International Journal of Engineering Applied Sciences and Technology, 2024
Vol. 8, Issue 11, ISSN No. 2455-2143, Pages 126-136
Published Online March 2024 in IJEAST (http://www.ijeast.com)

OPTIMIZATION AND MODIFICATION OF COMBUSTION CHAMBER FOR FOUR-STROKE SI ENGINE TO IMPROVE EFFICIENCY

Dr. N. Jagadeesh
Associate Professor,
Department of Automobile Engineering,
P E S College of Engineering, Mandya- 571401, Karnataka, India.

Tejashwini. N K
Quality Analyst Engineer,
Health Asyst Pvt. Ltd,
Bengaluru- 560008, Karnataka, India.

Abstract- **There are many inventions occurring continuously on the Internal Combustion Engine, but there are few methods to improve its efficiency. Furthermore, all of us know the cost of fuel and its availability of it in the market as well as the environmental impact of exhaust gases from engines. In order to reduce the above-mentioned problems, decided to modify combustion chamber of the engine. Modifications inside the cylinder head in internal combustion engines are one of the most important factors for controlling the combustion process. It governs the Air fuel mixing and burning rates in engines. The automotive industry is using simulation results from Computational Fluid Dynamics codes to develop and optimize new engines. Controlling the combustion process in internal combustion engines relies heavily on the fluid motion within the cylinder.**

In this work, through CFD Simulation the cylinder head is modified by making grooves for different configurations, which was compared to the non-grooved cylinder head for the performance of internal combustion engines. Engine performance experiments on an internal combustion engine were conducted. The results after conducting performance test and compared with the non-grooved cylinder head are, the fuel consumption of grooved cylinder head was decreased than non-grooved cylinder head. From the experiment conducted on the grooved cylinder head, brake thermal efficiency has increased than grooved cylinder head and it is noted that the volumetric efficiency in the grooved cylinder head is slightly increased compared to the volumetric efficiency of non-grooved cylinder head.

Key Words -**Combustion, Cylinder head, CFD, Performance, Efficiency**

I. INTRODUCTION

Performing experiments on an internal combustion engine are difficult. It is a complex mechanical system and therefore measurements are hard to carry out, especially flow measurements inside the cylinder. Numerical experiments have the advantage that an expensive and time-consuming set-up is not necessary. Because of the increasing power of computers, the processes in an internal combustion engine can be modeled in more and more detail.

Understanding the nature of the flows and combustion in internal combustion engines is important for improving engine performance. The flows in IC engines can be characterized by swirl, tumble, and compression in the cylinder. This flow motion has a strong influence on the engine combustion process and hence on the engine emission of pollutants. Recently, simulation results by Computational Fluid Dynamics codes are used in the development and optimization of new engines by car manufacturers (automotive industry). The in-cylinder fluid motion in internal combustion engines is one of the most important factors for controlling the combustion process.

Swirl and tumble are well-known approaches for in-cylinder flow enhancement 3-dimensional modeling became an important tool for investigating flow and combustion in reciprocal engines in this type of modeling, the physical processes of flow and combustion in-cylinder designed with suitable boundary conditions. The In-cylinder fluid dynamics have been shown to play an important role during the combustion process. In particular, in-cylinder fluid dynamics contributes towards fuel-air mixing which is one of the most important components for the control of the fuel-burning rate for diesel and petrol engines.

The in-cylinder flow pattern is very sensitive to the internal geometry of the intake duct and the chamber shape. In particular, the shape of the bowl-in-piston is a key parameter

controlling the turbulence level and fuel-air mixing. Even very small changes of the bowl shape may lead to important variations of the flow features such as turbulence intensity and thus considerably modify the combustion efficiency. Due to fast computing resources, the numerical solution of in-cylinder gas dynamics using Computational Fluid Dynamics (CFD) has gained prominence in the last decade. Side-by-side experimental engine testing provides an invaluable means of investigating in-cylinder fluid dynamics.
However, the customary in-cylinder velocity measurements are invasive and complicated to setup. Also, they only enable ascertaining the flow properties at a few discrete points in the cylinder. Further, these experimental tasks are quite costly and time-consuming. As a companion to experimental work, CFD analysis has emerged a useful tool in engine design. While CFD should not be used to the exclusion of experimental data due to its accuracy limitations, it does facilitate the visualization of fluid parameters throughout the domain and allows for a relatively easy modification of the domain.

II. PROBLEM STATEMENT

There are so many ways to improve the efficiency of an Engine. The efficiency can be improved by increasing fuel quality, modification in design, etc. The modification comprises port or inlet optimization, valve timing, super chargers, and combustion chamber shape. There is one more way to increase fuel efficiency and that is by making grooves on inlet valves or in the combustion chamber. From the literature, it is found that the reason increase in the efficiency of an engine.
The size and the shape of the groove are optimized using CFD. The data required for the CFD is worthy simulating software. Ansys is one of the software intended for the simulation. For a particular engine, the geometric model is created using any CAD software and geometry is imported into Ansys Fluent to simulate the airflow.
For the purpose of air engine simulation, computational fluid dynamics (CFD) codes are structured around numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power, all commercial CFD packages include sophisticated user interfaces to input problem parameters and examine the results. Hence for all codes contain three main elements.

