Computational Fluid Dynamics Simulation of A VCR Research Engine Working With Cold Flow Analysis

S.Prakash, M.Prabhahar, Obeth Eanest.B, Prabakaran.N, Prasanth. D

ABSTRACT: The fundamental goal of the task is to exhibit the aftereffects of the computational liquid elements reproduction of an immediate infusion single chamber motor utilizing diesel, biodiesel, or diverse blend extents of diesel and biodiesel and contrast the outcomes with a proving ground estimation in a similar working point. The motor utilized for checking the consequences of the reproduction is a solitary chamber explore motor from AVL with an open motor control unit, so the infusion timings and amounts can be controlled and broke down. In Romania, until the year 2020 all the fuel stations are obliged to have blends of in any event 10% biodiesel in diesel. The principle points of interest utilizing blends of biofuels in diesel are: the way that biodiesel isn't destructive to the earth; so as to utilize biodiesel in your motor no adjustments are required; the cost of biodiesel is littler than diesel and furthermore on the off chance that we contrast biodiesel creation with the great oil based diesel generation, it is more vitality proficient; biodiesel guarantees more oil to the motor so the life of the motor is expanded; biodiesel is a practical fuel; utilizing biodiesel keeps up nature and it keeps the individuals increasingly solid.

Keywords: CFD, Injection timings.

I. INTRODUCTION

The performance of an IC Engine depends upon complex interactions between mechanical, fluid, chemical, and electronic systemsThe laws for mass, force, and vitality transport on a 3D geometry, with sub-models for basic marvels like choppiness and fuel science. Understanding gave by CFD examination helps manage the geometry structure of parts, for example, ports, valves, and cylinders; just as motor parameters like valve timing and fuel infusion. The pollutants include oxides of nitrogen, sulfur oxides (SOx), CO (carbon monoxide), un-burned hydrocarbons (HC), and Poly Aromatic Hydrocarbons (PAH or “soot”), which are all products of the combustion process. The volumetric proficiency of the motor relies upon a few liquid powerful wonders in the admission and fumes tracts prompting the burning chamber. At the point when the air is siphoned into the ignition chamber during the admission cycle, it goes through the hole between the valve and the valve seat. As it crushes through the hole, the stream isolates from the dividers of the port and valve surfaces, framing a distracting plane. The stream from the valves encroaches on the chamber dividers and tumbles into the space between the valves and the cylinder. On the off chance that there is solid whirl (generally depicted by a standardized rakish force an incentive about the vertical pivot through which the cylinder movement is obliged) the stream may create stratification with areas of high and low speed. The admission port might be intended to grant extra rakish force to the air; and various admission valves or any halfway open fumes valves may have stream communication.

Revised Manuscript Received on .

***** Correspondence Author
S.Prakash -Research Scholar, Department of Mechanical Engineering, Aarupadai Veedu Institute of Technology, Vinayaka Mission Research Foundation, Deemed to be University Prakash.mech94@gmail.com
Dr. M.Prabhahar - Professor, Department of Mechanical Engineering, Aarupadai Veedu Institute of Technology, Vinayaka Mission Research Foundation, **Deemed to be University mprabhahar@gmail.com**
Obeth Eanest.B, Prabakaran.N, Prasanth. D, UG Scholars, Aarupadai Veedu Institute of Technology, Vinayaka Mission’S Research Foundation

II. MODELING CFD IN IC ENGINE DESIGN

The valves can be positioned as "straight", i.e. the valves are aligned with the cylinder axis. They are at an angle to the cylinder axis and normal to the surfaces of the combustion chamber (Figure 1.1: Kirloskar Valve Engine).

