Design Optimization of Intake Port in Diesel Engine By Using CFD Analysis

H.Mohamed Niyaz
Dy.Patil college of engineering, Akurdi, Pune-411044

Prof. A.S.Dhekane
Dy.Patil college of engineering, Akurdi, Pune-411044

Abstract

Design optimization of intake port in diesel engine is to be done by using computation fluid dynamics (CFD) analysis. The aim of the task is to achieve optimum mass flow rate inside the combustion chamber that is expected to increase the volumetric efficiency of the diesel engine. Intake port design evaluation is carried out in virtual flow bench "CFD" using the best available turbulence model available to reduce the time for product analysis. Compressible steady state CFD calculation was performed to match the conventional test rig flow bench including pressure losses due to design. The calculation was carried out at different valve lifts with increased valve seat diameter to obtain maximum air flow rate, mass flow as well as the discharge coefficients. CFD and conventional test rig results were compared and for its accuracy methodology has been developed for future iterations. Calculated flow coefficients and swirl ratio shows good agreement with the experiments.

Keywords: Swirl Ratio, CFD, Flow coefficients, Flow bench.

1. Introduction

Optimization of the inflow through intake ports is very important factor of development of an internal combustion engines. So that the charge movement generated by the intake flow considerably influence the volumetric efficient of the diesel engine. Swirl and tumble motion during the intake stroke determine the nature of the swirling flow in an engine, so CFD simulations and steady state flow tests are often used to characterize the swirl and discharge coefficient. This will helps to optimize the geometrical shape of the intake port or valve that produces the optimum efficiency.

Flow bench is one way of measuring fluid motion of the intake port. The major problem of flow bench is time consuming process and the designer has to do so many iteration to finalise the optimum design even with some compromise factor. Another major problem of the flow bench is that the cost involvement in building new design renders.

Even with too much time and cost involvement the flow bench testing does not provide a very efficient way to the optimum design because the flow bench does not provide in cylinder details of recirculation area, turbulence and pressure loses.

So the best way of analyse the intake port is to simulate the actual flow condition in the computational flow dynamics (CFD). By this way CFD simulation provides the fluid velocity and pressure throughout the solution domain with complex geometries and boundary conditions. CFD is the smart approach design modifications and boundary conditions on the alternate port design. So with this way the amount of experimentation required to develop a new design which ultimately reduces the time and money investment.

In conventional method it takes much time to make prototype of intake port and run the test that gives no information on the internal flow patterns. But the CFD virtual flow simulation makes it possible to analyse a new design in less time and CFD can also able to give complete information on the flow velocity as well as pressure throughout the computational intake port.

2. Flow chart

CFD calculation and experimental measurement process flows are shown in the Fig.1.

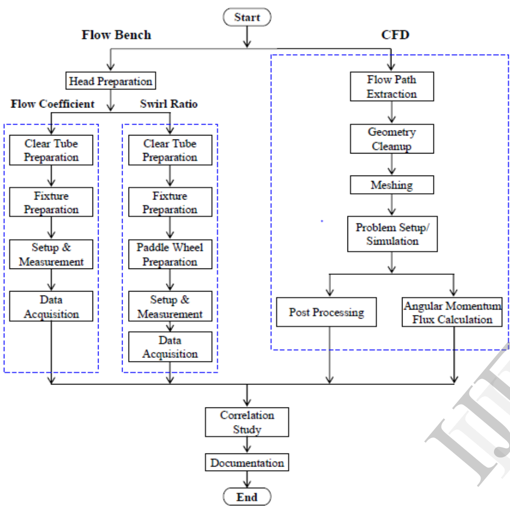


Figure 1. Work flow chart

3. Correlation between CFD & flow bench

3.1. Flow bench

The experimental setup shown in Fig.2. for the flow bench measurement that used to perform the steady state testing measurement for flow coefficient. Bell Shape mouth is used to reduce the entry pressure losses. The valve lifts are adjusted by the calibrated dial gauge. Vacuum pressure is provided by the compressor unit in the cylinder. Pressure sensor is used to monitor both the system pressure and the atmospheric pressure.

Compensation tank with pressure sensor is for the system pressure measurement at bottom of the cylinder tube. the dynamic head is recovered by this compensating tank by maintaining system pressure equal the stagnation pressure. Temperature sensor is placed in this setup to measure the atmospheric pressure. Air flow sensor also used to monitor the flow rate across the intake port.