III. DESIGN USING SOLID WORKS

SOLIDWORKS is used to develop Mechanical parts from beginning to end. At the initial stage, the software is used for planning, visual ideation, modeling, feasibility assessment, prototyping, and project management. The software is then used for the design and building of mechanical, electrical, and software elements. Here, has been used the following dimensions for designing the CD 100 engine cylinder head for CFD simulation.

Table.1 Dimensions for Designing the CD 100 Engine Cylinder Head

Elements	Dimensions
Bore	50 mm
Displacement	97.2cc
Compression ratio	18.8:1
Stroke length	49.40 mm
Inlet valve diameter	22 mm
Exhaust valve diameter	20 mm

The steps carried out in Solid works models are as revealed:

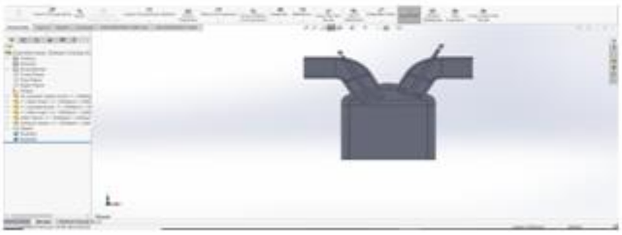

Figure 1 Design of Overall Cylinder without Groove

The figure 1displays the overall geometry of CD 100 cylinder in Solid works using cylinder dimensions are shown in the table1.

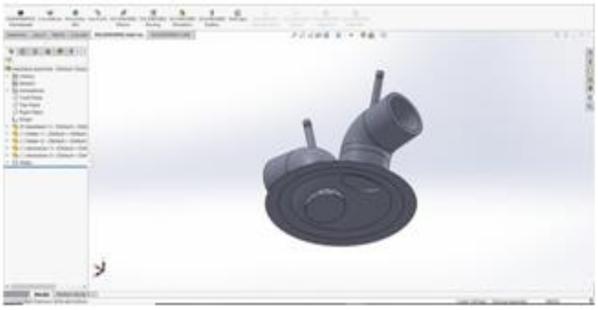

Figure 2 Design of Cylinder Head without Groove

The figure 2 shows the geometry of CD 100-cylinder head without groove in Solid works by means of the cylinder head dimensions displayed in the table1.

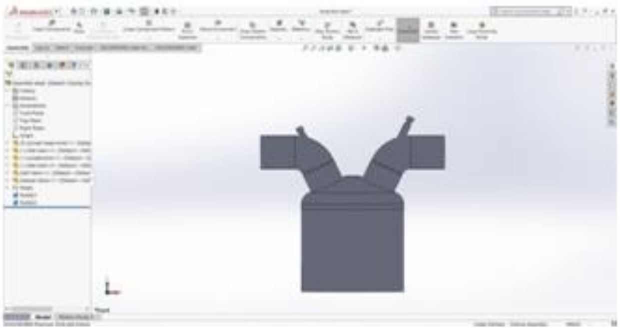

Figure 3 Design of Overall Cylinder of CD 100 Engines

Figure 4 Design of Cylinder Head with Groove

The figure 3 displays the overall geometry of CD 100 cylinder with in-let and exhaust valves in Solid works using the cylinder dimensions shown in the table1.

The figure 4 shows geometry of CD 100-cylinder head with groove which was done in Solid works using the samedimension displayed in the table1.

CFD Analysis Process

The general process for performing a CFD analysis is outlined underneath so as to provide a reference for understanding the various aspects of a CFD simulation. The process includes:

- Model the Geometry and Flow Domain
- Establish the Boundary and Initial Conditions
- Generate the Grid
- Establish the Simulation Strategy
- Establish the Input Parameters and Files
- Perform the Simulation
- Post-Process the Simulation to get the Results

A. Model the Geometry and Flow Domain

The body about which flow is to be analyzed requires modeling. This generally involves modeling the geometry with a CAD software package. Approximations of the geometry and simplifications may be required to allow an analysis with reasonable effort. Concurrently, decisions are made as to the extent of the finite flow domain in which the flow is to be simulated. Portions of the boundary of the flow domain coincide with the surfaces of the body geometry. Other surfaces are free boundaries over which flow enters or leaves. The geometry and flow domain are modeled in such a manner as to provide input for the grid generation.

B. Establish the Boundary and Initial Conditions

The finite flow domain is specified, physical conditions are required on the boundaries of the flow domain. The simulation generally starts from an initial solution and uses an iterative method to reach a final flow field solution.

Intake Velocity and Constant pressure is used as boundary condition at both the intake and the exhaust manifolds. Attach boundaries are specified on the coincident cell face near the cells above/below the valve. No slip wall boundary condition in conjunction with logarithmic law of wall is used. Walls are considered to be adiabatic. Hydraulic diameter 0.02m for S I engine and turbulent intensity 5% for SI engine. Engine parameters are shown in table.2

C. Generate the Grid

From the figure 5 shows that, the flow domain is discretized into a Mesh. The Mesh generation involves defining the structure and topology and then generating a grid on that topology. Currently all cases involve multi-block, structured mesh, the mesh blocks may be abutting, contiguous, non-contiguous, and overlapping.

