Analysis and Effect of Squish Generation Using CFD on Variable Compression Ratio C.I Engine

[*a] L.Saravanakumar, [b] B.R. Ramesh Bapu, [c] B.Durga Prasad
[d] Sharath Christy Anand, [d] R. Nagarajan, [d] E. Prasad, [d] S. Abdul Gani,
[*a]Associate Professor, Sri Sairam Engg. College, Chennai-44, India.
[b]Professor, Chennai Institute of Technology, Chennai, India.
[c]Associate Professor, JNTU College of Engg. JNTU, Ananthapur, India.
[d]Sri Sairam Engg. College, Chennai, India.

Abstract

A computational study of intake airflow system for a Direct Injection VCR engine has been carried out with modified combustion chambers. The test has been continued for various compression ratios and the performance characteristics were compared with standard piston. The comparison shows that a three dimensional model requires to predict the flow pattern of the charge motion and the effect on the performance in the engine. In general, modification in the piston bowl performs better where the swirl is lower and the effect of squish is significant. The performance of combustion system for CI engine purely depends on the nature and intensity of fluid motion. For this a diverged crescent shaped hemispherical combustion bowl has been introduced on the piston crown. Due to its unsymmetrical nature with the piston axis, there is a possibility to induce the turbulence in the intake mixture which in turn leads to thorough mixing of fuel and air within the chamber. The amount of air intake per unit engine displacement governs the quantity of fuel that can burn and hence it influences the engine power and the fuel efficiency. The design is to induce squish motion of air as the piston moves from the bottom to top dead centre; the air gets compressed during its travel from BDC to TDC and its average velocity shall be considered for the entire calculations of its travel. The air inlet pushes the air to move through the contour of the piston crown thus creating squish motion of air. The turbulence helps in complete mixing of diesel with air to form the charge. The design geometry of combustion bowl, influence the inlet charge motion were investigated.

Keywords: *Air motion, CFD, Piston crown, performance, Squish*

1. Introduction

Due to the diminishing petroleum reserves and the environmental causes, looking for alternative fuel is mandatory. In the recent years diesel engine design and its thermal efficiency has been improved much. In addition, the government has been introducing stringent emission norms for automobile emissions, particularly for roadway transportation. The air flow motion within the cylinder is one of the significant parameter to decide the combustion process. F.Payri et al. [1] stated that, in HSDE it is very critical to decide the combustion parameter near TDC and it is determined by the intake air flow charge motion at the inlet during suction process. Kono et al.[2] reported that swirl intensity effects the spray formation and the intake swirl has noticed as an important parameter. Gosman et al.[3] and Dillies et al.[4] assums that the valve as a single moving plate and designed his model as entirely different flow pattern. Marcelo J. Colaco[5] reported that the restrictive emission norms of diesel engines must require an optimized fuel such as plant derived biodiesel at once. He also proved that numerical simulation of heat transfer in the piston and in to the cylinder walls is a viable factor to optimise the combustion process. It is apparently known that, Direct Injection engines require squish air motion for complete mixture preparation. In general incorporating swirl in the combustion chamber helps improving the performance characteristics of the engine [6] Squish is nothing but the radially inward and transverse air motion that occurs at the end of the compression stroke thereby piston top and cylinder head approach each other [7]. The quantum of squish is defined by the percentage of squish area. i.e. the percentage of piston top area which closely approaches the cylinder head. The compact combustion chamber geometry may cause the squish generated air motion. However the efficiency of the VCR engine can be improved by increasing the burning rate of the mixture through the turbulence of intake air motion. The turbulence can be achieved by introducing the unique designed hemispherical bowl shape on the piston crown which is asymmetric with the axis of the piston. Also along with this squish by varying the compression ratio the turbulence of the air motion can be induced or generated. In-cylinder fluid motion is one of the essential parameters that enhance the fuel-air mixture and speeds up combustion a rate which in turn governs the performance and efficiency of the engine. The air jet enters the cylinder with higher velocity than mean speed of the piston. This intake jet interacts with cylinder walls and can create rotational flow patterns. The flow pattern depends upon the geometry of the combustion chamber contour and design of the piston crown. The flow pattern may be distinguished as swirl, tumble and squish which leads to enhancement of combustion rates and thermal efficiency. Min-Ho-Kim et al [8] reported that the air charge motion inside the intake manifold governs the performance, combustion and emission characteristics of C.I.engine. This can be accomplished in the intake manifold by modifying the design of the spacer and chamber walls in turn enhances the flow velocity and pressure distribution that improves the performance of the engine. During compression, the piston motion forces the mixture from the periphery of the cylinder to the contour on the piston that causes the radial flow towards the axis of the cylinder known as squish. In this study, the geometry of the piston bowl has been redesigned and the effect of inlet air charge motion (squish) is analysed. The result also shows that the effect of squish can enhance the performance.