Figure 2. Experimental set up for flow coefficient measurement

Swirl rate is measured by the paddle wheel setup shown in Fig.3. cylinder tube contains the paddle wheel and speed sensor which is used to calculate circumferential velocity of the paddle wheel.

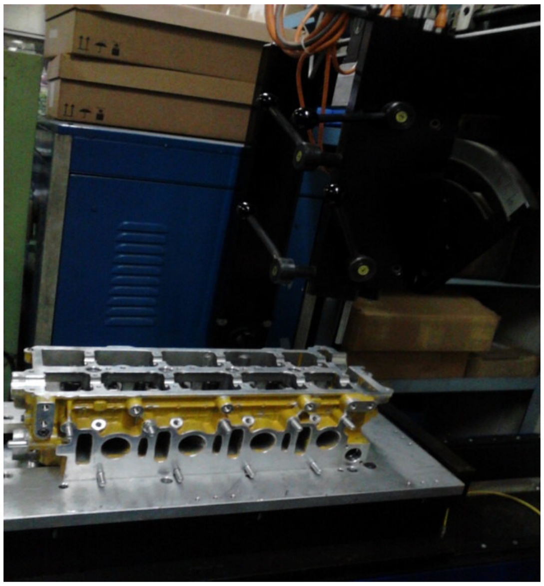


Figure 3. Experimental set up for swirl measurement

3.2. CFD model

3.2.1. Air flow model

The intake system CAD model consists of intake port, valve stem, valve, valve seat, combustion chamber. Bell shaped hemispherical inlet is used to create the atmosphere without any entrance loss and the bottom of the cylinder is BDC of the piston in actual engine. Fig.4. shows the computational model for the CFD analysis.

3.2.2. Charge motion model

Swirl monitoring plane placed at the bottom of the cylinder where the swirl ratio and flow coefficient are measured. This 2D interior plane replicates the paddle wheel in the flow bench setup. Compensation tank is not considered in the CFD analysis, because static pressure at the outlet boundary condition can take care of the stagnation pressure. Fig.5. shows the location of the swirl monitoring plane that used to manipulate the angular moment flux in the CFD analysis.

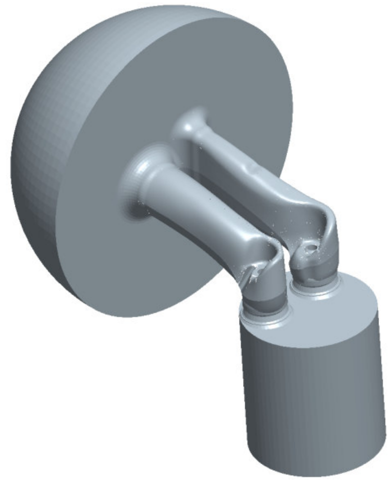


Figure 4. 3D cad model of intake port

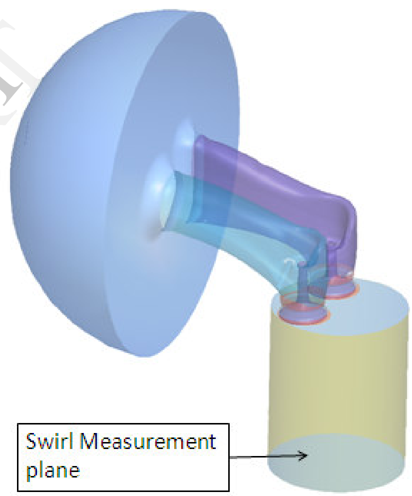


Figure 5. Intake port computational model

4. Mesh generation

Mesh creation is the most important and time consuming process in the CFD analysis Fig.6. shows the unstructured tetrahedral elements that are used to discretize the flow medium. Fig.7. shows the cross section view of the intake port mesh geometry. Meshes are created such a way that to capture the air velocity stream coming out through the throat nozzle

