Cold Flow Simulation in an IC Engine

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

[1] *M.Tech student, Department of Mechanical Engineering ,VVCE, Karnataka, India*
[2] *Professsor and PG Coordinator, Department of Mechanical Engineering ,VVCE, Karnataka, India*

---***---

Abstract - ***Fluid flow dynamics inside an engine combustion cylinder plays an important role for air-fuel mixture preparation. IC Engine model is developed using CATIAV5R20 tool. The model is then imported to Finite Element preprocessing tool HYPER MESH for the meshing analysis. The model is then imported to Finite Element solver tool. ANSYS FLUENT is used for post processing the results. The flow dynamics inside the cylinder for different minimum valve lift is studied using FEA. Dynamic motion is visualized and velocity magnitude is plotted for different crank angle from 0° to 730°. Finally velocities and crank angles for various valve lifts are compared.***

Key Words: ***Cold Flow Simulation, Flow dynamics, Valve lift***

1. INTRODUCTION

Cold flow analysis involves modeling the airflow and possibly the fuel injection in the transient engine cycle without reactions. The goal is to capture the mixture formation process by accurately accounting for the interaction of moving geometry with the fluid dynamics of the induction process. The changing characteristics of the airflow jet that tumbles into the cylinder with swirl via intake valves and the exhaust jet through the exhaust valves as they open and close can be determined, along with the turbulence production from swirl and tumble due to compression and squish.

Karthikeyan, et al. [1] in their paper describes the simulation of 3D air motion without fuel combustion of In-Cylinder model using the software ANSYS Fluent. Flow dynamics inside engine combustion plays an important role for air-fuel mixture preparation. This enables a better cylinder combustion, efficiency and engine performance. Piston motion, valve opening and closing in the engine model were defined in terms of crank angle. Suction stroke was found to influence air mixing and turbulence in combustion chamber. Compression stroke was found to play a key role in controlling the temperature and pressure of the air mixture in the combustion chamber.

Yogesh et al. [2] in their paper have used Hybrid approach for in cylinder cold flow CFD simulation of four stroke petrol engine using ANSYS fluent. The simulation is carried out using parameter valve lift. Dynamic motion was visualized and velocity magnitude is plotted for crank angle starting from 0° to 720°. The engine is simulated and displays the swirl and tumble zones of fluid and piston layer. Swirl, x-tumble, y-tumble and moment of inertia are written in working directory in text file.

From the literature survey, it is found that solving these cases is time consuming and experimental studies are more expensive than computational studies. Also CFD codes are used for simulation and compared simulated results with experimental results. The efficiency of the engine is the major issue, hence an attempt is made to improve these engines to attain the maximum efficiency.

2. OBJECTIVE

1. To create IC Engine model using CATIA V5R20 tool and solve by ANSYS FLUENT.
2. To study the flow dynamics inside the cylinder for different valve lift using FEA.
3. To compare the valve lifts of 0.1mm and 0.2mm for different velocities.

3. METHODOLOGY

Finite Element analysis was used to determine the characteristics of the IC Engine. For the purpose of this study, the IC Engine model were developed using CATIAV5R20 tool. The model was then imported into Finite Element preprocessing tool HYPER MESH for the meshing analysis. The model was then imported into Finite Element solver tool ANSYS FLUENT for post processing the results.

The steps to carry out the analysis are as follows

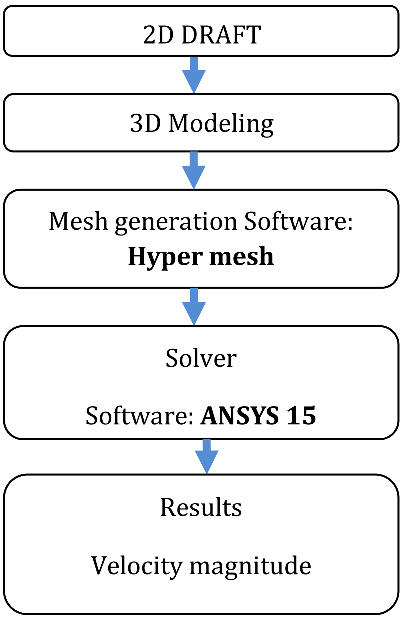

Fig.1 Flow chart of Methodology

CATIA V5R20 software is used to develop the 3D model of the engine parts. Some of the engine parts such as Piston body, Outlet port, Inlet port, Inlet and Exhaust valves are shown in Fig.2 to Fig.5.

