Modification in intake Manifold

Shridhar Jadhav[1], Anand Devkar[2], Prof.P.P. Kharche[3]
Dipak Bhise[4], Akshay Daberao[5]
Department of Mechanical Engineering, SavitribaiPhule Pune University, Pune, Maharashtra
[3]Assistant Professor at Smt. KashibaiNavale College of Engineering, Pune , Maharashtra ,India
[1245]Student from Mechanical Department of Smt. KashibaiNavaleCollege of Engineering, Pune , Maharashtra ,India

Abstract -For proper mixing of fuel and gas, there is a need for hovering in the AI engine. Using Virtual Bench "CFD" The value of the engine consumption port's designs has been significantly revised which reduces the use of the best time. In this case, direct injection is being done using Computational Action Diesel Engine Seven Port Fluent.The CFD calculation is done to simulate the test rig "flow bench" work. Engine with two ports is considered to be designed to study the movement of gas flow. To analyze the air flow movements in the diesel engine, the 3D model of the flow area was calculated. It has been calculated on various valve lifts to calculate the speed of air flow in different parts of the engine. Two set of engine geometry is said to be a traditional valve and the other with a skirt is the valve. It has been comparatively analyzed to fill the cylinders well. This analysis was made to develop standard methods for future revisions.

Key Words: Turbulence, Flow, Fluent, Pressure, Skirt, Intake manifold

1. INTRODUCTION

Internal combustion engines are seen every day in automobiles, trucks, and buses. The name internal combustion refers also to gas turbines except that the name is usually applied to reciprocating internal combustion (I.C.) engines like the ones found in everyday automobiles. There are basically two types of I.C. ignition engines, those which need a spark plug, and those that rely on compression of a fluid.In compression ignition engines the mixture ignites at high temperatures and pressures. The performance of an IC Engine depends upon complex interactions between mechanical, fluid, chemical, and electronic systems. However, the central challenge in design is the complex fluid dynamics of turbulent reacting flows with moving parts through the intake/exhaust manifolds, valves, cylinder, and piston. The time scales of the intake air flow, fuel injection, liquid vaporization, turbulent mixing, species transport, chemistry, and pollutant formation all overlap, and need to be considered simultaneously.

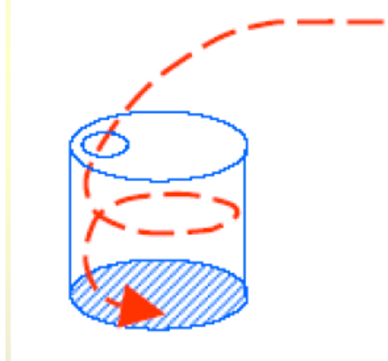
2. LITERATURE REVIEW

Abdul Rahim Ismail, Rosli Abu Bakar and Seminet.al. studied the effect of discharge coefficient of intake andExhaust valve for direct injection engine, the discharge coefficient has been successfully calculated theoretically and its value varies depending to test pressure and valve lift to diameter ratio.

T. Lucchini, G. D'Errico, D. Ettorre, E. Spagnoli, G. Ferrari et.al.have discussed Development and application of numerical models (Lib-ICE) to simulate in-cylinder flows and combustion in IC engines using the OpenFOAM to improve the existing combustion models to provide advanced diagnostic and development tools to design and simulate Diesel engines.

Bassem Ramadanet.al.conducted a computational study of Direct Injection (DI) engine intake system using KIVA-3V. The effect of adding shrouds to intake valve is studied

3. SWIRL MOTION IN IC ENGINE



Turbulence is necessary to break the frame front into pieces so that each and every part of combustion chamber gets flame to ignite homogeneous air fuel mixture. If there is uneven flame distribution then there is incomplete combustion which gives less torque and also causes pollution.

In case of CI engines fuel is injected in highly compressed air. There is always possibility that the portion of air near the injector will get more fuel and thus again farthest point from injector will get less fuel supply. This again gives rise to unequal fuel distribution and incomplete combustion.

For generating swirl there are different methods for example skirting and shrouding. Shrouding and skirting are done to improve charge motion by doing changes in the valve of the engine. In this case we have studied the changes in charge motion when skirting is added to the valve and swirl generation is tested.

4. USE OF FLUENT

IC engines involve complex fluid dynamic interactions between air flow, fuel injection, moving geometries, and combustion. Fluid dynamics phenomena like jet formation, wall impingement with swirl and tumble, and turbulence production are critical for high efficiency engine performance and meeting emissions criteria. The design problems that are encountered include port-flow design, combustion chamber shape design, variable valve timing, injection and ignition timing, and design for low or idle speeds.