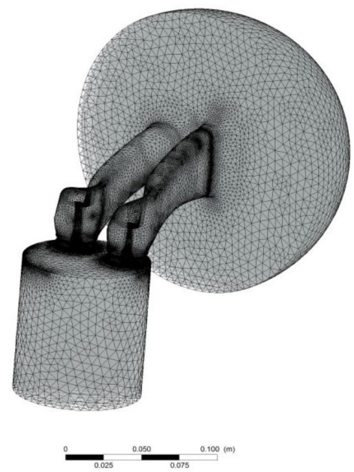


Figure 6. Mesh creation of intake port

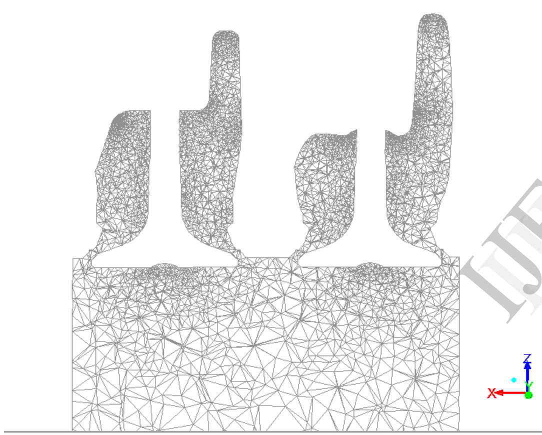


Fig.7. Valve and throat inflation

5. Calculation parameters

5.1. Flow coefficient

The flow coefficient (αk) 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.

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

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

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

The theoretical mass flow rate is calculated using the following expression.

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

Piston area is calculated from piston bore diameter

$$\rho_s = \frac{P_1}{R * T_{std}} [\frac{P_2}{P_1}]^{\frac{1}{k}}$$

Flow velocity is calculated for isentropic flow

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

$$P_1 = 1\ bar$$

$$P_2 = P_1 - \Delta P$$

Gauge static pressure $P2$ measured in the compensating tank can be considered as stagnation pressure for ΔP calculation as the dynamic component is fully recovered.

5.2. Swirl ratio

Swirl ratio is defined as the ratio of circumferential air speed in the cylinder to the axial speed of the air flow in the cylinder

$$SR = \frac{Circumferential\ velocity(C_u)}{Axial\ velocity(C_A)}$$

Circumferential Air velocity in the cylinder is calculated by using

$$C_A = \frac{V_{real}}{D_{cyl}^2 * \frac{\pi}{4}}$$

Theoretical volumetric flow rate is calculated using orifice meter equation as,

$$V_{real} = V * \sqrt{\frac{\rho_{std}}{\rho_{real}}} = V * \sqrt{\frac{P_{std} * T_{amb}}{T_{std} * P_{amb}}}$$

$$P_{std} = 100000\ N/m^2$$

$$T_{std} = 288.7\ K$$

6. Calculation method

6.1. Experimental measurement

Flow bench is the experimental way of evaluating steady state flow of air in the cylinder head. Constant pressure difference is applied between the cylinder tube and the atmosphere. Flow bench controller will control the test pressure in the system. It also records the flow rate, system pressure and temperature. Rotational speed of the paddle wheel will gives ratio of swirl. Discharge coefficient and swirl ratio are tabulated in table No.1. from the experimental setup.

Table 1. Experimental flow measurement

Lift (mm)	Flow coefficient	Swirl Ratio
1	0.14	0.20
2	0.34	1.08
3	0.48	1.18
4	0.56	1.43
5	0.70	1.65
6	0.77	1.76
7	0.79	1.71
8	0.83	1.66

6.2. Virtual flow calculation

Steady state virtual flow calculation is carried out for the different intake valve lift to investigate the flow properties. Adequate mesh refinement has been created near throat area to take care of the rapid flow velocity changes in this region and adapting the velocity and pressure gradients is key for the simulation accuracy.

6.2.1. Governing equation

Navier stroke equation for mass , momentum and energy is the general to solve the CFD problems. Steady state, κ–ε model is used to capture the flow involving rotational. To represent the flow near wall, no slip and stationary wall function is selected for the wall treatment. Walls are the main source of the mean velocity and turbulence. So non-equilibrium wall function is used to represent the wall bounded turbulent flows.

6.2.2. Solver

Pressure based segregated solver is used to solve the transport equation for mass, momentum and energy. Pressure field is obtained using SIMPLE algorithm for Pressure-Velocity coupling.

