent flame speed versus engine speed (Fig. 1) and equivalence ratio (Fig. 2) which indicate that the turbulent flame speed increases with the increase in engine sped and equivalence ratio respectively. Contour of Turbulent flame speed (m/sec) at 410.50˚ of crank rotation is shown in Fig. 3.

Fig. 1: Turbulent flame speed v/s Engine speed [5]

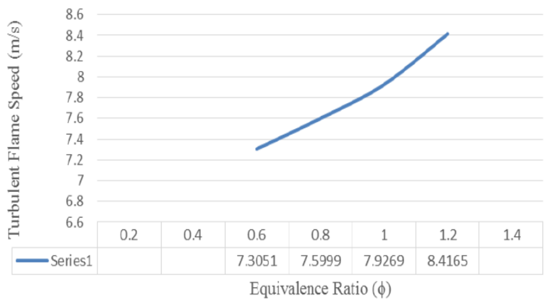


Fig.2: Turbulent flame speed v/s Equivalence ratio [5]

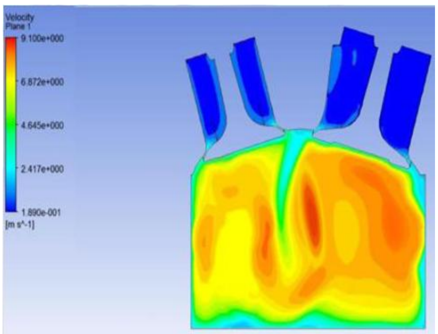


Fig. 3: Contour of Turbulent flame speed (m/sec) at 410.50° of crank Rotation [5]

Rashid Ali et al.,[6] have worked a mathematical model of flame velocity for iso-octane air mixture which has been developed in terms of fraction of the charge burnt, flame radius, equivalence ratio, clearance height and pressure ratio. The effects of engine speed, equivalence ratio, unburnt mixture temperature, compression ratio and ignition timing have been studied and found that the flame velocity increases with compression ratio.

Wildman et al.,[7] opine that appropriate charge motion control could produce late robust combustion. It is desirable to have a fast initial (0-10%) burn rate, but a slower later (10-90%) burn rate. Thus the late robust combustion concept depends on the control of the engine turbulence. In particular, the breakup process of the large scale motion such as swirl and tumble into the small scale turbulence is responsible for flame propagation.

There are studies made to establish a relation between the flame parameters and the engine parameters. Turbulent flame speed and propagation speed are observed to have a correlation with the equivalence ratio and compression ratio of the engine and is directly proportional to them. There are studies which are made using CFD and numerical analysis considering the premixed mode of combustion. Whereas partially premixed is a condition which is closer to the actual phenomenon happening in the engine. Hence there is a need to analyse the combustion process as partially premixed mode and study the flame parameters and the associated phenomenon.

2. INVESTIGATION PROCESS

2.1 Objective: By analysing the partially premixed model of the mixture better prediction of peak temperature and pressure can be achieved. Hence in this study the focus is to:

a. Simulate the In-cylinder partially premixed combustion process using ANSYS Fluent.
b. Understand the flame behavior in terms of speed and patterns.

2.2 Methodology: In order to make a comparative study between premixed and partially premixed combustion mode, the method followed is:

a. Modelling of Cylinder (combustion chamber) using a pure layering approach on a 2D axisymmetric geometry.
b. Setting up In-cylinder partial premixed combustion.
c. Setting up the spark model.
d. Flame parameter analysis by using user defined function (UDF).

2.3 Assumption: At the spark event, the mixture is assumed to be homogeneous.

2.4 Validation: The validation for this study is done with the work by Anurag Mani Tripathi et al.,5in which they predicted the turbulent flame speed of a 4 stroke SI engine using Ansys Fluent for premixed combustion mode. 5The engine parameters were considered from the Honda Shine user manual 8 as shown in Table 1.

2.5 Model Creation: The Model is created using Ansys 14.5. Since the model is symmetrical about the axis, only half of the model is created. The distance between axis and cylinder wall is taken as 26.2 mm which is half of the bore dimension. The combustion chamber dimension is anyhow variable since the piston moves between TDC and BDC. When the piston is at TDC, it is assumed that the distance between the piston top and the cylinder head is 15 mm. Meshing and analysis is carried out using 2D layered axisymmetric approach. The results obtained by Anurag Mani Tripathi et al.,5are tabulated as shown in Table 2 which is used as a validating benchmark for our study.