2. Inlet Charge Motion Analysis

A primary study has been conducted using CFD analysis, to explore the effect of intake charge motion. Since the feasibility of Modelling and analysis in software is very viable than experimentation the Pro-E and Ansys Fluent software is used for design and analysis. The increase in performance is obtained by proper mixing of fuel and air in particular it is valid for D.I. Diesel engine. The mixing of fuel-air mixture can be enhanced by introducing squish motion in the combustion chamber. Moreover the efficiency of an engine can be improved by proper burning rate of fuel-air mixture [9,10]. In this study this is accomplished by creating a crescent shaped hemispherical bowl over the piston head for generating the squish air motion, in turn higher turbulence mixing of fuel-air. A 4stroke, VCR Kirloskar TV-1 C.I. Engine has been utilised for experiment.

The details of computational mesh are shown in Figure 4.1 and Figure 4.2 .The simulations were performed on the piston head only when it is in static position at TDC, BDC and half the way of its stroke. The valve systems were not considered for the analysis.

2.1 Piston shape and description

The engine considered for the study is a naturally aspirated engine with the specification specified in the table 1.

The combustion chamber of the engine has a hemispherical bowl at the top of the piston. Based on previous literature published in the topic of squish of in-cylinder flow, it is expected that slanted piston crown facilitate combustion at the end of compression stroke by providing in cylinder swirl, also combustion at the end of the compression stroke reduces the possibility of knocking. Therefore hemispherical bowl is replaced by a diverged crescent shaped groove which actually posses slant groove which helps in inducing air swirl when the piston nears TDC. The newly designed contour is expected to have high in-cylinder swirl and turbulence at the top dead centre in compression stroke. The fuel injection occurs at 23^{o} bTDC and is maintained constant for both the piston. Further modification will require more sophisticated testing methodology and hence with this modification is brought about in the combustion chamber of the engine is tested. Before the testing of the engine, the in-cylinder motion and the fluid flow is studied by conducting simulation study on the piston.

Table 1: Engine Specification

Engine parameter	Specifications
Bore (mm)	87.5
Stroke (mm)	110
Swept volume	661cc
Compression ratio	12-18
Power	3.5kw
Speed (rpm)	1500
Compression ratio length	12-18.1
Connecting rod length (mm)	234mm
Armlength (mm)	185

2.2 Piston structure and modelling

The standard piston has a diameter of 87.5mm and constitutes of a hemispherical bowl. The piston is made of Aluminium alloyed with Silicon, Iron, Manganese, Magnesium, Copper and Nickel. The new piston was modelled using pro/E as shown in Figure 2.1, Figure 2.2 and Figure 2.3. The piston was modified by adding material on the standard piston and machining on the surface to obtain the depicted contour. The piston was modelled to facilitate crevice flow.

