International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 04 Issue: 12 | Dec-2017 www.irjet.net p-ISSN: 2395-0072

A Comparative CFD Analysis for Air Swirl on Conventional Valve and Modified Valve with Skirting of Diesel Engine

Nilesh Kakad[1], Prof. S.B. Bawaskar[2], Prof. Ratnadip Bhorge[3]

[1]Student, Dept. of Mechanical Engineering, SVCET Rajuri, Maharshtra, India
[2]Faculty, Dept. of Mechanical Engineering, SVCET Rajuri, Maharshtra, India
[3]Faculty, Dept. of Automobile Engineering, DPCOE Pune, Maharshtra, India

---***---

Abstract - *In diesel engines swirl is needed for proper mixing of fuel and air. Evaluation of design of engine intake port using virtual bench "CFD" is significantly improved by identifying best practices which reduces the time consumption. In DI engines swirl is needed for proper mixing of fuel and air. Evaluation of design of engine intake port using virtual bench "CFD" is significantly improved by identifying best practices which reduces the time consumption. The engine with two ports is considered which is designed to study air flow motion. 3D model of the flow field were computed to analyze air flow motion in the considered diesel engine. Calculation was carried out at different valve lifts to calculate air flow velocity inside different sections of engine. Two sets of engine geometries are considered one is with conventional valve and another is with valve with skirting. The comparative analysis was done to determine better swirl in cylinder .The analysis was done to develop standard methodology for future iteration.*

Key Words: Swirl, fluent, Skirting, IC engine.

1. INTRODUCTION

Internal combustion engines are seen every day in automobiles, trucks, and buses. 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. Compression ignition engines take atmospheric air and fuel, compress it to high pressure and temperature, at which time combustion occurs. These engines are high in power and fuel economy. Engines are also divided into four stroke and two stroke engines. In four stroke engines the piston accomplishes four distinct strokes for every two revolutions of the crankshaft. Combustion in diesel engines depends upon the injection process and fuel jet interaction with the air inside the cylinder. Knowledge of the air movement inside the cylinder is of great importance for the improvement of the combustion process. In DI engines swirl is needed for proper mixing of fuel and air. Moreover, the efficiency of engine can be improved by increasing burn rate of fuel/air mixture. Swirl and fuel motion can have significant effect on fuel air mixing, combustion, heat transfer, and emissions. Turbulence inside the cylinder is high during the intake and then decreases as the flow rate slows near the bottom dead centre (BDC). It increases again during the compression stroke as swirl, squish and tumble increase near the top dead centre (TDC). The high turbulence near TDC when ignition occurs is much required for combustion process of the engine. It breaks up and spreads the flame front many times faster than that of a laminar flame. Both fuel and air is consumed in a very short time, whereby self-ignition and knock are avoided. This turbulence is enhanced by the expansion of engine cylinder during the combustion process.

2. PROBLEM DEFINATION

The problems associated with conventional test rig flow bench are the time and cost consumed by them. Design engineers should work a lot on the design and construction of the model and the errors involved in test rig affect the results.

3. METHODOLOGY

1. Creating geometry using using ANSYS Design Modeler or import geometry from any of the CAD Software.
2. Creating fluid body (Flow model)
3. Mesh the model.
4. Selecting material.
5. Defining zones.
6. Importing in ANSYS fluent.
7. Defining boundary conditions.
8. Obtaining result.

4. 3D MODEL

In this project 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.

Table -1: Valve design specification

Parameter	Dimensions(mm)
Base diameter of valve	26.3
Stem length of valve	67
Diameter of stem	7
Diameter of pipe	27.3

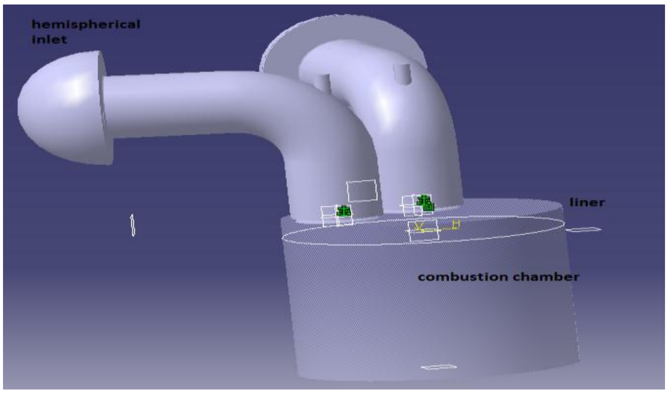

Fig -1: Engine geometry after assembly

5. ANALYSIS

Flow bench testing does not provide a very efficient path to the final design because the designers do not have an insight on the details of recirculation areas, turbulence and design-imposed pressure losses. CFD (Virtual Flow Bench), an analysis tool used to improve the port design by simulating the flow in alternative port designs. CFD simulation provides fluid velocity and pressure throughout the solution domain with complex geometries and boundary conditions.

So the analysis is done using ANSYS fluent. 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. The engine used for simulation is stationary engine with bore diameter 79.5mm and stroke length 95.5mm. The RPM of engine used is 2000.

The geometric details of considered engine are given in Table 2 below

Table -2: Geometrical specifications of engine

Sr. No.	Parameters	Dimensions
1	Bore	79.5mm
2	Stroke	95.5mm
3	Squish	1mm
4	Engine RPM	2000
5	Intake valve opening	4 degrees ATDC
6	Intake valve closing	25 degrees ABDC

Table -3: Input values for air flow calculations

2	Parameters	Value
1	Area at inlet	0.00052209 m^2
2	Density	1.225 kg/m^3
3	Mass flow	0.006395603 kg/s
4	Factor	1.5
5	Turbulent intensity	0.05
6	Epsilon turbulent Dissipation rate	0.09
7	Kinetic energy	0.375
8	Epsilon turbulent dissipation rate	0.147625498

Giving the above inputs values to ANSYS fluent and post processing is done.