6.2.3. Spatial discretization

CFD analysis is the control volume technique numerical scheme for solving mass, momentum and energy second order upwind scheme is selected for the x,y,z velocities as well as κ–ε turbulence factor governing equation.

k-ε model:

The k-ε model is the most widely used turbulence model, particularly for industrial computations and has been implemented into most CFD codes. It is numerically robust and has been tested in a broad variety of flows, including heat transfer, combustion, free surfaces and two-plane flows. Despite various shortcomings which has been discovered over the past three decades of use and validation, it is generally accepted that k-ε model usually yields reasonably realistic predictions of major mean flow features in most situations. It is particularly recommended for a quick preliminary estimation of the flow field, or in situations where modelling other physical phenomena, such as chemical reaction, combustion, radiation, multi-phase interaction, brings in uncertainties that outweigh those inherent in the k-ε turbulence model.

The k-ε model consists of transport equations for k & ε. In the k-ε model viscous diffusion is treated in the same form, but for high Reynolds numbers flow can be neglected (except close to the solid wall).

Summary of the k-ε model

$$\rho\frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial X_j} = P + G - \varepsilon + \frac{\partial}{\partial X_j}(\mu + \frac{\mu_t}{\sigma_k}\frac{\partial \varepsilon}{\partial X_j})$$

$$\rho\frac{D\varepsilon}{Dt} = \left(C_{\varepsilon 1} P + C_{\varepsilon 3} G + C_{\varepsilon 4} k \frac{\partial U_k}{\partial X_j} - C_{\varepsilon 2}\, \varepsilon\right)\frac{\varepsilon}{k} + \frac{\partial}{\partial X_j}\left(\frac{\mu_t}{\sigma_\epsilon}\frac{\partial_\epsilon}{\partial X_j}\right)$$

Where,

$$P = \frac{-2\mu_t S}{S - \frac{2}{3}[\mu_t(trS) + k](trS)}$$

$$G = -\frac{\mu_t}{\rho\sigma_\rho}\nabla\rho$$

$$\mu_t = C_\mu \rho \frac{k^2}{\epsilon}$$

6.2.4. Boundary condition

Bell shaped hemispherical inlet is considered as stagnation pressure (1 bar) and static pressure (95000pa) is considered in the swirl monitoring plan. Thus boundary conditions are exactly same as the test pressure in the flow bench measurement. Walls are assumed to be adiabatic and no slip.

Steady state CFD analysis results are tabulated in the below table 2. That shows the flow coefficient and swirl ratio for different valve lift conditions.

Table 2. CFD flow measurement

Lift (mm)	Flow coefficient	Swirl Ratio
1	0.10	0.30
2	0.28	0.80
3	0.43	1.30
4	0.50	1.40
5	0.63	1.50
6	0.72	1.72
7	0.75	1.74
8	0.80	1.60

6.2.5. UDF method for swirl

User defined function (UDF) is to calculate the angular momentum flux at the swirl monitoring plane shown in Fig.5. cell centriod from the swirl axis, mass flux and also velocity magnitude at each cell in the monitoring plane is obtained from the solver by the UDF at the end of the calculation. Thus inputs were used to calculate angular velocity and axial velocity which are required to calculate swirl ratio.

7. Intake manifold design

Performance of the engine is closely related to the air motion within the intake manifold. Hence the flow phenomena inside the intake manifold should be understood in order to achieve higher volumetric efficiency and lower emission. Minimization of flow resistance which offer better breathing effect in well designed intake manifold. Thus geometry of intake manifold has strong influence on the volumetric efficiency of I.C. engines.

Uniform distribution of air and fuel mixture in the each cylinder is the important criteria for good intake manifold. Tuning effect and ram are controlled by low air flow resistance and runner branch length. Runner and branch size must be large enough to maintain the air flow without reducing the air velocity to become too low to transport fuel droplets. Runner length and its diameters should be optimized to have uniform amount of air and fuel in engines. The runners should have no protrusion edges to minimize the air flow resistance. Air will always available in turbocharged engine in boost conditions as per required control by throttle plate. Proper design of intake system should offer the benefits of volumetric efficiency, torque increase, fuel economy and better emission control. Flow characteristic and geometrical shapes are having large amount of attention for the best intake system design.

The air flows across all the cylinders in the engines are not identical even under steady state conditions, because runner and branch length are not same for all cylinders and other geometrical details that affect the flow in the each cylinder. Also flow coefficient across each cylinder plays a vital role in the flow distribution.