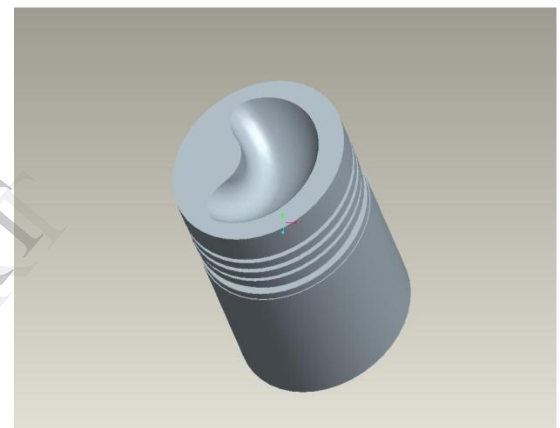

Figure 2.1 modified piston (3d isometric view)

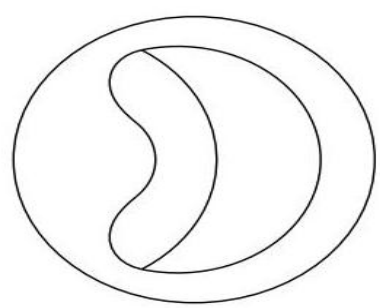

Figure 2.2 modified piston (wireframe top view)

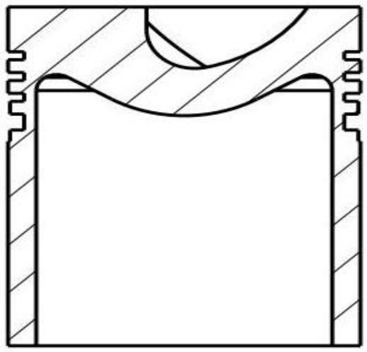

Figure 2.3 modified piston (sectional view)

3. Flow analysis- Problem setup

3.1 Pre-processing

In the pre processing setup the model which needs to be analyzed for its flow properties was modelled using pro engineer 5.0. In this a single fluid flow body is only needed for flow analysis. The necessary cavities were within the solid where the fluid flow is not required. The approach used here is top- down approach. This is converted into IGES format and imported into ANSYS workbench. This is then meshed using ANSYS mesh. The mesh created here is a triangular mesh which forms triangular elements. The name selections are created during mesh step to define various boundaries of the model under study. Here medium size discretization is done to obtain precise flow analysis.

3.2 Setup

The meshed model is imported into fluent and scaled and checked for its mesh quality. In this step the required component views were selected using the display options and the following steps were followed

- The type of flow used in this flow analysis is turbulent and the corresponding model used is k-epsilon
- Here the body is static and non deformable
- The materials used here are air as fluid and aluminium for solid

Sl. No	Fluid Zone Names	Fluid Zone mesh Requirements
1	Fluid-Bowl	Any type Of Mesh

Then the boundary conditions are specified for inlet valve, exhaust valve and the wall conditions. Finally reference here is taken as the entire body

3.3 Solution

Solution controls involves the following

Solver	***options***
Formulation	Implicit
Pressure Discretization	Standard
Momentum Discretization	Second order upwind
Turbulent kinetic energy	First order upwind
Specific dissipation rate	First order upwind
Pressure-Velocity coupling	SIMPLE

ISSN: 2278-0181
Vol. 2 Issue 9, September - 2013

The solution is then initialized using hybrid initialisation and the calculation is then performed for 200 iterations to converge at the solution.

4. Results and Discussion

The comparative study of flow simulation and experimental results of the standard and the modified piston were performed. The simulation is done using flow analysis software and the experiment was performed on a VCR engine and the results are generated.

The solution solved can be represented as vectors, contours, streamlines etc. also velocity profile at the interior of the model can be generated. In this manuscript flow of fluid at various positions of the piston such as top dead centre, travel and bottom dead centre are shown in Fig.4.3 to Fig.4.8 for both the standard and the modified piston. Also the mesh figures are shown in Fig.4.1 and Fig.4.2