Table 1: Engine parameters [5]

Input Parameter	Value of parameter
Type	1 Cyl, Air cooled, 4 stroke, OHC engine
Bore	52.4 mm
Stroke	57.8 mm
Displacement	124.7cc
Inlet valve opening (IVO)	5 ° BTDC
Inlet valve close (IVC)	30 ° ABDC
Exhaust valve close (EVC)	0 ° TDC
Exhaust valve open (EVO)	50 ° BBDC

Table 2: simulation results of the work by Anurag Mani Tripathi et al.[5]

Equivalence Ratio	**Engine Speed (rpm)**	**Turbulent Flame Speed (m/sec)**
0.6	1500	7.3051
0.8	2000	7.5999
1.0	2500	7.9269
1.2	3000	8.4165

Similar settings were made for this study and the results obtained were compared with those of existing work. The results were comparable and showed near coincidence. Fig. 4 shows the contours at the engine speed of 1500rpm. The maximum turbulent flame speed is observed as 7.04m/s against 7.3 m/s of the tabulated values. The percentage of error is 3.5% which is acceptable.

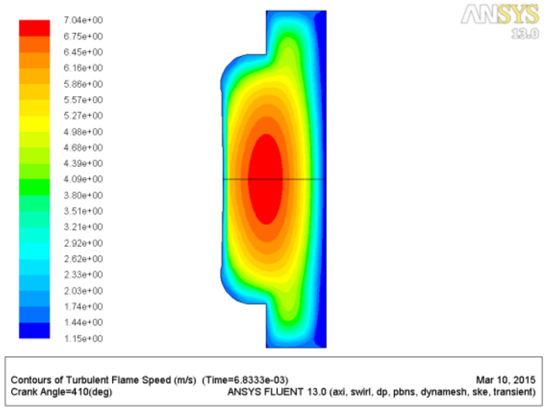


Fig.4: Turbulent flame speed at CA 410^{o}and 1500rpm.

The second trial was done for the engine speed of 3000rpm and the turbulent flame speed obtained was 8.2m/s against 8.42 m/s in the reference table as shown in Fig. 5. The percentage of error is 2.6% hence acceptable. The error may be due to reasons like valve consideration, geometry and meshing consideration etc. Assuming that the values in-between also follow the same trend, the method is thus validated. Same settings and values are preserved throughout the study.

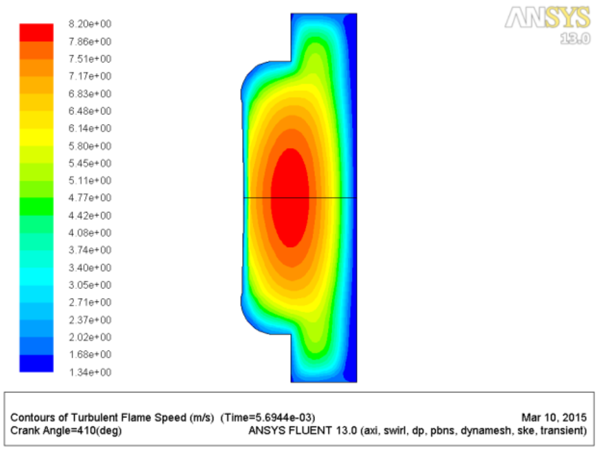


Fig.5: Turbulent flame speed at CA 410^{o} and 3000rpm.

2.6 Work carried out: The same engine parameters are preserved that were taken for the validation (Table 1). The model is prepared as shown in Fig.6.

During the simulation run, the half model is mirrored and analysis is carried out for the full combustion chamber. [9]

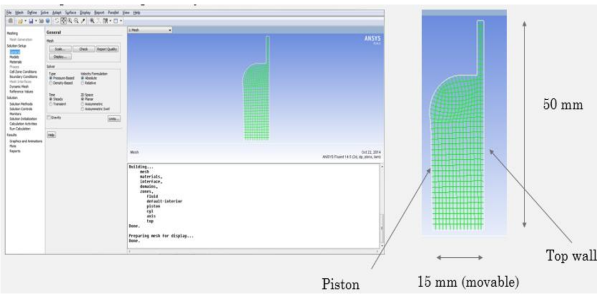


Fig.6: Meshed model of the half cylinder

The valves are not considered because this simulation is carried out between closures of inlet valve (IVC) and opening of outlet valve (EVO). In this case, the piston reaches the bottom dead centre at a crank angle of 540 °. The Inlet valve opens at 570 ° i.e. 30 ° ABDC (after BDC) and the piston reaches the TDC at 720°. The Outlet valve opens at 50° before BDC (BBDC). The piston reaches BDC at 900°. This covers 360° which is equal to 2 strokes between which the IVC and EVO takes place. The valve timing diagram for this condition is as shown in Fig.7. [10]

The simulation is carried out between the crank angles 570 and 850 ° i.e. between IVC and EVO.