Existing intake manifold (Side entry inlet) Modified intake manifold (Center entry inlet)

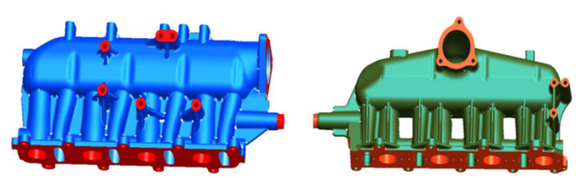


Figure 8. Intake Manifold CAD model by using Pro/Engineer

Existing intake manifold (Side entry inlet) Modified intake manifold (Center entry inlet)

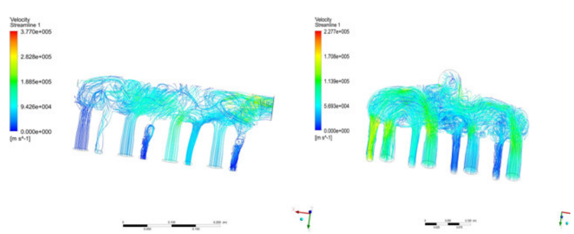


Figure 9. Intake Manifold streamline CFD model

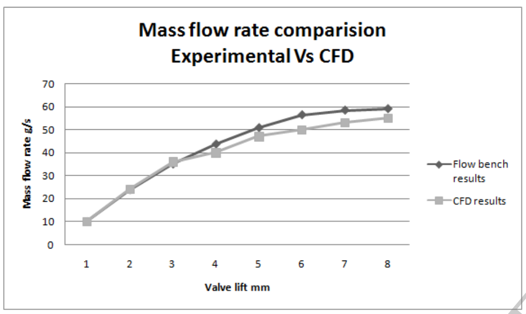


Figure 10. Mass flow rate comparison graph

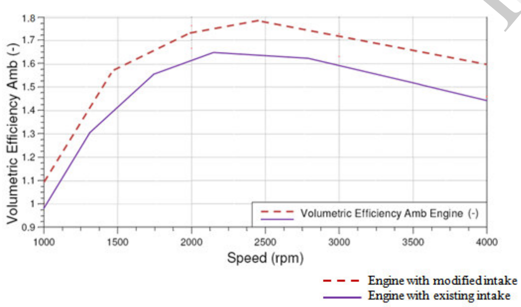


Figure 11. Volumetric efficiency comparison graph

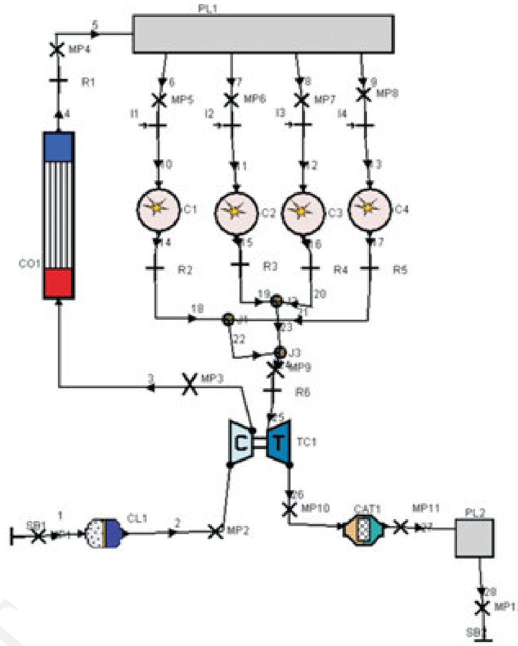


Figure 12. One dimensional analysis model for engine performance characteristic by using AVL boost software

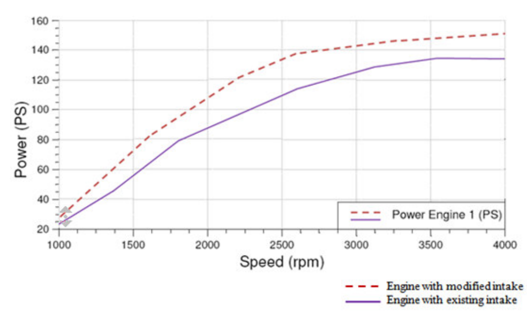


Figure 13. Power comparison graph

8. Results and discussion

Experimental measurement and CFD analysis are correlated with each other. Flow coefficient and swirl ratio were compared between both CFD and flow bench.