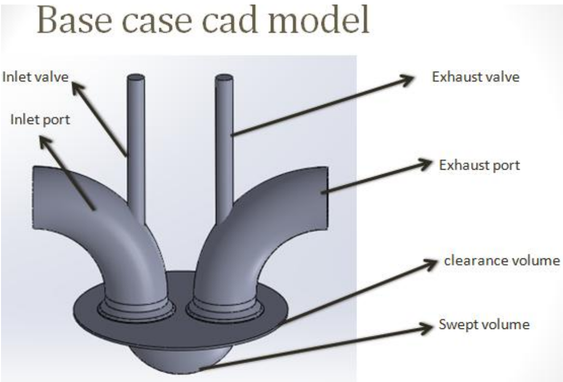


Figure 1 - Kirloskar Valve Engine

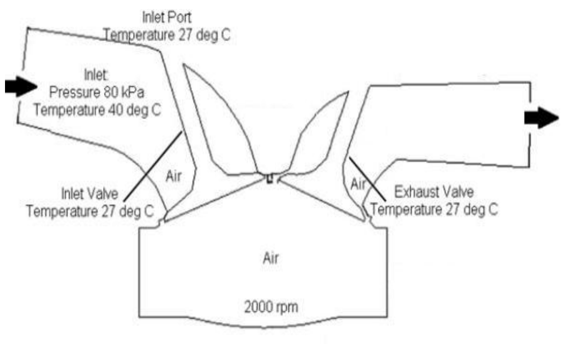


Figure 2 – Designed Model of Kirloskar Valve Engine

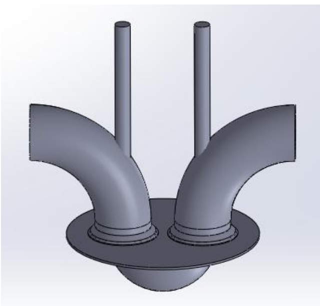
There are several tools which are used in practice during the design process.
Once the analysis has been set up, it takes many hours or days of computation to get the solution and evaluate the results.

III. DESIGN AND ANSYS FLUENT

The Kirloskar IC engine designed using Catia,
The Model designed for one Cylinder, Bore 87.5mm, Stroke – 110mm,
Opens the Properties panel where you define the parameters and controls required for the analysis. It controls the behaviour of the downstream components. For details on configuring the properties, refer to Setting up an IC Engine Analysis System for IC Engine (Fluent). Geometry Opens Design Modeller where you define the geometrical constraints of your analysis or import an existing geometry. You will prepare the geometry for decomposition (divide the geometry into smaller volumes before meshing. The compression ratio the clearance volume as been modified has 0.75, 1.00 and 1.25.

0.75mm clearance volume

1.00 mm clearance volume



1.25 mm clearance volume

Figure 3 - Modified cad model with different clearance volume

A. MESHING

In computational fluid dynamics, meshing is a discrete representation of the geometry that is involved in the problem. Several parts of the mesh are grouped into regions where boundary conditions may be applied to solve the problem. Moreover, the uses of meshing are not limited to computational fluid dynamics. Also, meshing can be used to solve partial differential equations using numerical techniques. Meshing is carried out in ansys workbench. Hybrid meshing approach is used (combination of hexahedral and tetrahedral mesh). Around 5lakh mesh elements were used for the analysis.

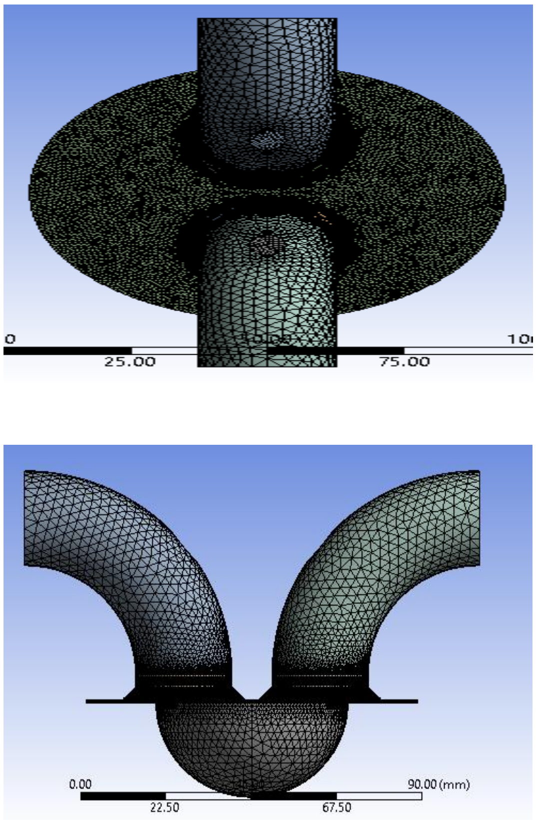


Figure 4: Mesh Images (Front View and Top View)

IV. RESULT AND DISCUSSION

Ansys fluent is computational fluid dynamics solver. It works using Navier Stokes. In this paper we comparing engine with different clearance volume 0.75, 1 and 1.25mm. Dynamic meshing is carried out in this analysis.
It is set to a default value of 0. The piston pin offset is generally used to reduce the stress on the reciprocating parts. It enables these parts to be lighter, which increases the efficiency, and decreases the power loss in the engine. It also results in higher rpm.