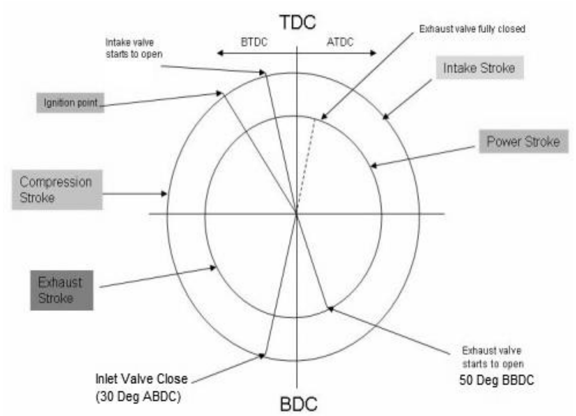


Fig.7: Valve timing diagram indicating the IVC and EVO and different strokes.

2.7 Preparation and general setting: ANSYS Fluent 2D version is started with double precision and enabled UDF compilation environment. The solver settings are set to pressure based, transient, axisymmetric swirl. Under the scales tab, the mesh created and viewing scales are set to mm. To define the mesh motion, the rigid body motion of the intake and exhaust valves, as well as that of the piston, are described using profiles or user-defined functions (UDFs) which prescribe the geometric translation of the moving wall zones as a function of time or engine crank angle.

Once the mesh motion and schemes are defined, FLUENT 13.0's mesh preview feature is invoked. This allows the user to cycle only the mesh through its full range of motion without the calculation of flow physics and provides a quick visualization and mesh integrity check prior to the submission of the more CPU intensive flow calculation.

Internal combustion Parameters: The dynamic mesh is enabled and layering method is selected. In-cylinder is enabled and the parameters are entered as follows.

Crank Shaft Speed: 3000 (rpm); Starting Crank Angle: 360^o; Crank Period: 720^o; Crank Angle Step Size: 0.25^o; Piston Stroke: 80 (mm); Connecting Rod Length: 140 (mm)

2.8 Dynamic mesh zone setting: The dynamic mesh zones for fluid, Piston, and top is defined under stationary, rigid body, deforming or User defined. The Piston and the fluid are classified as rigid bodies because they change their position with respect to time (in terms of crank angle). Whereas top is considered to be stationary object since it does not change its position with reference to time.

2.9 Mesh motion preview: Mesh motion preview is done in order to bring the piston to the IVC position. In the case considered, the piston is at TDC i.e. beginning of a cycle represents the crank angle 0^o. But for the purpose of calculation, we have to consider it as 360^o. When the piston reaches BDC it would have travelled 180^o. Thus a crank angle of 540^o is obtained. But the inlet valve closes 30o after the piston starts moving towards TDC i.e. at 570^o (30^o ABDC). This is the starting point of focus. In order to achieve this, a crank step size has to be set and number of steps to be increased has to be suitably selected in order to bring the crank angle from 360^o to 570^o. In this case a step size of 0.25o of crank angle is considered and number of steps are 840 which brings the crank angle from 360° to 570°.

2.10 Setting up the turbulent combustion models: The standard k-ε model is a model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). [10] Species model are defined based on the combustion mode i.e. partially premixed combustion mode. Since spark characteristics are not variables in the study, typical characteristics are considered. The spark size is fixed, shape of the spark is round, energy imparted by the spark is 0.1 Joules which indicates the smallest value and the duration of the spark is 0.001 seconds. Calculation activities are monitored by saving the results in JPEG image format and a notepad file that contains the data of pressure and volume values at various crank angles. The save points can be set at any number of steps. In this study it is taken as 90 steps. Until this point, the mesh is still not mirrored. Since the model is symmetrical about the axis, the mirror plane is chosen as axis. Once it is mirrored, the mesh looks as shown in Fig. 8.