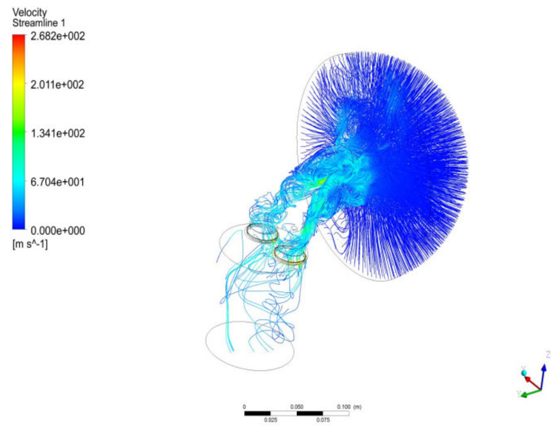
8.1. General flow properties

All the flow coming from the inlet goes directly to the port around the valve stem. The flow coming from lower part of port exits in the same direction, But the flow from upper part of port exits in opposite direction.

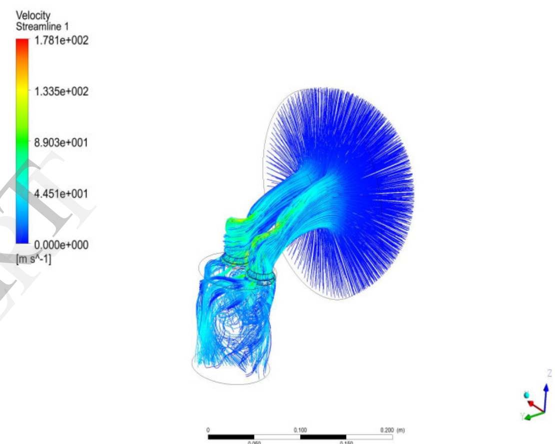
Swirl inside the chamber mainly created by the flow from upper part of the port. Streamlines released from inlet shown in Fig.14. much of the flow coming from the inlet goes directly into the chamber but some hugs the outer diameter of the port around the valve stem. Especially, the flow coming from the lower part of the port (flow from inner diameter) exits in the same direction whereas the flow from upper part (flow from outer diameter) of the port exits in the opposite direction.

The flow from upper part is mainly responsible for swirl inside the engine cylinder. Streamlines released from inlet shows this flow feature and is shown in fig.14. Swirl velocity is created stronger in lower lift due to strong pressure different in the throat. But in the high lift Swirl velocity is less.

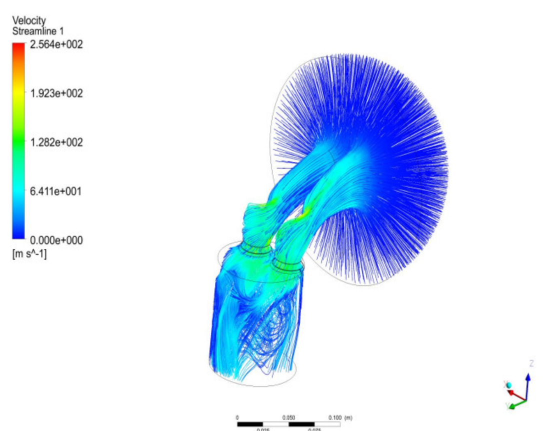
Strong pressure differential exists near the throat which resulted in flow acceleration through the throat and hence high velocity. Fig.15. shows the static pressure and velocity across the throat for various valve lifts.



Low lift(2mm valve lift)



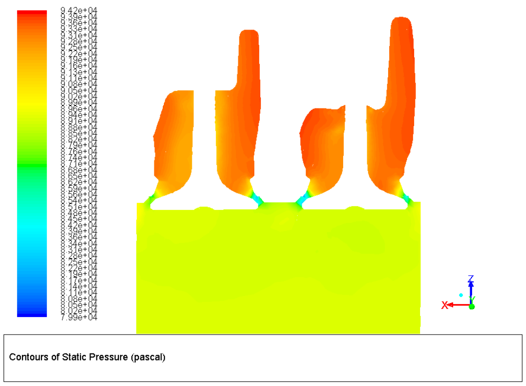
Medium lift(5mm valve lift)