In IC Engines the induction and exhaust processes give importance to the efficiency and performance of the engine. In the two stroke engine the flow is regulated by the piston covering and uncovering ports, but in the four stroke engine the induction and exhaust processes are controlled through valves. The four types of valves used are poppet, rotary, sleeve, and disc valves.

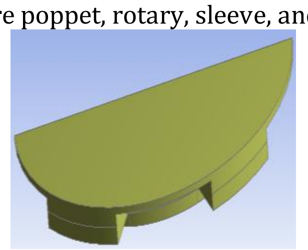
Fig.2 Piston body

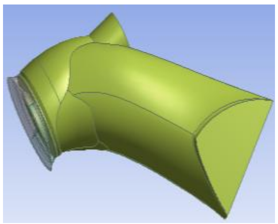
Fig.3 Outlet port

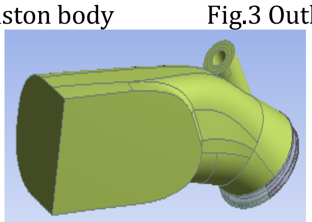
Fig.4 Inlet port

To allow the air enter into the cylinder or the exhaust gases to escape from the cylinder valves are provided known as inlet and exhaust valves respectively are shown in Fig.4. These valves are mounted either in the cylinder head or in the cylinder block. Materials used for the engine valves are forged steel, steel, alloy.

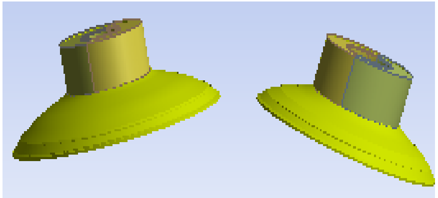
Fig.5 Inlet and exhaust valves

Technical specifications considered for an IC Engine (Diesel Engine) for fluid dynamic analysis purpose [2] is as shown in Table 1.

Table 1: Specifications of Engine model

SN	PARTS	DIMENSIONS
1	Connecting rod length	144.3 mm
2	Crank radius	45 mm
3	Wrench	0
4	Engine speed	2000 rpm
5	Minimum lift	0.1 mm

Parts created by CATIA tool is assembled as shown in Fig.6. The model is prepared with CAD software. The engine is four stroke single cylinder diesel engine with canted valves namely inlet valve and exhaust valve. It is an in-cylinder engine having piston and cylinder in line. Valve seats are provided in both the valves. By using Scheme file it automatically sets up necessary motions for valves and pistons along with solution parameters for the in-cylinder simulation

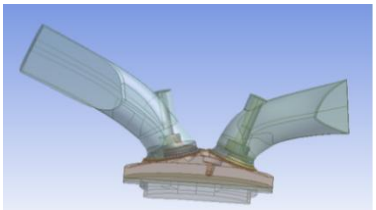
Fig.6 Assembled part of Engine

The next stage is to create a proper "meshing" for the 3D model. "The discritzation of a continuous system with infinite degree of freedom (DOF) to finite degree of freedom (DOF) by nodes and elements is known as meshing". The accuracy of the analysis is purely based on the level of meshing attained by the designer. Hyper mesh tool is used to carry out the meshing as shown in Fig.7.

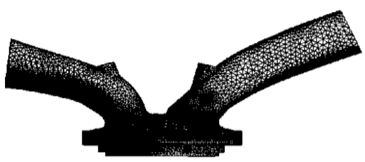
Fig.7 Mesh structure for the geometry

3.1 BOUNDARY CONDITIONS

To calculate the solutions apply boundary conditions and material properties are applied in ANSYS FLUENT. Boundary conditions is as shown in Table 2 and valve lift profile are applied.