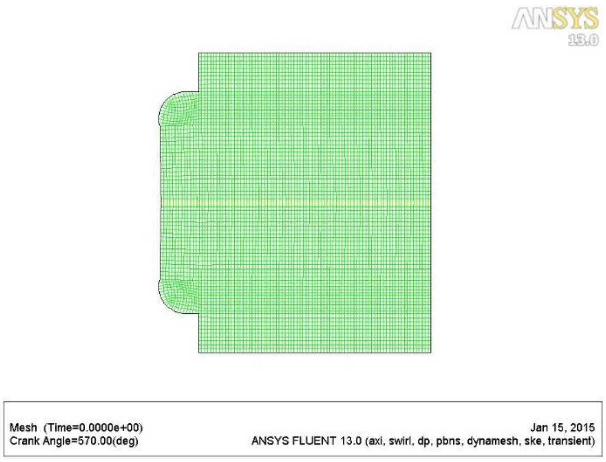


Fig. 8: Meshed and mirrored model at the CA 570°.

The calculation will be run until the Exhaust valve opens. The calculation starts at 570° and ends at 850°. The crank angle duration is 280°. In order to carry out the calculation for 280° with a step size of 0.25° total number of steps are 1120. The output obtained are then stored as images in a separate folder.

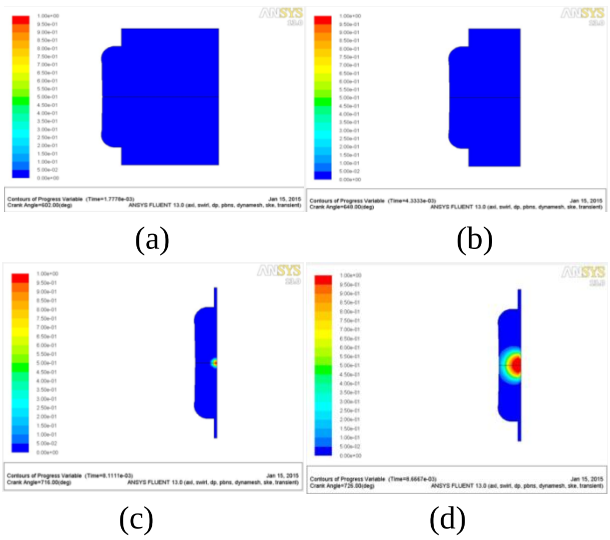
3. RESULTS & DISCUSSION

The simulation results which contain the pressure and volume values at the instantaneous crank angle are saved in the notepad. This is used to plot the graph of pressure and volume versus the crank angle. Fig 9 shows the pressure values of partially premixed mode at different crank angles between IVC and EVO.

Following spark ignition, there is a period during which pressure rise due to combustion is insignificant. As flame continues to grow and propagates, pressure rises steadily. Pressure is maximum after TDC but before the charge is fully burned, and then decreases as the cylinder volume continues to increase in its expansion stroke.

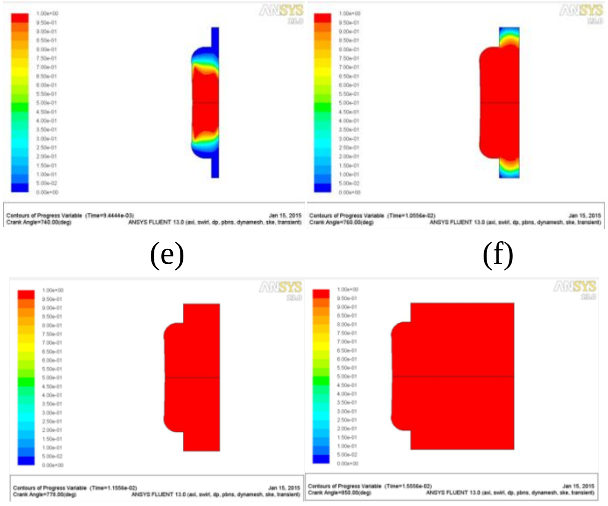
Fig.9: Pressure and Volume versus CA

The flame patterns for the partially premixed combustion mode at various crank angles are saved as images. Fig. 10 shows the flame patterns at few prominent points of crank angles.



(a) (b)

(c) (d)



(e) (f)

(g) (h)

Fig. 10: Flame patterns of the partially premixed combustion modes at various crank angles

Fig. 10 (a): CA 602° is the near beginning of the cycle. The pressure is at minimum value i.e. 2.97×10^4 Pa.