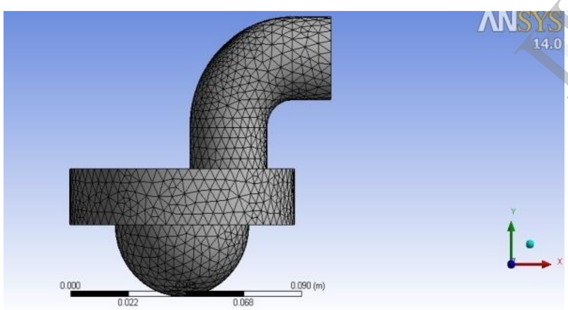

Figure 4.1 meshed view of standard piston

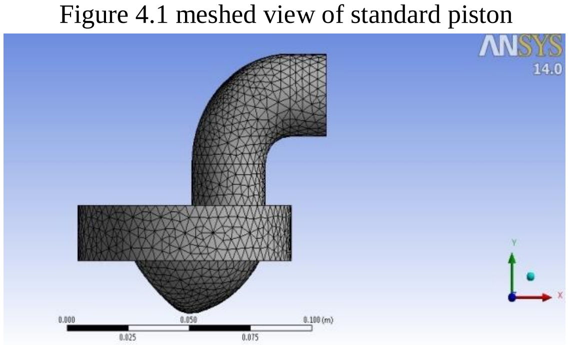

Figure 4.2 meshed view of modified piston

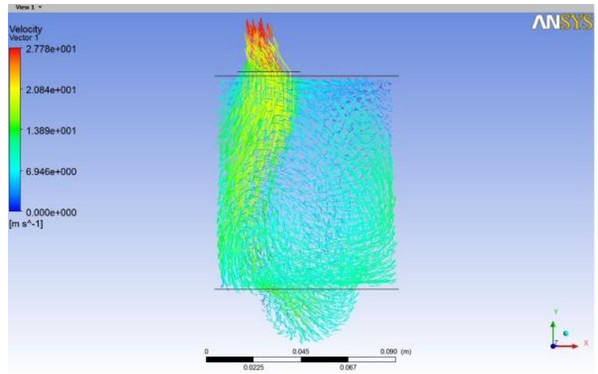

Figure 4.3 Standard piston at BDC (Front)

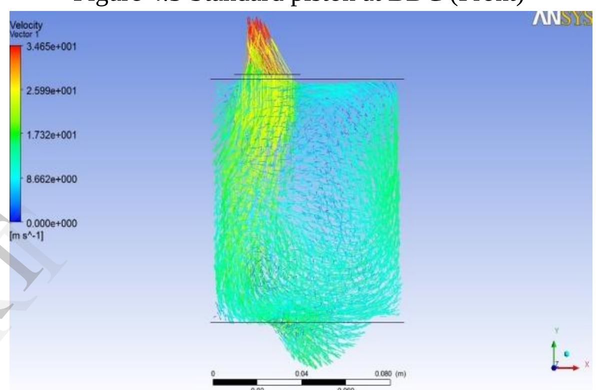

Figure 4.4 modified piston at BDC (Front)

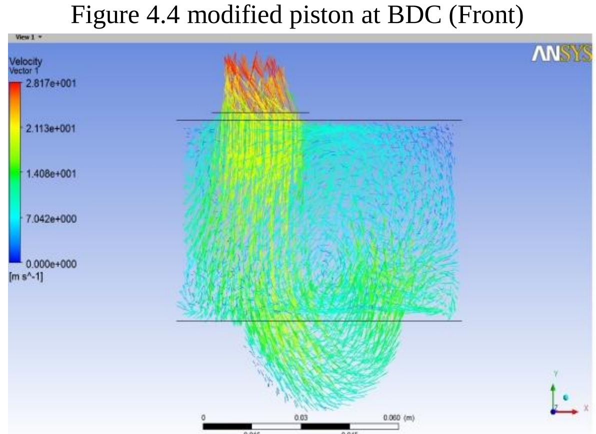

Figure 4.5 Standard piston during travel (Front)