The mesh should exhibit some minimal mesh quality as defined by measures of orthogonality (especially at the boundaries), relative mesh spacing (15% to 20% stretching is considered a maximum value), and mesh skewness. The resolution of boundary layers requires the grid to be clustered in the direction normal to the surface with the spacing of the first mesh point off the wall to be well within the laminar sub layer of the boundary layer.

Table.2 Boundary and Initial Conditions

Inlet	
Velocity Specification Method	Magnitude, Normal to Boundary
Reference Frame	Absolute
Velocity Magnitude[m/s]	8.72
Turbulent Specification Method	Intensity and Viscosity Ratio
Turbulent Intensity[%]	5
Turbulent Viscosity Ratio	10
Outlet	
Backflow Reference Frame	Absolute
Turbulent Specification Method	Intensity and Viscosity Ratio
Backflow Turbulent Intensity[%]	5
Backflow Turbulent Viscosity Ratio	10

International Journal of Engineering Applied Sciences and Technology, 2024
Vol. 8, Issue 11, ISSN No. 2455-2143, Pages 126-136
Published Online March 2024 in IJEAST (http://www.ijeast.com)

Backflow Pressure Specification	Total Pressure
Wall	
Wall Motion	Stationary Wall
Shear Boundary Condition	No Slip

Figure 5 Generation of Mesh

D. Establish the Simulation Strategy
The strategy for performing the simulation involves determining such things as the use of space-marching or time-marching, the choice of turbulence or chemistry model, and the choice of algorithms.

E. Establish the Input Parameters and Files
A CFD code generally requires that an input data file be created listing the values of the input parameters consistent with the desired strategy. And a mesh file containing the grid and boundary condition information is generally required. The files for the grid and initial flow solution need to be generated.

Simulation
The simulation is performed with various possible options for interactive or batch processing and distributed processing.

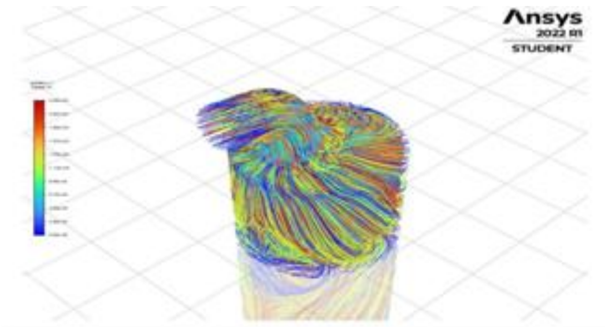

Figure 6 CFD Simulations of Cylinder Head

The figure 6 displays the simulation performed on the overall cylinder geometry which was imported from solid works by giving boundary and initial condition shown in the Table 2.

Post-Processer
Post-Process the Simulation to acquire the Results

This involves extracting the desired flow properties from the computed flow field. The figure 7 displays the simulation performed on the overall cylinder geometry with variation of turbulence Kinetic energy.

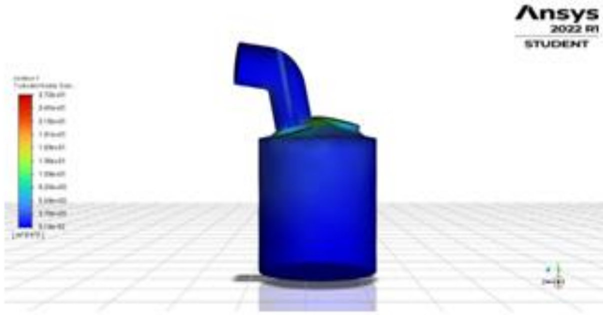

Figure 7 CFD Simulations of Cylinder Head

Selection of optimized groove
After simulating turbulence for various configurations of grooves, the triangular groove having the dimension 4mm x 7mm x 2mm (base x height x depth)is found to be producing more turbulence with less change in compression which is shown in the figure 8.Therefore, the same groove is taken for machining and the performance is conducted for the same.

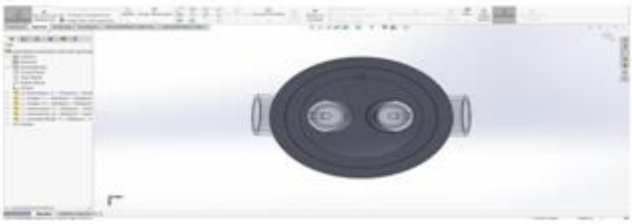

Figure 8 CFD Simulations of Grooved Cylinder Head

Machining
In this work it has been chosen CNC machining for making the groove on the cylinder head

Process carried out in CNC Machining
- Designing the CAD model
- Converting the CAD file to a CNC program
- Preparing the CNC machine
- Executing the machining operation

Designing the CAD model
The CNC machining process begins with the creation of a 2D vector or 3D solid part CAD design either in-house or by a CAD/CAM design Service Company. Computer-Aided Design (CAD) software allows designers and manufacturers to produce a model or rendering of their parts and products along with the necessary technical specifications, such as dimensions and geometries, for producing the part or product.