Fig. 10 (b): An intermediate stage between the start of cycle and the spark initiation is taken randomly as CA 648° where the pressure is at 1.72×10^5 Pa.

Fig. 10 (c): At CA 719° the spark is initiated and the corresponding pressure is 2.19×10^6 Pa.

Fig. 10 (d): Next stage is taken when the flame reaches the piston top i.e. at CA 726°. The pressure is noted as 2.22×10^6 Pa.

Fig. 10 (e): At CA 740° the pressure is noted as 2.46×10^6 Pa.

Fig. 10 (f): At CA 760° the flame front almost covers the combustion chamber and the pressure is 2.31×10^6 Pa.

Fig. 10 (g): At CA 778° the flame completely covers the combustion chamber and the pressure is 1.64×10^6 Pa.

Fig. 10 (h): At the final stage i.e., CA 850° the pressure is 4.54×10^5 Pa.

3.1.1 Study on different contours in partially premixed combustion mode:

The contours are captured at a crank angle 750° where the peak pressure is achieved. The different contours under study are progress variable, total pressure, turbulent kinetic energy and total temperature.

3.1.1.1 Contours of progress variable (flame pattern): At CA 750°, the spark is initiated and the flame front has almost covered the combustion chamber as shown in Fig. 11. The fuel air mixture in the red coloured region is fully burnt and the gradient which is observed indicates the partial combustion and the blue region indicates the unburnt fuel.

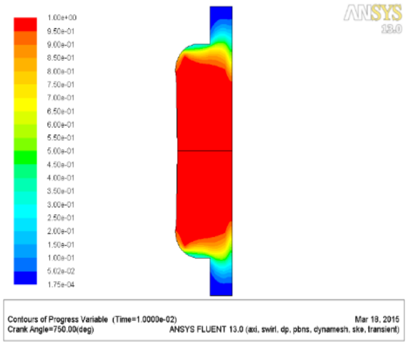


Fig. 11: Progress variable at CA 750°(at peak pressure)

3.1.1.2 Contours of total pressure: The total pressure contours of the partially premixed combustion is shown in Fig. 12.

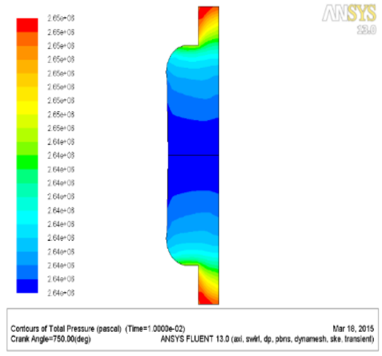


Fig. 12: Total pressure at CA 750° (at peak pressure)

3.1.1.3 Contours of turbulent kinetic energy: Fig. 13 shows the contour of turbulent kinetic energy. The region of the maximum kinetic energy is slightly more and away from the vortex. The centre region of the combustion chamber bowl on either side of the axis is apt for proper combustion. The profile of the kinetic energy indicates a better tumble motion.

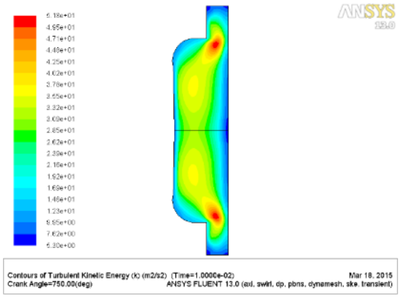


Fig.13: Turbulent kinetic energy at CA 750°(at peak pressure)

3.1.1.4 Contours of total temperature: Fig. 14 indicates the contour of temperature gradient at the CA of peak pressure. Temperature is minimum at the farther edges of the axis and maximum at the spark plug region. This is a healthy profile for the combustion. This will not give rise to knocking because the maximum temperature point is the spark plug itself.

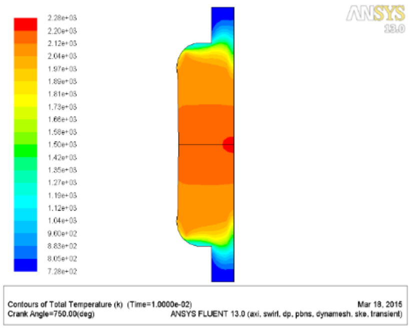


Fig. 14: Total temperature at CA 750° (at peak pressure)

4. CONCLUSION

The In-cylinder fluid motion is one of the major factors that control the fuel-air mixing and combustion process in spark ignition engines. In the present in-cylinder analysis of the engine, partially premixed combustion mode was considered.