The simulation as run for 0.75, 1.00mm And 1.25 mm Clearance Volume

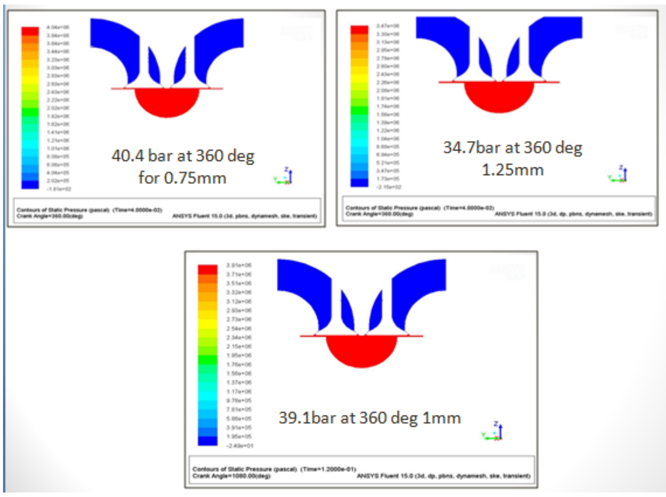


Fig 5.1 contours of Static pressure

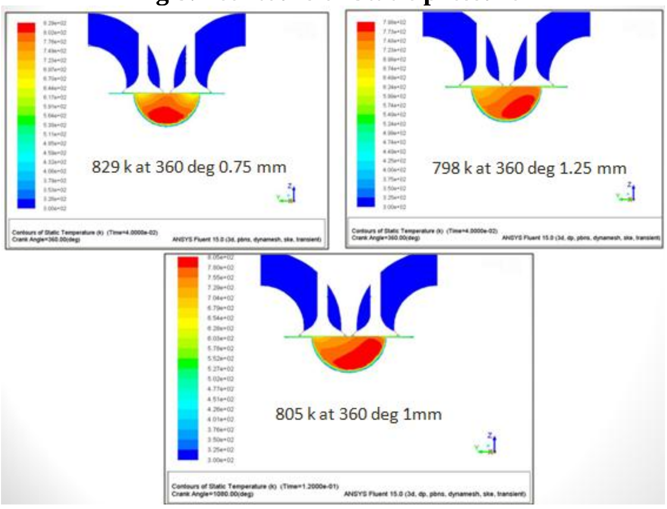


Fig 5.2 contours of Static temperature

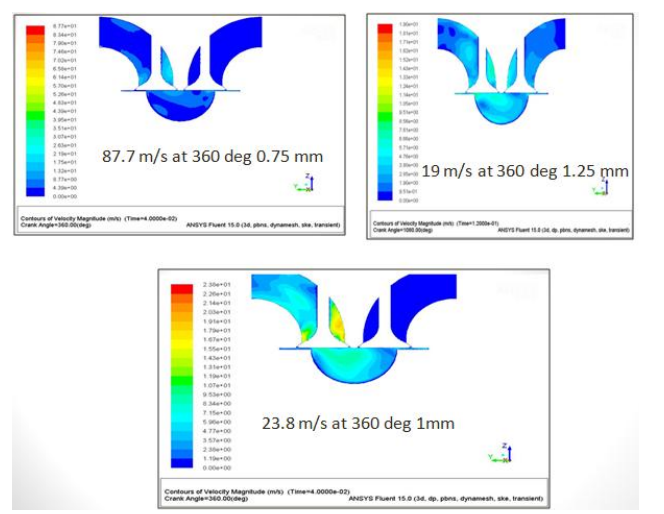


Fig 5.3 contours of velocity magnitude

The Mass Average Static Pressure shows a maximum pressure reading position. This process-cycle follows the exact operation as in the spreadsheet model that makes it easy to compare between the CFD and spreadsheet models. The slower drop in pressure during expansion stroke as compared to the compression stroke is due to the lower rate of heat removal from the cylinder after the energy release stroke in the cylinder that resembles combustion. The Value has been noted 40.4 bar pressure at 360 degree for 0.75 mm, 39.1 bar pressure at 360 degree for 1.00 mm and 34.7 bar pressure at 360 degree for 1.25 mm. as shown in below graph.