CAD File Conversion
The formatted CAD design file runs through a program, typically Computer Aided Manufacturing (CAM) software, to extract the part geometry and generates the digital programming code which will control the CNC machine and manipulate the tooling to produce the custom-designed part.
CNC machines used several programming languages, including G-code and M-code. The most well-known of the CNC programming languages, general or geometric code, referred to as G-code, controls.

CNC CODE

```
G21
G90
G00 X0 Y0
G01X7.28Y0
G01 X5.717175 Y0.199655
G01X0 Y0
G00X0Y0
```

Above is the CNC code for the Groove which was simulated using CFD analysis, with the dimension (4x7x2) which done on the squish band with pointed edge sand flat bottom. This shape is act as nozzle which can throw the air-fuel mixture with high intense velocity.

Machine Setup
Before the operator runs the CNC program, he must prepare the CNC machine for operation. These preparations include affixing the work piece directly into the machine, onto machinery spindles, or into machine vises or similar work holding devices, and attaching the required tooling, such as drill bits and end mills, to the proper machine components.

Machining Operation Execution
The CNC program acts as instructions for the CNC machine; it submits machine commands dictating the tooling's actions and movements to the machine's integrated computer, which operates and manipulates the machine tooling. Initiating the program prompts the CNC machine to begin the CNC machining process, and the program guides the machine throughout the process as it executes the necessary machine operations to produce a custom-designed part or product.

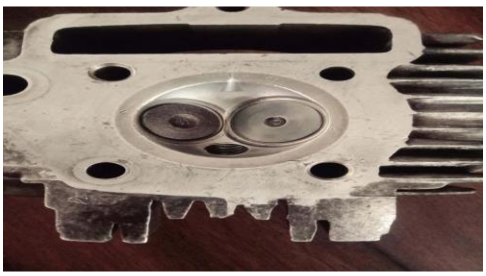

Figure 9 Cylinder Head Machined with Grooves

The Groove is machined with the reference of the result from the simulation; the triangular groove with the dimensions 4mm x 7mm x 2mm is found more optimistic producing more turbulence with minimum change in the compression ratio from the figure 9.

Table.3 Engine Specifications

Engine maker	**Hero Honda**
Working Cycle	Two Stroke
Compression Ratio	8.8:1
BHP	7.5hp@5500rpm
Engine Speed	3000rpm
Engine Stroke	49.9mm
Engine Bore	50mm
Method Of Cooling	Air Cooled
Method Of Ignition	Spark Ignition
Air box orifice diameter	20mm
Radius of Dynamo Arm	0.2m

PERFORMANCE TEST
The performance test was conducted on the Stock HERO HONDA CD 100 Engine, and also the CD 100 Engine with the optimized groove. The results were calculated for both the engine configurations and were parallelly compared. The engine specifications are given beneath in table 3.
An eddy current dynamometer was used to test the engine while maintaining a constant load and varying speed. The results, such as mechanical efficiency and brake thermal efficiency, are obtained. Analyses are conducted on the results.

Maximum Load Calculations:
For Engine,
BHP=$2\pi \times N \times T/4500 \times G$ & $T= W \times r$
Where, N= Rated speed of the engine, rpm
T=Engine torque, N-m
G=Gear reduction
W= Load on the engine, Nr=Radius of the arm, m
T=5.64 kgf-m & W=28.24 kgf
For Dynamometer,
Td=Wd*rd
Where, Td= Dynamometer torque, N-m
Wd=Force on a dynamometer, N

W=2/0.2 = 10 kgf

Observations:

Table 4 Cylinder Head without Grooves.

Sl. No	Dynamo Speed(rpm)	Load (Kg)	Time Taken to Consume 10cc of Fuel t(sec)	Manometer Reading (h1-h2)	Temperature
1.	400	0	78.64	1.6	220
2.	400	2	77.87	1.8	252
3.	400	4	76.55	1.8	268
4.	400	6	73.48	1.9	289
5.	400	8	69.67	2	302

Table.5 Cylinder Head with Groove

Sl. No	Dynamo Speed(rpm)	Load (Kg)	Time Taken to Consume 10cc of Fuel t(sec)	Manometer Reading (h1-h2)	Temperature
1.	400	0	143	1.6	232
2.	400	2	135	1.8	289
3.	400	4	132	1.9	292
4.	400	6	129	2.0	302
5.	400	8	122	2.1	308

The overhead table 4 displays the readings taken down from performance test conducted on cylinder head without grooves keeping speed as constant at 400 rpm and varying the load. The table 5 shows the readings taken down from performance test conducted on cylinder head with grooves of dimension (4mm x 7mm x 2mm) keeping speed as constant at 400 rpm and varying the load.

Table 6–Results of Cylinder Head without Groove

Sl. No	Load (Kg)	Speed (rpm)	Time Taken to Consume 10cc Of Fuel t(Sec)	Brake Power (KW)	Mass Of Fuel Consumed (Kg/hr)	Brake Specific Fuel Efficiency BSFC (Kg/Kw-hr)	Break Thermal Efficiency (%)	Volumetric efficiency (%)
1.	0	400	78.64	0	0.3433	0	0	81.20
2.	2	400	77.87	0.16436	0.34673	2.1095	3.750	86.13
3.	4	400	76.55	0.32873	0.3527	1.07291	7.3743	86.13
4.	6	400	73.48	0.4931	0.3659	0.74204	10.66	88.48
5.	8	400	69.67	0.6574	0.3875	0.58944	13.424	90.79

The table 6 displays the calculated result sheet for the readings taken from table4 and Table7 shows the calculated result sheet for the readings taken from table 5.