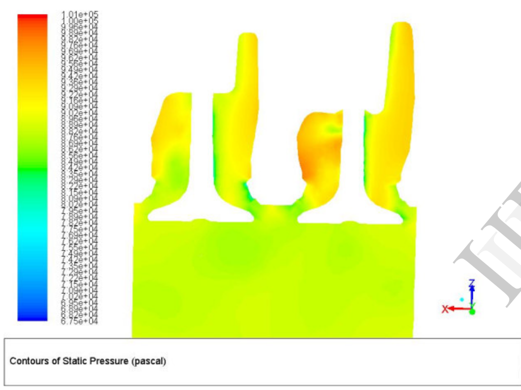
High lift(8mm valve lift)

Figure 14. Velocity magnitude streamlines

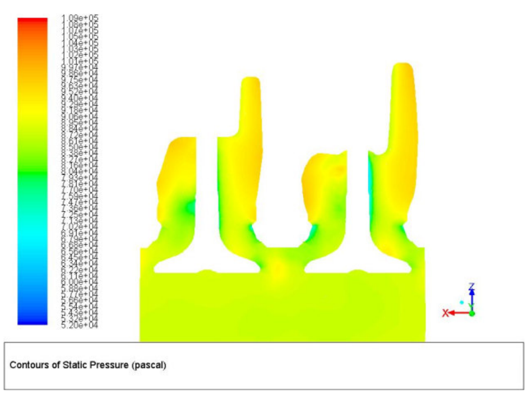
8.2. Pressure and velocity distribution



Low lift(2mm valve lift)

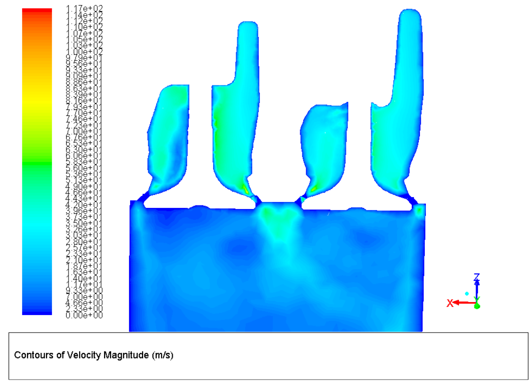


Medium lift(5mm valve lift)

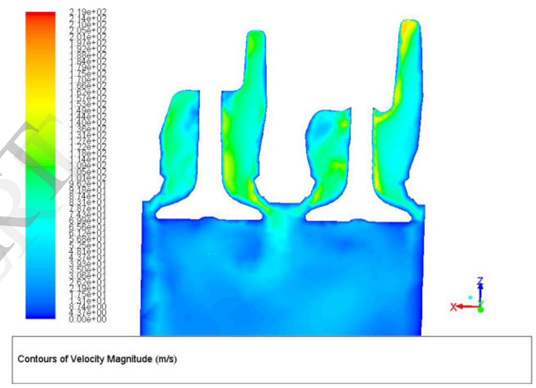


High lift(8mm valve lift)

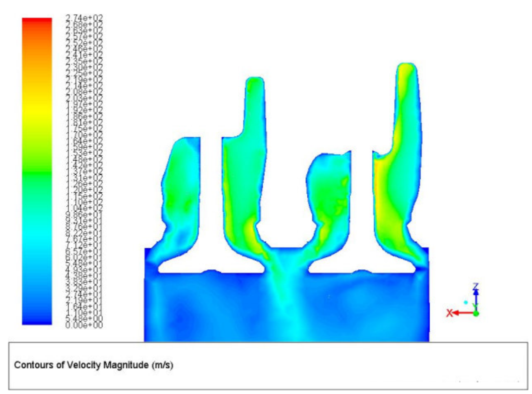
Figure 15. Static pressure contours



Low lift(2mm valve lift)



Medium lift (5mm valve lift)



High lift (8mm valve lift)

Figure 16. Velocity contours

Pressure different are stronger in throat which results in high velocity flow through the throat.

Fig.15 & 16 shows the static pressure and velocity contour across the throat for different valve lift conditions.

The flow is attached to the wall at the lower valve lift which causes less turbulence near the throat. Some of the air get coming out from the throat creates the tumble motion below the valve head. Some of the air jets having higher velocity flow out from the throat creates angular torque around the cylinder axis which can turn in to evaluating swirl ratio.