Table 2: Boundary Conditions

Type	Zones	Values
Wall (invalve1)	**In valve1-stem, In valve 1-ob, Invalve 1-ch, Invalve 1-ib**	**300 K**
Wall (exvalve1)	**Exvalve1-stem, Exvalve 1-ob, Exvalve 1-ch, Exvalve 1-ib**	**300 K**
Wall (in valve-port)	**Invalve 1-port**	**300 K**
Wall (exvalve-port)	**Exvalve 1-port**	**300 K**

1. 3000 time steps were set for calculation and simulation for every time step it consists of 0.25° of crank angle which will be approximately equal to 720° of crank angle (i.e. 1 complete cycle of four stroke engine).
2. Dynamic motion was visualized and velocity magnitude is plotted for crank angle starting from 0° to 730°.
3. Results of the analysis are plotted as the contours of velocity magnitude at various time steps and crank angle.

3.2 Velocity Contours for valve lift of 0.1mm

Dynamic meshing of the IC Engine fluid computational domain was done and analysis was performed. Then velocity magnitudes for different crank angle were plotted.

Surface grid is generated, as shown in Fig.8 in the consequent analysis, surface grid diagram is avoided and only contours of velocity magnitude diagram is indicated for analysis purpose.

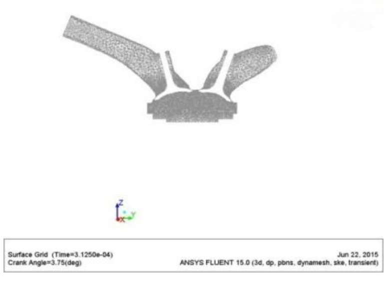

Fig.8 Surface Grid at 3.75°

Fig.9.a shows contours of velocity magnitude. At this stage expansion stroke takes place. Maximum velocity obtained at 3.75° is 1.80m/s.

Fig.9.b shows contours of velocity magnitude at 90°. Maximum velocity obtained at 90° is 11.6m/s.

Fig.9.c shows the contours of velocity magnitude at 182.5°. At this stage exhaust stroke takes place where the piston is at bottom dead centre (BDC). There is a slight opening in the exhaust valve so that compressed air fuel mixture will be exhausted. Maximum velocity obtained at 182.50° is 81.3m/s.

Fig.9.d shows the contours of velocity magnitude at 272.5°. At this stage completely exhaust valve is opened and the air-fuel mixture will be exhausted through it. Maximum velocity obtained at 272.5° is 120m/s.

Fig.9.e shows the contours of velocity magnitude at 361.25°. At this stage it completes half cycle of four stroke diesel engine. Piston will be in TDC position where the exhaust valve will remain closed and the inlet valve will be open i.e., beginning of intake stroke. Maximum velocity obtained at 361.25° is 23.6m/s.

Fig.9.f shows contours of velocity magnitude at crank angle 450°. The interaction of jet with the walls and piston head clearly indicate that there is a significant acceleration considering the abrupt restriction for the passing jet. The turbulence levels seem to grow with the fuel mixture during this stroke. Maximum velocity obtained at 450° is 89.2m/s.

Fig.9.g shows the contours of velocity magnitude at 540 deg. This stage is beginning of compression stroke where both the inlet and exhaust valve will be closed, the piston returns to TDC by compressing the air fuel mixture into the cylinder head. The maximum velocity obtained at 540° is 52.6 m/s.

Fig.9.h shows contours of velocity magnitude at 632.5°. At this stage mixture seems to have attained a uniform mixing process with high turbulence. Maximum velocity obtained at 632.5° 16.9 m/s.

Fig.9.i shows contours of velocity magnitude at 730°. Maximum velocity obtained at 730° is 8.14m/s.