III. RESULT AND DISCUSSIONS

The results from the modeling and CFD simulation using FLUENT results are revealed and discussed. Results are shown in term of contours and graphs for the simulation results for velocity distribution, turbulent kinetic energy and turbulent intensity.

Table7–Results of Cylinder Head with Grooves

Sl. No	Load (Kg)	Speed (rpm)	Time Taken to Consume 10cc Of Fuel t(Sec)	Brake Power (KW)	Mass Of Fuel Consumed (Kg/hr)	Brake Specific Fuel Efficiency BSFC (Kg/Kw-hr)	Break Thermal Efficiency (%)	Volumetric efficiency (%)
1.	0	400	143	0	0.1888	0	0	81.20
2.	2	400	135	0.16436	0.20	1.01728	6.4698	86.13
3.	4	400	132	0.3287	0.2045	0.62214	12.7137	88.48
4.	6	400	129	0.4931	0.21835	0.4428107	17.867	90.79
5	8	400	122	0.6574	0.2213	0.336629	23.50	93.03

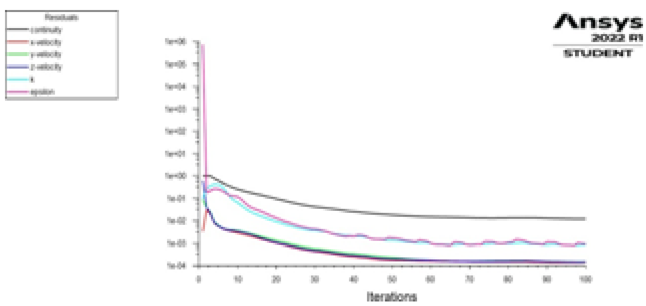

Figure 10 Simulation Graph for Non-grooved Cylinder Head

International Journal of Engineering Applied Sciences and Technology, 2024
Vol. 8, Issue 11, ISSN No. 2455-2143, Pages 126-136
Published Online March 2024 in IJEAST (http://www.ijeast.com)

Figure 10 displays the continuity, k and epsilon values and the velocity of gasoline vapor in x, y, z direction coordinates on Y axis with number of iterations on X axis.