Several techniques and sub-models are used for modeling moving geometry motion and its effect on fluid flow. Using CFD results, the flow phenomena can be visualized on 3D geometry and analyzed numerically, providing tremendous insight into the complex interactions that occur inside the engine. This allows you to compare different designs and quantify the trade-offs such as soot vsNOx, swirl vs tumble and impact on turbulence production, combustion efficiency vs pollutant formation, which helps determine optimal designs. Hence CFD analysis is used extensively as part of the design process in automotive engineering, power generation, and transportation.With the rise of modern and inexpensive computing power and 3D CAD systems, it has become much easier for analysts to perform CFD analysis

5.AIR FLOW MOTION

The flow coefficient (α_k) is defined as the ratio of the actual or measured mass flow rate at standard condition and the theoretical mass flow rate. Cylinder bore diameter is used as characteristic length for calculating the theoretical mass flow rate.

$$\alpha_k = \frac{m_{std}}{m_{theor}}$$

The actual or measured mass flow rate at standard condition is calculated using the following expression.

$$m_{std} = V \times \frac{P_{std}}{T_{std} \times R}$$

The theoretical mass flow rate is calculated using the following expression

$$m_{theor} = A \times \rho_s \times C_s$$

Piston area is calculated from piston bore diameter

$$\rho_s = \frac{P_1}{T_{std} \times R}\left[\frac{P_2}{P_1}\right]^{1/k}$$

Flow velocity is calculated for isentropic flow

$$C_s = \sqrt{\frac{2k}{k-1} \times R \times T_{std} \times \left[1 - \left(\frac{P_2}{P_1}\right)^{1/k}\right]}$$

$$P_1 = P_2 - \Delta P$$

6. TURBULENCE

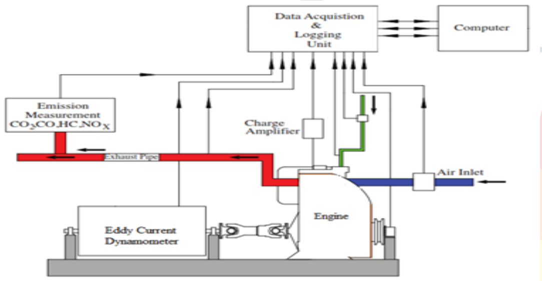
In laminar flow the adjacent layers slide past each other in an orderly fashion. Turbulent flow on the other hand is random and chaotic in its nature. There are six main characteristics for turbulent flow:

- Irregular

- Diffusive
- Has a relatively high Reynolds number
- Three-dimensional
- Dissipative
- Property of the flow (i.e. not the fluid)

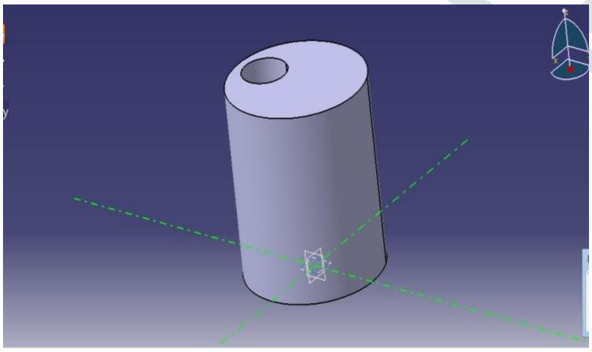
7. EXPERIMENT DETAILS

In this experiment two sets of engines are taken into account. In one case the swirl motion in conventional valve engine is done and in another case swirl motion is studied by adding skirting to the valve. Skirting acts as barrier to the flow and hence causes more swirling.