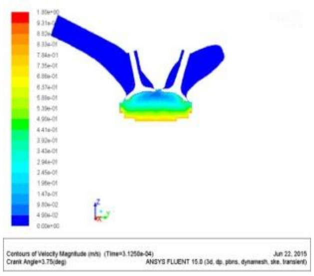

Fig.9.a

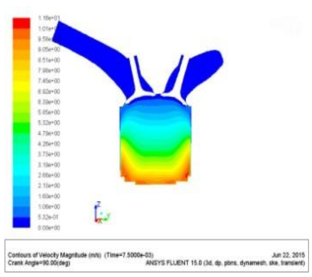

Fig.9.b

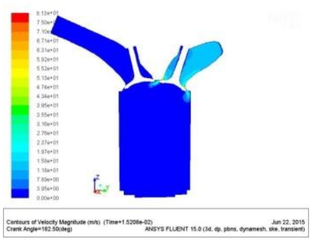

Fig.9.c

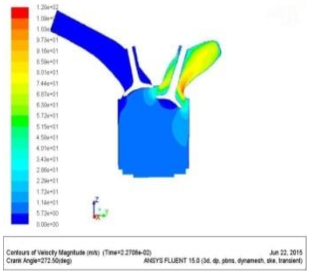

Fig.9.d

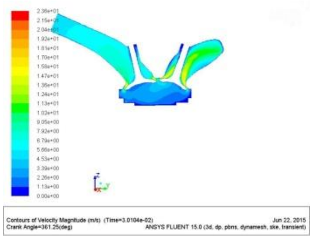

Fig.9.e

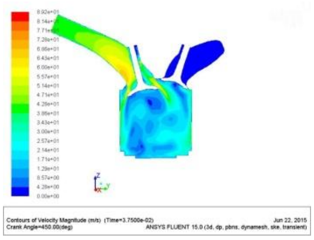

Fig.9.f

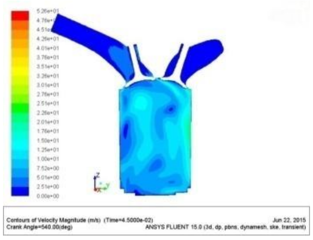

Fig.9.g

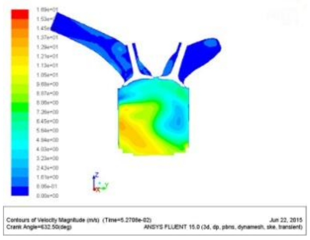

Fig.9.h

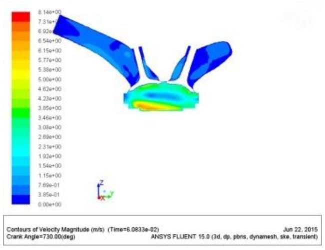

Fig.9.i

Fig. 9 Velocity magnitude for different Crank angles at valve lift of 0.1mm

The plots of velocity magnitude for different crank angle from 0° to 730° at valve lift of 0.1mm are shown in Fig.10. X-axis indicates Crank Angle in "degrees" and Y-axis indicates Velocity Magnitudes in "m/s". Maximum velocity obtained is 120m/s at 272.5°.

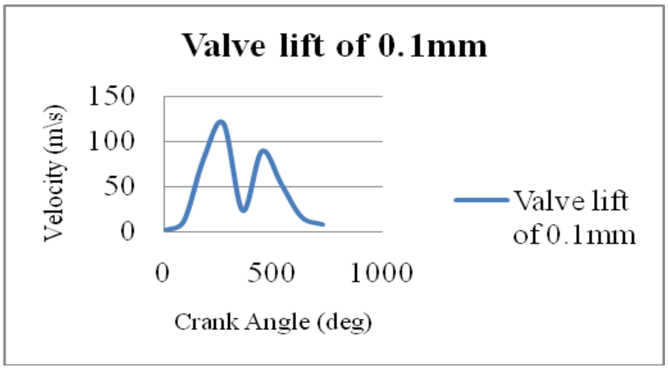

Fig.10 Crank Angle vs. Velocity Magnitude

3.3 Velocity Contours for valve lift of 0.2mm

Valve lift for the same IC Engine model was changed from 0.1mm to 0.2mm. Technical specifications considered for an IC Engine (Diesel Engine) for fluid dynamic analysis purpose for valve lift of 0.2mm is same as shown in Table 1 except for minimum valve lift.

At 3.75° expansion stroke takes place, at this position both the inlet and exhaust valves remains closed. Now the piston is at top dead centre (TDC) and moves to bottom dead centre (BDC) with the increase in volume of cylinder. The maximum velocity obtained at 3.75° is 0.98m/s as shown in Fig.11.a.

Fig.11.b shows the contours of velocity at 90° crank angle. At this stage expansion stroke takes place where the piston is at middle of cylinder i.e., which moves from TDC to BDC. The maximum velocity obtained at 90° is 10.6m/s.