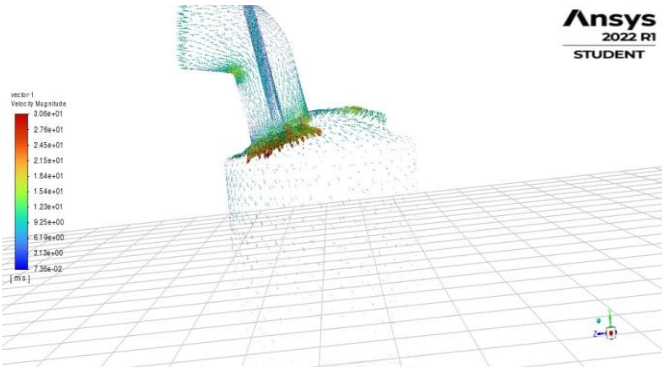

Figure 11 Velocity Distribution Contour in Non-grooved Cylinder Head

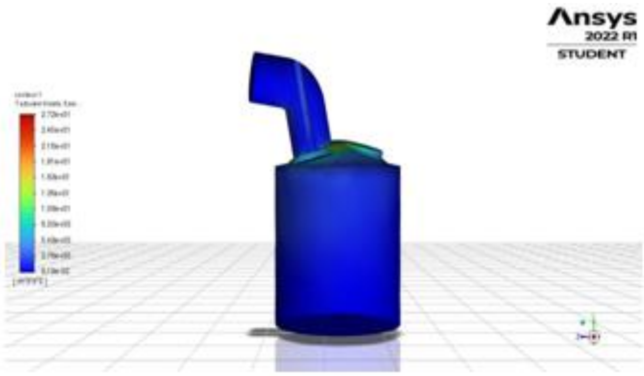

Figure 12 Turbulence Energy Contour in Non-grooved Cylinder Head

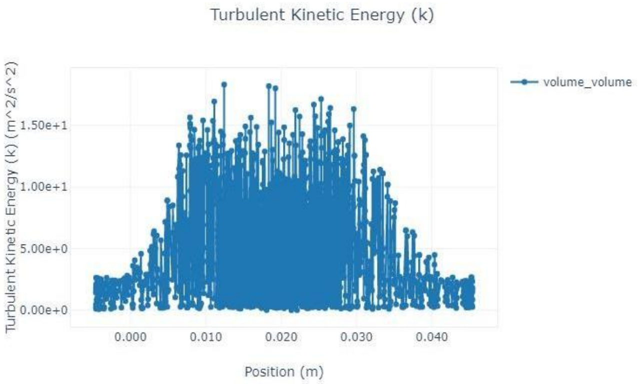

Figure 13 Turbulence Variations in Non-grooved Cylinder Head

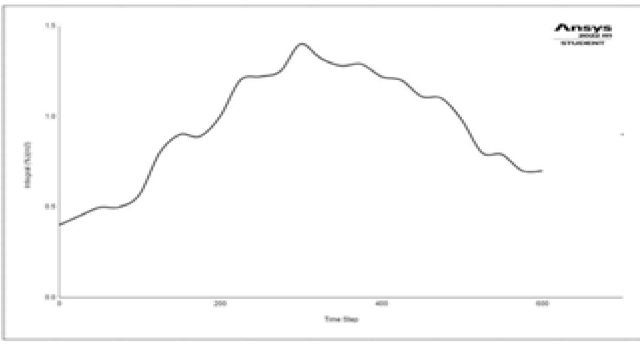

Figure 14 Turbulence Velocities in Non-grooved Cylinder Head

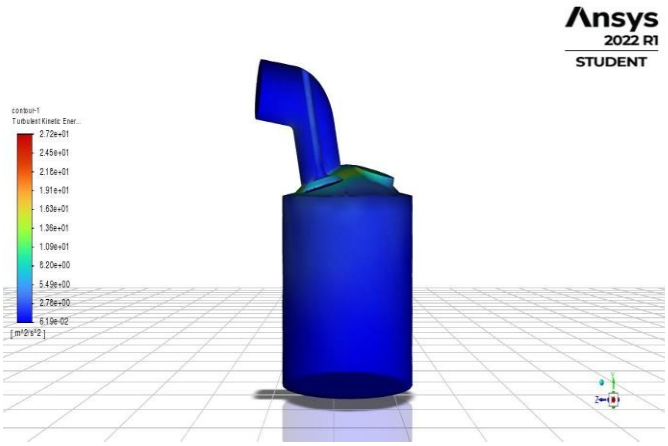

Figure.15 Turbulence Energy Contour in Grooved Cylinder Head

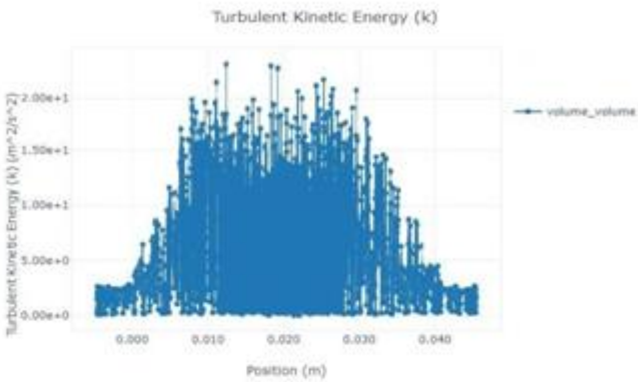

Figure 16 Turbulence Variations inside Grooved Cylinder Head

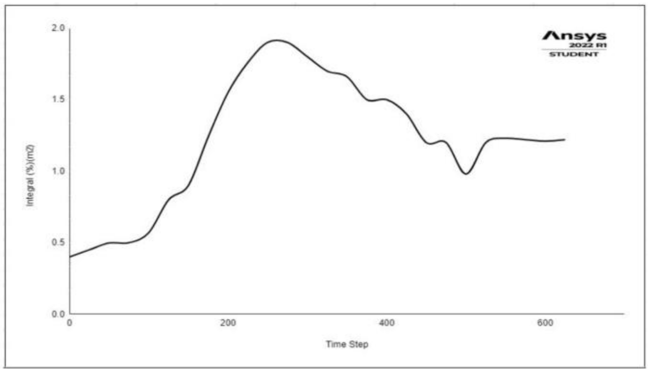

Figure.17 Turbulence Intensities in Grooved Cylinder Head

The figure.11and figure.12 shows the velocity and the turbulence kinetic energy contours in a non-grooved cylinder. Figure.13 displays the turbulence intensity with respect to the position. The intensity of turbulence in non-grooved cylinder is shown in figure 14.
Figure 14 and figure 15 show the velocity and the turbulence kinetic energy contours in a grooved cylinder. The turbulence intensity with respect to the position is shown in Figure 16. The intensity of turbulence in grooved cylinder is exposed in figure 17.
The brake thermal efficiency increases to produce the same brake power as shown in Figure, due to reduced fuel consumption as shown in Figures16 and 17.
The figure 18 displays the comparison of turbulence intensity for grooved and non-grooved cylinder head; it is found that the turbulence intensity is increased in triangular groove cylinder head having the dimension 2 mm x 4 mm x 7mm.
Experimental investigations were carried out for performance of the conventional SI engine and modified grooved engine. The test results obtained from the comprehensive experimental investigations are analyzed and described below.