5. RESULT AND DISCUSSION

Case I: Considering Pressure contour

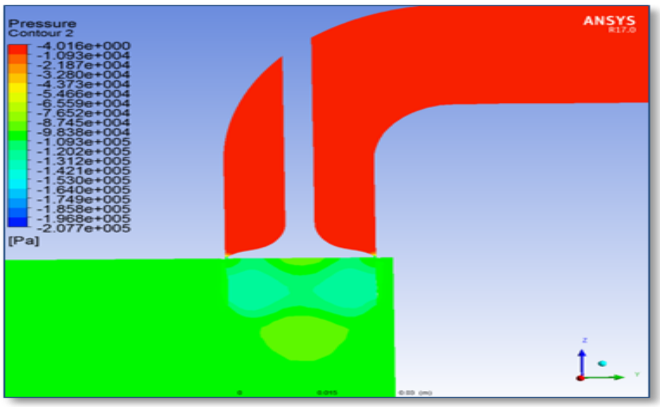

Fig -2: Pressure contour for conventional valve with minimum valve lift

By observing the contour in Fig 2 we can find that the pressure is higher in the pipe side and it reduces when air enters the cylinder.

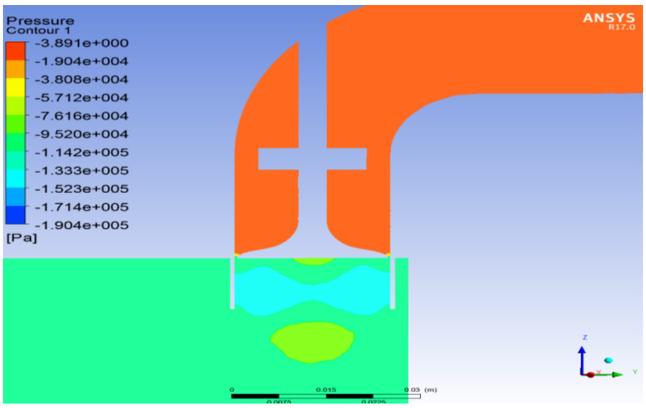

Fig -3: Pressure contour for modified valve

By observing the contour fig 3 we can find that the pressure is higher in the pipe side and it reduces when air enters the cylinder.

Case II: Considering velocity contour

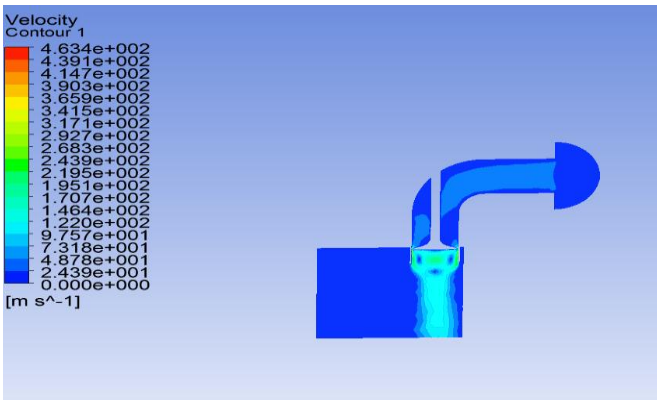

Fig -4: Velocity contour for conventional valve

By observing the contour in fig 3 we can find that the velocity is higher near valve side and it reduces when air enters the cylinder.

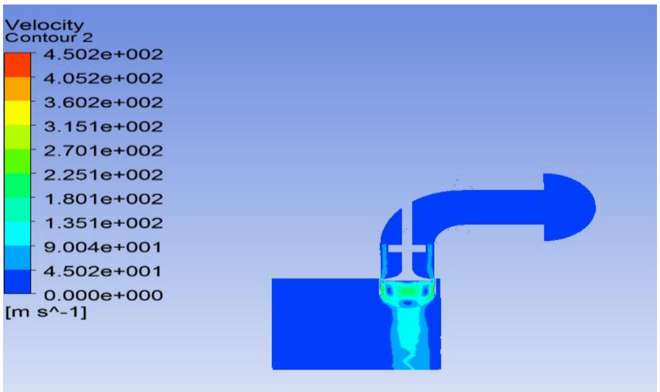

Fig -5: Velocity contour for modified valve

By observing the contour fig 5 we can find that the velocity is higher near the valve side and it reduces when air enters the cylinder. It is also observed that near the skirting velocity is slightly increased.

6. CONCLUSIONS

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.

REFERENCES

[1] Laxmikant P. Narkhede & AtulPatil, Optimization For Intake Port, International Journal Of Mechanical And Production Engineering Research And Development (Ijmperd), 2014

[2] Bassem Ramadan, study of swirl generation in KIVA-3V, kettering university.

[3] T. Lucchini, G. Montenegro, G. D'Errico CFD Simulations of I.C. Engines: Combustion, Internal Flows, integrated 1D-MultiD simulations 2007

[4] Klas Fridolin, Cfd For Air Induction Systems With Openfoam Chalmers University Of Technology Gothenburg, Sweden 2012

[5] Jorge Martins, Senhorinha Teixeira, Stijn Coene, Design Of An Inlet Track Of A Small I. C. Engine For Swirl Enhancement, 20th International Congress of Mechanical Engineering, 2009

[6] H.Mohamed Niyaz, Prof. A.S.Dhekane, Design Optimization Of Intake Port In Diesel Engine By Using CFD Analysis, International Journal Of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 2 Issue 11, November – 2013

[7] Ervin Adorean, Gheorghe-AlexandruRadu, diesel engine in-cylinder calculations with openfoam.