ISSN: 2278-0181
Vol. 2 Issue 9, September - 2013

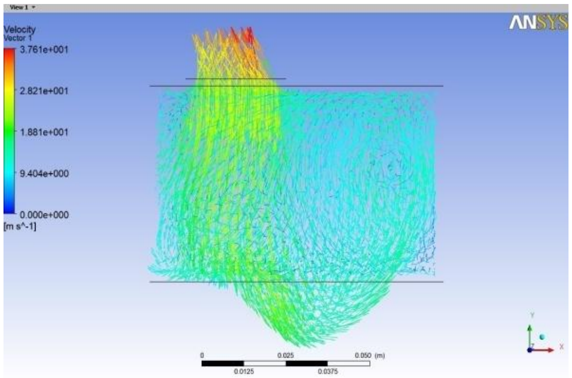

Figure 4.6 modified piston during travel (Front)

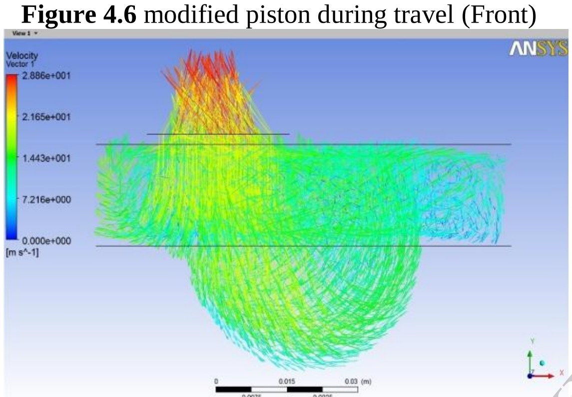

Figure 4.7 Standard piston at TDC (Front)

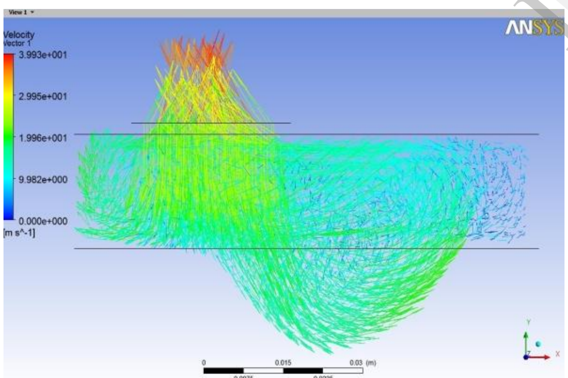

Figure 4.8 modified piston at TDC (Front)

Figure 4.3 to 4.8 shows the variations of fluid velocity at different θ values for both standard and modified piston, the velocity of the fluid entering is increasing manner while the piston travels from TDC to BDC. On comparing the inlet velocity for both stock and modified piston, there is significant increase in turbulence occurs in the modified piston and it is continued during its travel along half the way of its stroke. There is noticeable increase in velocity occurs at TDC position in the modified piston. It may result in better charge preparation which in turn may increase the burning rate and heat release rate. It is observed that the maximum squish velocity occurs at 12° bTDC and increase in squish velocity with modified piston is around 35% compare with standard piston. The CFD analysis reveals that the increase in squish velocity increases the charge air motion characteristics in modified piston engine.

4.1 Graphical Interpretation

The experiment is performed on the kirloskar TVI single cylinder 4-stroke engine with the facility to change the compression ratio. The engine is loaded by using an eddy current dynamometer. The engine load_is added in a step by step manner with a constant increment of 2 kg. The time for a specific quantity of fuel is found and the corresponding speed of the engine is noted. In this manuscript we are comparing the performance of the engine by changing the piston. The necessary graphs are included.