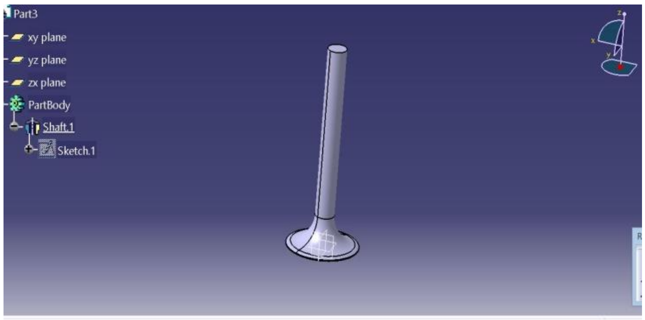
8. MODELING OF INTAKE MANIFOLD IN CATIA

8.1 CYLINDER

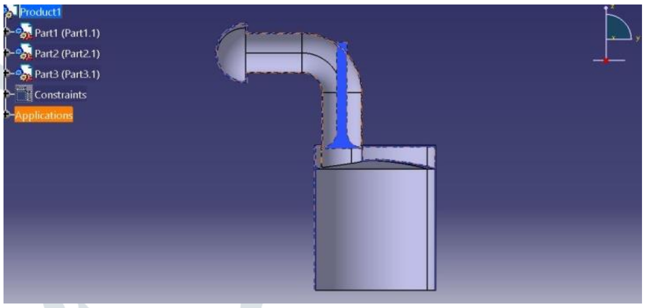


8.2 INLET PIPE

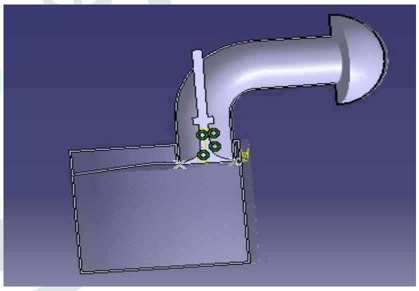
8.3 SWELL VALVE



8.4 CYLINDER WITH CONVENTIONAL VALVE



8.5 CYLINDER WITH MODIFIED VALVE

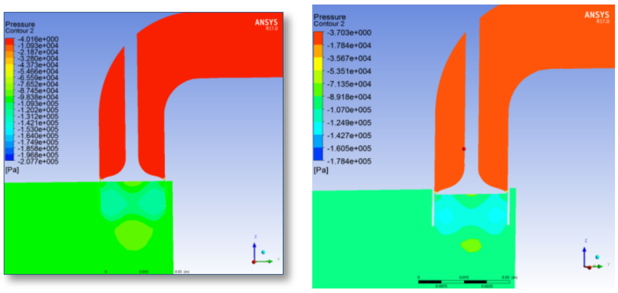


9. Results and Discussion

9.1 Air Flow Calculations

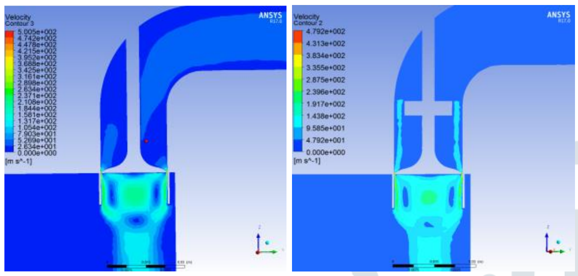
Steady state air flow calculations are performed for three different intake valve lifts viz. low lift 0.02mm and high lift 1.373mm to investigate the flow features. Sufficient mesh refinement has been provided near the throat area because the flow velocity changes rapidly in this region and capturing the gradients are keys for an accurate simulation.

9.2Pressure contours



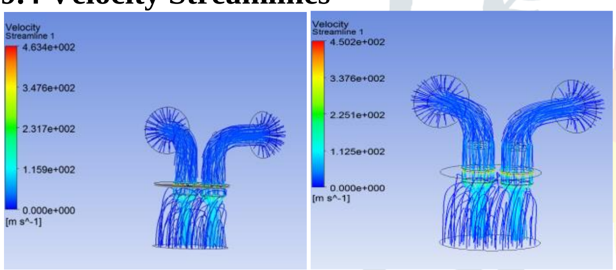
Pressure contours for conventional valve and modified Valve respectively.

9.3 Velocity Contour



Velocity contours for conventional valve and modified Valve respectively.

9.4 Velocity Streamlines



Velocity Streamlines for conventional valve and modified valve respectively.

10.Conclusion

For better combustion of the fuel in engine, air fuel mixing plays an important role. Velocity of air should be more near valve for proper mixing of the air and fuel mixture, velocity of air in liner part is very important. Higher will be the velocity in that region more turbulence will be created. Which will lead to better swirl motion. More turbulence also ensures better air and fuel mixture.

11. References

1. Bassem Ramadan, "Study of swirl generation in KIVA-3V",Kettering University.
2. Laxmikant P. Narkhede&AtulPatil,"Optimization For Intake Port, International Journal Of Mechanical And Production Engineering Research And Development (Ijmperd), 2014"
3. T. Lucchini, G. D'Errico, D. Ettorre, E. Spagnoli, G. Ferrari A "C++ object-oriented library for internal combustion engine simulations: spray and combustion modeling, Internal Combustion Engine Group", Department of Energy Politecnico di Milano, 2010
4. Ervin Adorean, Gheorghe-AlexandruRadu, "Diesel engine in-cylinder calculations with openfoam."
5. T. Lucchini, G. Montenegro, G. D'Errico"CFD Simulations of I.C. Engines: Combustion, Internal Flows, integrated 1D-MultiD simulations" 2007
6. T. Lucchini, "Engine Simulation with Piston and Valve Action in OpenFOAM", 2011
7. Abdul Rahim Ismail, Rosli Abu Bakar and Semin, "The effect of discharge coefficient of intake and exhaust valve for direct injection engine", ICME07-TH-17, 2007

BIOGRAPHIES

Prof.P.P.Kharche
Assistant Professor at Smt. KashibaiNavale College of Engineering, Pune, 411041

ShridharJadhav
Student from Mechanical Department of Smt. KashibaiNavale College of Engineering, Pune, 411041

AnandDevkar
Student from Mechanical Department of Smt. KashibaiNavale College of Engineering, Pune, 411041

AkshayDaberao
Student from Mechanical Department of Smt. KashibaiNavale College of Engineering, Pune, 411041

DipakBhise
Student from Mechanical Department of Smt. KashibaiNavale College of Engineering, Pune,, 411041