Comparison of Grooved and Non-grooved Cylinder Head Graphs

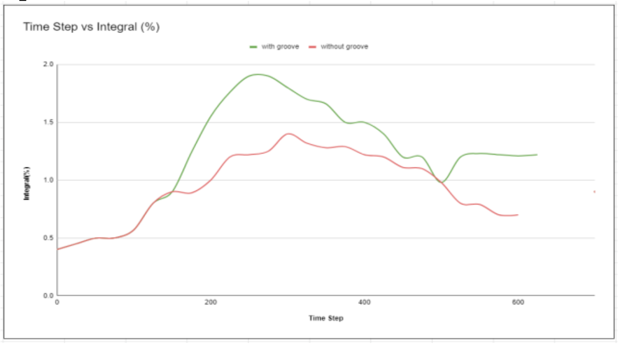

Figure 18 Comparison of Turbulence Intensity inside Non grooved and Grooved Cylinder Heads

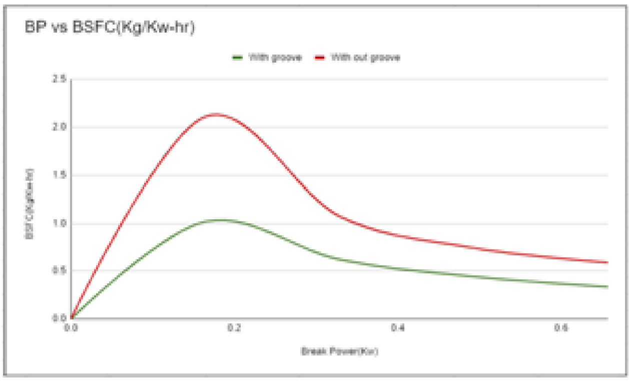

Figure 19 Comparison of Grooved and Non-grooved Cylinder Head for Brake Power versus Brake Specific Fuel Consumption

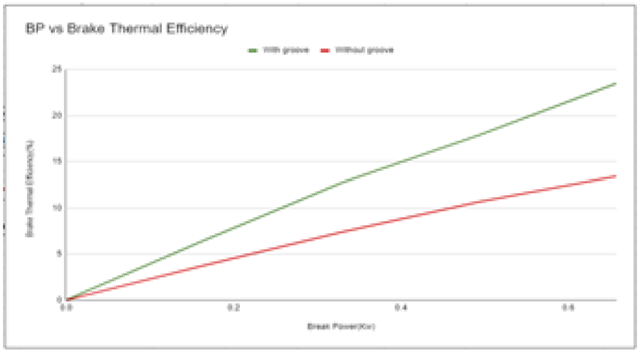

Figure 20 Comparison of Grooved and Non-grooved Cylinder Head for Brake Power versus Brake Thermal Efficiency.

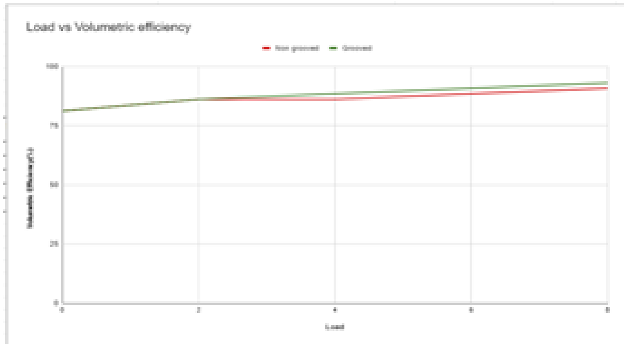

Figure 21 Comparison between Load and Volumetric Efficiency for Grooved and Non-grooved Cylinder Head

The grooved cylinder head show that the fuel consumption less than that of the original cylinder head as shown in figure 19 it is because the groove created acts as nozzle which create more intense velocity inside the cylinder head which increases the turbulence. According to Figure20, grooved cylinder head consume less fuel compared to the original cylinder head consumes less fuel than that of normal.

The increase in turbulence causes increase in flame propagation and less time is required to propagate the flame from spark plug to squish band. As the groove is made on a squish band it allows the flame to propagate into the fuel which is trapped between the squish band and the top of the piston when it is at TDC. Because of the groove the surface area to propagate the flame increases and makes path to flow in between the squish band and the top of piston surface. Hence at the same time flame propagates from spark plug to squish as that of from spark plug to the exact below surface of piston.

The brake thermal efficiency increased while the brake power increases this is due to, the less fuel consumed which is shown if Figure.19 therefore increases in the total brake thermal efficiency which is further shown in Figure.20. Further the original cylinder head has less brake thermal efficiency than that of the modified cylinder head.

The efficiency of the grooved cylinder headed engine has been increased in comparison with the original cylinder headed engine; result has been shown in table. In modified cylinder head the fuel consumption is still reduced than normal engine. This may be because the high intensity Air fuel mixture inside the cylinder head which helps in complete combustion of fuel.

The modified cylinder head with groove shows that the fuel consumption is less than that of the original cylinder head geometry because the groove produces more turbulence which increases the flame speed and hence improved efficiency.

IV. CONCLUSION

In the present work, simulation and performance tests were carried out on SI four stroke engines with conventional cylinder heads and modified cylinder heads. From the outcomes, it was observed that a grooved engine with groove modification requires lesser fuel consumption for the same distance travel as compared to an engine with a conventional engine cylinder head. As the fuel economy of the grooved engine is more than the conventional engine, it is observed that brake-specific fuel consumption of the grooved engine is a smaller amount as compared to a conventional SI engine.

By analyzing turbulence flow results from CFD analysis, it has been suggested that by modifying cylinder head geometry it is possible to increase the fuel flow rate inside the cylinder head. The volumetric efficiency in the grooved cylinder head has increased with increase in the brake thermal efficiency, expansion and compression happens to Air-fuel mixture inside the cylinder head in the grooved cylinder.