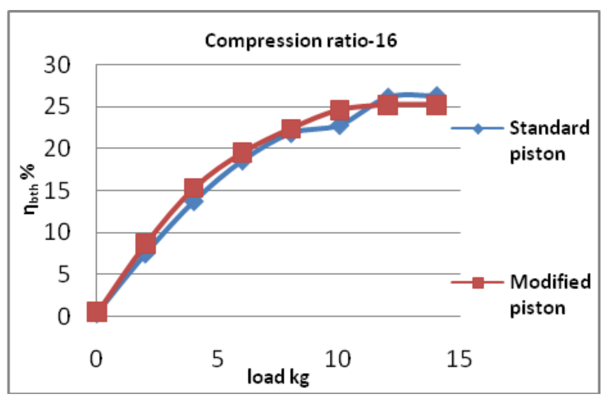

Figure 4.1.1 load vs brake thermal efficiency compression ratio-16

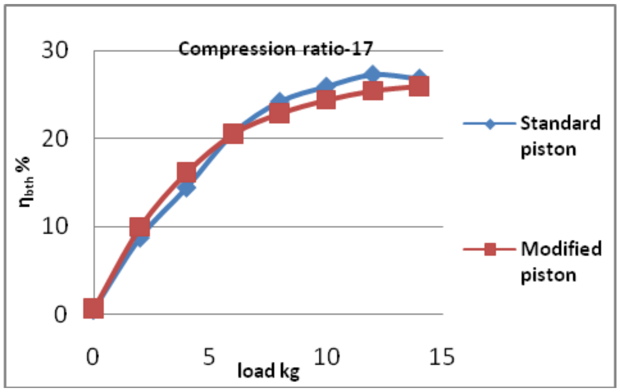

Figure 4.1.2 load vs brake thermal efficiency compression ratio-17

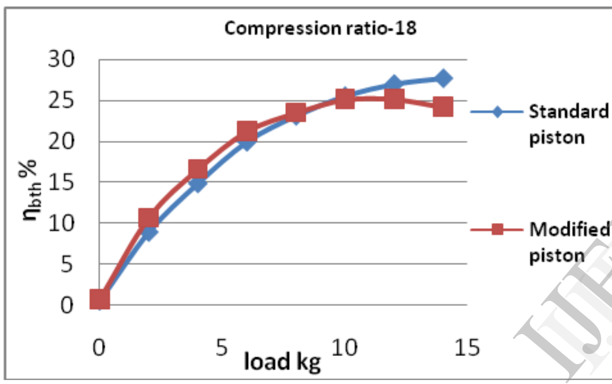

Figure 4.1.3 load vs brake thermal efficiency compression ratio-18

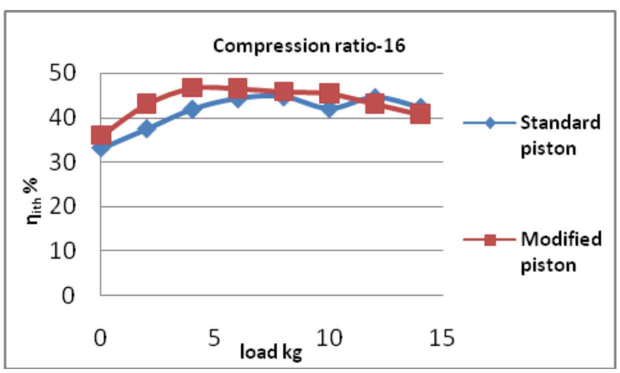

Figure 4.1.4 load vs indicated thermal efficiency compression ratio-16

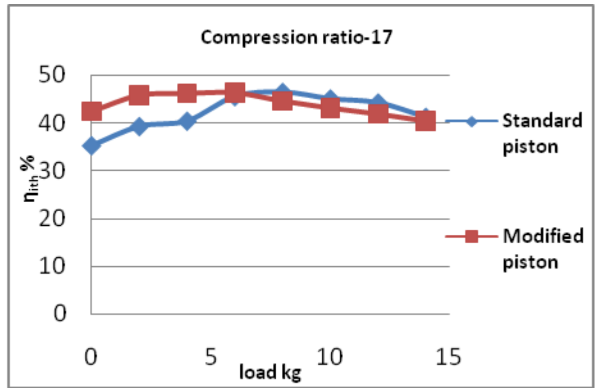

Figure 4.1.5 load vs indicated thermal efficiency compression ratio-17

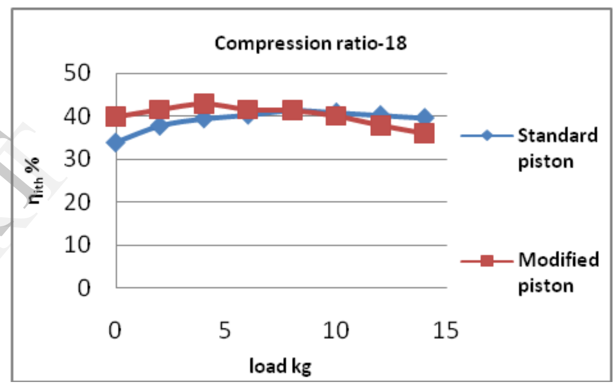

Figure 4.1.6 load vs indicated thermal efficiency compression ratio-18

5. Conclusions

The aim of the work was to induce the turbulence of intake charge through squish. The movement of air inside the combustion chamber can be brought about either by changing the orientation of the inlet manifold and the inlet valve or by changing the contour on the piston crown. Here the latter option is implemented on the compression ignition engine which invariably has a heterogeneous charge mixture for combustion. The flow analysis of air inside the combustion chamber can be simulated and analysed by using CFD analysis for both the modified piston and the standard piston. It is observed the there is significant improvement for the squish generated by modified piston than the standard piston. Squish effect will commence when the piston is about to reach the top dead centre. From the flow analysis figures, the effect of squish is less when the piston is at the bottom dead centre, when at mid way certain amount of air turbulence in terms of velocity is mildly increased. When the piston is at TDC large amount of air turbulence is formed. Using diesel as base fuel load test on the diesel engine with standard piston and the modified piston was carried out. From the above shown graphs between load vs. break thermal efficiency and load vs. Indicated thermal efficiency it is evident that both the efficiency has increased in significant manner. Here efficiency increase is pronounced at the compression ratio 16 for both indicated and brake thermal efficiency. Hopeful results were obtained those are in excellent concurrence with the test data.