From the results obtained, following conclusions were drawn:

• The spark initiation in partially premixed combustion modeoccurs when the piston is very much close to TDC. This helps to achieve more compression of the charge and when ignited, higher explosion pressure is obtained.

• Partially premixed combustion mode gives a safer working environment due to less peak pressure value.

• As the region covered by the flame front is more, PPC mode gives a better combustion quality of the charge. Hence lesser smoke is emitted, fuel economy is better, carbon deposits in the combustion chamber, piston crown and exhaust system is less.

• It can also be observed that the gradient is more uniform in case of PPC which makes the operation of the engine smoother and the ill effects on the engine is also reduced.

• The profile of the kinetic energy indicates a good tumble motion. Hence combustion quality in terms of pollutant emission and acoustic emission is good in case of PPC mode.

• In case of partially premixed combustion mode the region of peak temperature remains in the spark plug point and hence the spark for the next cycle is initiated only at the spark plug. This has a major impact on noise emission, particulate emission, engine durability etc.

Thus, it can be concluded that if an SI engine is designed with PPC mode viewpoint, an efficient and safer design can be obtained.

5. FUTURE SCOPE

By studying the conclusions and other positive effects of the partially premixed combustion, further studies can be focused on fuel substitution, spark initiation timing and shape, combustion chamber geometry design changes etc. This may have a positive impact on environment, population, economy and performance.

6. REFERENCES

1. Z. Barbouchi and J. Bessrour “TURBULENCE STUDY IN THE INTERNAL COMBUSTION ENGINE,” Journal of Engineering and Technology Research, Vol.1 (9), December, 2009, pp. 194-202.
2. Pandey K. M. and Roy Bidesh “INTAKE VALVE DESIGN FOR HIGH LEVEL SWIRLINDUCTION IN CARBURETED SPARK IGNITION ENGINE,” Journal of Environmental Research And Development Vol. 7 No. 1A, July-September 2012

3. Wendy Hardyono Kurniawan, Shahrir Abdullah, Kamaruzzaman Sopian, Zulkifli Mohd. Nopiah and AzhariShamsudeen "CFD INVESTIGATION OF FLUID FLOW AND TURBULENCE FIELD CHARACTERISTICS IN A FOUR-STROKE AUTOMOTIVE DIRECT INJECTION ENGINE," Journal - The Institution of Engineers, Malaysia, Vol. 69, No.1, March 2008.
4. Heywood J.B. INTERNAL COMBUSTION ENGINE FUNDAMENTAL," McGraw-Hill International Editions, New York, 1988.
5. Anurag Mani Tripathi, Parth Panchal, Vidhyadhar Chaudhari "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,
6. Rashid Ali &Nafis Ahmad "TURBULENT FLAME VELOCITY MODEL FOR SI ENGINE," International Journal of Applied Research in Mechanical Engineering (IJARME), ISSN: 2231 –5950 Volume-1, Issue-2, 2011.
7. Wildman, Craig, and Wai K. Cheng. "THE EFFECTS OF CHARGE MOTION AND LAMINAR FLAME SPEED ON LATE ROBUST COMBUSTION IN A SPARK-IGNITION ENGINE," SAE International Journal of Engines 3(1), 2010, pp 202-213.
8. User manual – Honda CB shine.
9. Ansys 13.0 help contents.
10. Dr.Kirpal Singh, Automotive Engineering (Vol 2), Standard Publishers Distributors, ISBN: 81-8014-103-9.
11. B. E. Launder and D. B. Spalding, THE NUMERICAL COMPUTATION OF TURBULENT FLOW,Computer Methods in Applied Mechanics and Engineering. Volume 3, 1974, pp 269–289.

Author Profile

Sharath. S completed his B.E.in Automobile Engineering from PES College of engineering in 2009. During 2010-2013, he worked in MHB filter India Pvt ltd and Auram Machines India Pvt ltd in plant quality and technical marketing fields. He is now pursuing Mtech in Vidyavardhaka college of engineering, Mysuru.

Dr. G. V. Naveen Prakash completed his Ph.D from Visvesvaraya Technological University, Belagavi in 2011. He has published several papers in National and International level adding to his vast experience in teaching profession. He is currently working as professor and PG coordinator in the Dept. of Mechanical Engineering, Vidyavardhaka College of Engineering, Mysuru.