Fig.11.c shows the contours of velocity magnitude at 182.50°. At this stage exhaust stroke takes place where exhaust valve open and creates interface between valve layer and chamber. Also time step is reduced for solution stability. The maximum velocity obtained at 182.50° is 78.9 m/s.

Fig.11.d shows the contours of velocity magnitude at 272.50°. At this stage exhaust valve opens so that the fluid inside the chamber deforms and makes fluid as rigid. The maximum velocity obtained at 272.50° is 114 m/s.

Fig.11.e shows the contours of velocity magnitude at 361.25°. This represents that the end of the exhaust stroke and even completes half cycle of 4-stroke diesel engine. The maximum velocity obtained at 361.25° is 22.6 m/s.

Fig.11.f shows the contours of velocity magnitude at 450°. Air fuel mixture is drawn inwards to form an annular jet. The spread and mix of the jet can also be noticed. The maximum velocity obtained at 450° is 85.9 m/s.

Fig.11.g shows the contours of velocity magnitude at 540°. At this stage both the inlet and exhaust valve is closed. The maximum velocity obtained at 540° is 50.1 m/s.

Fig.11.h shows the contours of velocity magnitude at 632.50°. This represents start of compression stroke. The piston returns to the top of the cylinder compressing the air/fuel mixture into the cylinder head. The maximum velocity obtained at 632.50° is 16.1 m/s.

Fig.11.i shows the contours of velocity magnitude at 730°. This represents the end of compression stroke and also ends of one full cycle of four stroke diesel engine. The maximum velocity obtained at 730° is 7.69 m/s.

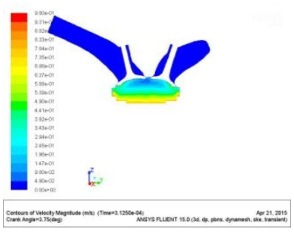

Fig.11.a

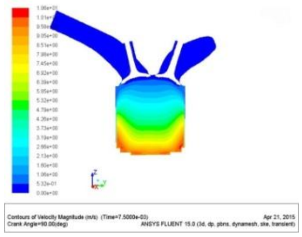

Fig.11.b

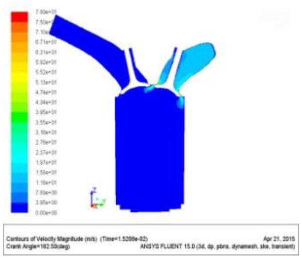

Fig.11.c

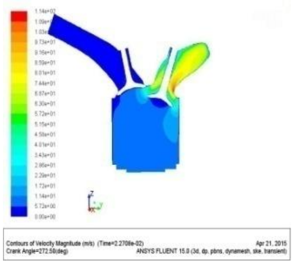

Fig.11.d

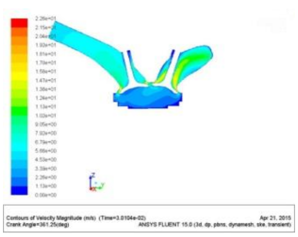

Fig.11.e

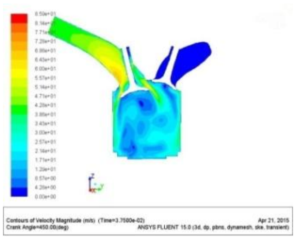

Fig.11.f

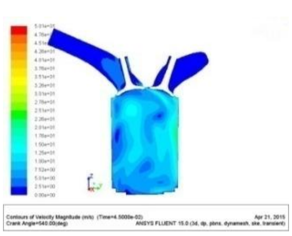

Fig.11.g

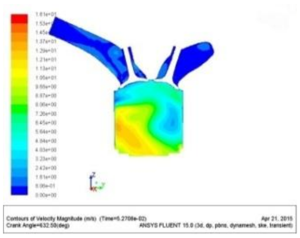

Fig.11.h

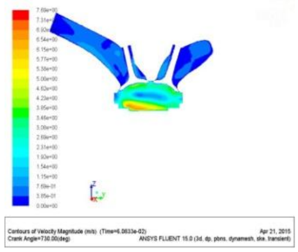

Fig.11.i

Fig. 11 Velocity magnitude for different Crank angles at valve lift of 0.2mm

The plots of velocity magnitude for different crank angle from 0° to 730° at valve lift of 0.2mm is shown in Fig.12. X-axis indicates Crank Angle in "degrees" and Y-axis indicates Velocity Magnitudes in "m/s". Maximum velocity obtained is 114m/s at 272.5°.