6. References

[1] F. Payri, J.Benajes, X.Margot, A.Gil, CFD modelling of the in-cylinder flow in direct-injection Diesel engines, Computer & Fluids, 33 (2004), 995-1021.

[2] Kono.S, Terashita.T, Kudo H., Study of the swirl effects on spray formations in DI engines by 3D numerical calculations, SAE 910264, 1991.

[3] Gosman AD, Tsui YY, Watkins AP., Calculation of three dimensional air motion in model engines, SAE 840229, 1984.

[4] Dillies B, Ducamin A, Lebrere L, Neveu F., Direct injection Diesel engine simulation: a combined numerical and experimental approach from aerodynamics to combustion. SAE 970880, 1997.

[5] Marcelo J. Colaco, Claudio V. Teixeira, Luciana M. Dutra, Thermal analysis of a diesel engine operating with diesel-biodiesel blends, Fuel, 89 (2010), 3742-3752.

[6] S. L. V. Prasad and V. Pandurangadu. Reduction of emission by intensifying air swirl in a single cylinder DI diesel engine with manifold inlet manifold.

[7] B.P.Pundir I.C.Engines Combustion and Emissions,.

[8] Min-Ho Kim, Woo-In Chung, In-Bum Chyan. Three-Dimensional flow characteristics and engine performance for the geometry modification of intake manifold in Multi-cylinder diesel engine. Min-Ho Kim, Woo-In Chung, In-Bum Chyan.

[9] Poulos, Stephen G. And Heywood; Johns, “The effect of chamber Geometry on spark ignition engine combustion” SAE, paper 830334, 1983.

[10] Alkidas, A.C. and Suh, In-soo, “ The effects of intake flow configuration on the het release and heat transfer characteristics of a signle cylinder four valve SI Engine” SAE paper 910296, 1991.

[11] B. V. V. S. U. Prasad, C.S. Sharma, T.N.C. Anand, R.V. Ravikrishna. High swirl-Inducing piston bowls in small diesel engine for emission reduction.

[12] Dr. K. Hemachandra Reddy, Dr. M.Srinivasa Reddy. Effect of In-cylinder fluid flows and NOx emission with central squish-lip in the piston bowl of DI diesel engine: A CFD prediction.

[13] Dehong Zhang and Philip G .Hill Effect pf swirl on combustion in a short cylinder chamber, university of British Columbia V6TIZ4