From the existing results, it is not feasible to comment on the emission rate of any particular pollutant. To comment on that, it is required to carry out a number of emission tests on the same setup.

The conclusions originating from a present experimental investigation which is conducted on 4-stroke, single cylinder, air cooled petrol engine by using different configurations of grooves on the cylinder head and a conventional cylinder head is that from the first set of results it can be concluded that the groove configuration has given the better performance in the sense of brake specific fuel consumption, acceleration performance and emission parameters.

V. REFERENCE

[1]. Somender Singh, (29 May 2001), “Design to Improve Turbulence in Combustion Chambers”. US Patent # 6,237,579.

[2]. R. Deshmukh & A. B. Atpadkar: (2019), “Analysis of I.C. Engine to Improve Performance Due to Grooves on Engine Cylinder Head”, E-ISSN: 2395-0056, P-ISSN:2395-0072,Volume:06 ,Issue: 07.

[3]. Dr. S. L.V. Prasad, Prof. V. Pandurangadu, Dr. P Manoj Kumar & Dr G. Naga Malleshwara Rao: (2013), “Enhancement of Air Swirl in a Diesel Engine with Grooved Cylinder Head” ISSN:2319-8753, Volume-2, Issue-8.

[4]. Mahmut Kaplan (2019), “Influence of Swirl, Tumble and Squish Flows on Combustion Characteristics and Emissions in Internal Combustion Engine Review” E-ISSN: 2146-9067.

[5]. J Paul Rufus Babu, B MadhuBabu, T Dada Khalandar and P S Bharadwaj: (2015), “Experimental Investigation of Rhombus Shaped Grooves on Piston Crown of a SingleCylinder4-StrokeDiDieselEngine”, ISSN:2278-0149, Volume 4, Number1.

[6]. B. Madhubabu, Prof. K. Govindarajulu and Dr. S.L.V. Prasad: (2014),“Experimental Investigation of a Single Cylinder 4- Stroke Diesel Engine by Swirl Induction with Two Different Configuration Pistons”, ISSN:2315-4847,Volume.3, Issue.11.

[7]. Dr. S L V Prasad, V. V Naga Deepthi, Vootukuri Pandurangudu &V.V. Pratibha Bharathi: (2011), “Experimental Study of The Effect of In Cylinder Air Swirl on Diesel Engine Performance and Emission”, ISSN: 0975-5462, Volume3, Number 2.

[8]. Dr. Abdul Siddique, Shaik Abdul Azeez & Raffi Mohammed, (2016), “Simulation and CFD Analysis of Various Combustion Chamber Geometry of a C.I Engine Using CFX”, E-ISSN:2319-1831, P-ISSN:2319-1821,Volume.5, Issue.8.

[9]. Yogendra Aniya & Sharad Chaudhary, (2019), “CFD Analysis of In-Cylinder Flows in Single Cylinder SI Engine”ISSN:2456-3315, Volume. 4, Issue. 8.

[10]. Madhusudan Barot, Prof. Abhishek Shah & Prof. Mit Patel:(2017), “CFD Analysis of Single Cylinder Four Stroke Gas Fuelled Engine for Prediction of Air Flow Rate During Suction Stroke”, ISSN: 2321-9939, Volume.5, Issue. 2.

[11]. A.M. Indrodia, N.J. Chotai & B.M. Ramani: “Investigation of Different Combustion Chamber Geometry of Diesel Engine Using CFD. Modelling of

in Cylinder Flow for Improving the Performance of Engine”. (AIMTDR 2014).

[12]. K.M. Pandey & Bidesh Roy, (2012), “CFD Analysis of Intake Valve for Port Petrol Injection SI Engine” ISSN: 0975-5861 VOLUME12 -ISSUE 1.

[13]. V.V. Pratibha Bharathi and Dr. Smt. G. Prasanthi: “Investigation on The Effect of EGR With Diesel and Grooved Piston with Diamond Mesh Cut In an Internal Combustion Engine” e-ISSN: 2249-4596 p-ISSN: 0975-5861 VOLUME 14, ISSUE 2 VERSION 1. 2014.

[14]. Dr. P.N. Shrirao, R. M. Shrerekar & P. K. Bhoyar: “Effect of Air Swirl with Dimpled Cylinder Head on Performance Characteristics of Diesel Engine” ISSN: 2248-9347, ISSN: 2248-9355, Volume. 4, Number. 2, (2014).

[15]. F. Payri, J. Benajes, X. Margot &A. Gil, “CFD Modeling of the in-cylinder flow indirect-injection Diesel engines”, 2003.

[16]. Kota Sridhar, R. B. V. Murali, Sk. Mohammad Younus & K. Mohan Lakshmi," Computerized Simulation of Spark Ignition Internal Combustion Engine”, ISSN: 2278-1684,Volume 5,Issue 3, (2013).

[17]. S. Gavudhama Karunanidhi, Melvin raj C R, Sharath Das K P & G SubbaRao,“ CFD Modeling and Study of Combustion in Diesel Engine for Different Grid Generation”, ISSN: 2248-9622 Vol. 3, Issue. 4, 2013.