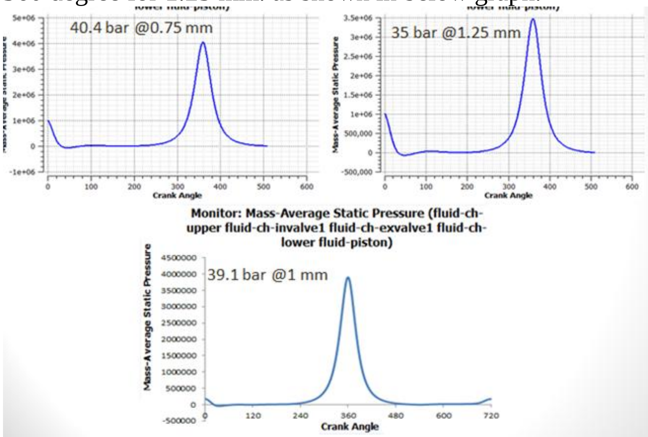


Fig 5.4 Static pressure

The High temperature was found to persist inside the cylinder volume for 360 crank angle Degrees, which might have influenced the slow drop of pressure inside cylinder Volume. However, this high temperature predicted by the cfd model is significantly higher than that predicted by the spreadsheet model, 829 K at 360 degree for 0.75 mm, 805 K at 360 degree for 1.00 mm, and 798 K at 360 degree for 1.25 mm, as shown in below Graph.

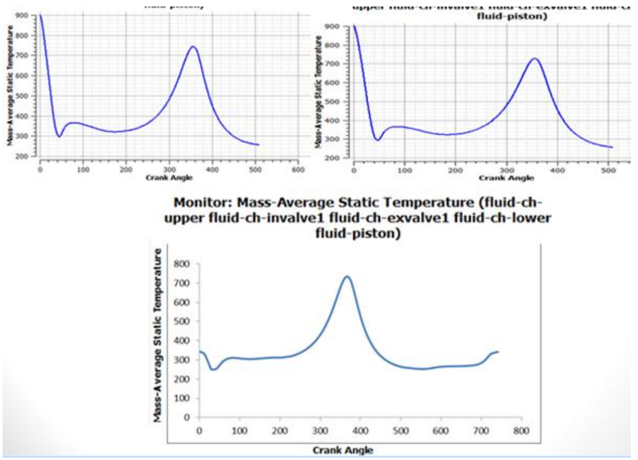


FIG 5.5 Static Temperatures

V. CONCLUSION

- The comparative study clearance volume change in kriloskar engine was studied using cfd.
- From the study 0.75 mm has high pressure and temperature was found, for higher temperature Nox formation is more.
- From 1.25 mm pressure and temperature are lower as compared with 0.75 and 1mm. Due to lower pressure and temperature improper air fuel mixture in cylinder chamber, resulting un burnt fuel and UHBC are present in exhaust.
- As compared with 0.75 and 1.25, 1 mm gives better result and it located in between 0.75 and 1.25mm. This result also matches with experiments.

REFERENCE

1. Wendy and Abdullah, "Numerical analysis of the combustion process in a four-stroke co mpressed 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. DivyanshuPurohit, Pragya Mishra, Vishwanath Banskar, "Flow Simulation of an I.C. Engine in FLUENT, ANSYS 14.0" (ICETMEE-13th-14th March 2014).
6. 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
7. Patil A and Navale LG, "Experimental Verificat ion and CFD Analysis of Single Cy linder Four St rokes C.I. Eng ine Exhaust System " international journal of science, spirituality, business and technology, vol. 3, no. 1, dec 2014
8. Stefania Z, "Numerical analysis of methane direct injection in a single - cylinder250 cm3 spark ignition engine " 69th Conference of the Ita lian Thermal Engineering Association, ATI 2014 Sc ience Direct.
9. 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
10. Internal Combustion Engines in Workbench ANSYS, Inc. Southpointe 275 Technology Drive Canonsburg, PA 15317 ansysinfo@ansys.com http://www.ansys.com (T) 724-746-3304 (F) 724-514-9494

AUTHORS PROFILE

S.Prakash -Research Scholar
Department of Mechanical Engineering,
Vinayaka Mission Research Foundation,
Deemed to be University
Prakash.mech94@gmail.com

Dr. M.Prabhahar - Professor
Department of Mechanical Engineering,
Aarupadai Veedu Institute of Technology
Vinayaka Mission Research Foundation,
Deemed to be University
mprabhahar@gmail.com