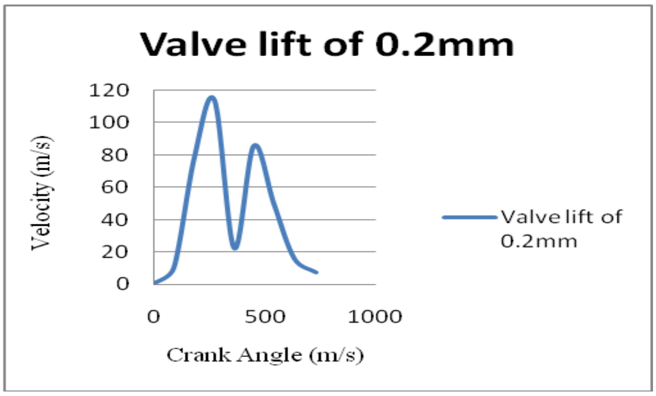

Fig.12 Crank Angle vs. Velocity Magnitude

3.4 Comparison for valve lifts of 0.1mm, 0.2mm

The plots of velocity magnitude for various crank angles at different valve lifts are compared in the following Fig.13. X-axis indicates Crank Angle in "degrees" and Y-axis indicates Velocity Magnitudes in "m/s". For first half cycle 270° is the highest peak and for second half cycle 450° is the highest peak i.e., exhaust and intake stroke has the maximum influence on the air mixing and turbulence in combustion chamber. Fig.13 indicates that as valve lift increases the velocity decreases.

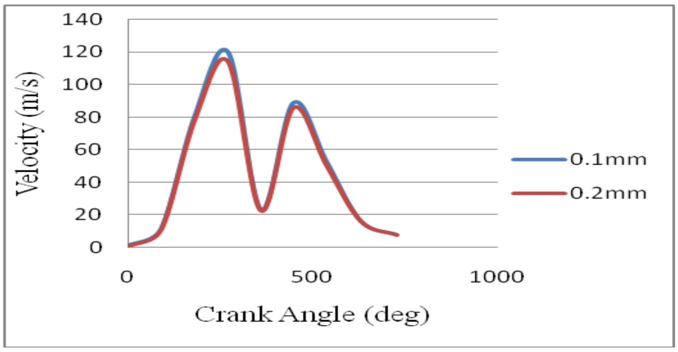

Fig 13 Comparison of Valve lifts

4. CONCLUSION

All four strokes and their effect on in cylinder air motion are studied effectively and following conclusions are obtained:

1. Dynamic motion is visualized and velocity magnitude is plotted for different crank angle from 0° to 730°.
2. When the valve lifts increases velocity obtained decreases.
3. Exhaust stroke has the maximum influence on the air mixing and turbulence in the combustion chamber.
4. CFD can be used as a useful tool to fix the parameters related to engine performance.

REFERENCES

[1] Karthikeyan CP, Lakshman A and Davidson Jebaseelan, "CFD studies on In-cylinder air motion during different strokes of an IC Engine", SET Conference, VIT University Chennai, (2012).
[2] Pathak Yogesh R, Deore Kailas D and Patil Vijayendra M "In cylinder cold flow CFD simulation of IC Engine using Hybrid Approach" 2321-7308.
[3] Shahrir Abdullah and Azhari Shemsudeen "A computational fluid dynamics study of cold flow analysis for mixture preparation in a motored four stroke DI Engine", journals of applied science, 2710-2724, (2004).
[4] Nureddin Dinler, and Nuri Yucel, "Numerical Simulation of Flow and Combustion in an Axisymmetric Internal Combustion Engine", World Academy of Science, Engineering and Technology 36 (2007).
[5]]ANSYS user's guide, ANSYS, Inc. Southpointe, Canonsburg, PA 15317, (2012).

BIOGRAPHIES

[1]**Rohith. S** completed his B.E.in Mechanical Engineering from Vidyavardhaka College of Engineering in 2013. He is now pursuing M.Tech in Machine Design at Vidyavardhaka college of Engineering, Mysuru.

[2]**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.