8.3. Flow coefficient comparison

Flow coefficient and mass flow rate are compared between CFD and flow bench. The value shows less different in low valve lift. This could be because of air flow is closely attached with the wall at lower valve lift. But at higher valve lift flow start separating from the wall causes turbulence near the throat . Thus the flow rushing into the chamber creates more upstream which results in more air flow at higher valve lift. The flow properties are shown in Fig.15 & 16.

The flow coefficient comparison between CFD and flow bench shown in Fig.17.

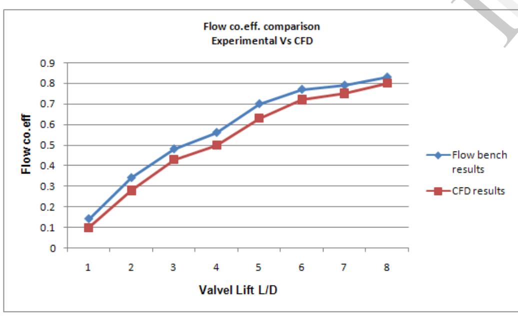


Figure 17. Flow co-efficient comparison graph

8.4. Swirl ratio comparison

Swirl ratio are having more different in the low valve lift condition then the high valve lift because at low valve lift CFD is over predicting the circumferential flow velocity and under predicting the mass flow rate. These deviation starts to come down when the valve lift increases.

Overall CFD analysis closing matching with the flow bench measurement, but CFD analysis always under predicts compare with flow bench, because mass flow rate is measured in the compensating tank in flow bench, but it was measured in BTD of the cylinder in the CFD and swirl angular momentum measured in the 2D swirl monitoring plane. But it was measured in the paddle wheel in the flow bench, more over the flow is highly three dimensional so it is always little difficult to match the circumferential velocity of CFD and flow bench.

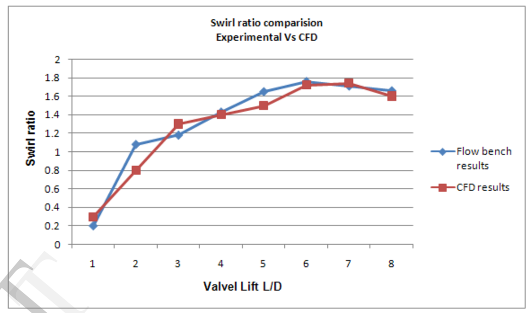


Figure 18. Swirl ratio comparison graph

9. Conclusion

The following conclusions were made with regards to flow properties:

- CFD analysis and flow bench measurement are closely matching with each other.
- Simulation approach is carried out for existing system and modified intake configurations to achieve the volumetric efficiency increase from 168% to 178% by increasing the flow coefficient from 0.65 to 0.80 at high valve lift segment and optimized swirl torque those enhance optimum performance for the engine and operating conditions. Simulation results shows that modified inlet port with lower swirl ratio and higher discharge coefficient exhibits better engine performance results.
- At lower valve lift more deviation is coming for swirl ratio, because (i) Cumulative calibration error in the flow bench system, (ii) Type of turbulence

model used to resolve boundary layer at the throat, (iii) Flow bench intake port may have little deviation from the CAD model.

- Swirl ratio is closely matching with CFD and flow bench, even though the paddle wheel speed is different between flow bench and CFD. But at high valve lift it is having little deviation because 2D monitor plane is used in CFD in place of paddle wheel.

Nomenclature

Symbols

A	Piston Area
C_s	Flow velocity
D_{cyl}	Cylinder bore diameter
D_{MFL}	Mean paddle wheel diameter
k	Specific heat ratio = 1.40
n	Paddle wheel speed (swirl)
P_1	Stagnation pressure upstream valve
P_2	Stagnation pressure in compensating tank
P_{amb}	Test ambient pressure
P_{std}	Standard ambient pressure
R	Universal Gas Constant
T_{amb}	Test ambient temperature
T_{std}	Standard ambient temperature
V	Volume flow rate measured in test bench
V_{real}	Theoretical volume flow rate
ρ_{real}	Density under test ambient conditions
ρ_s	Density under isentropic conditions
ρ_{std}	Density under standard ambient conditions

Subscripts

s	Isentropic conditions

Notation

Δ	Difference

Abbreviation

cyl	Cylinder Bore
MFL	Mean paddle wheel
std	Standard conditions
real	theoretically calculated
amb	ambient conditions

References

[1] He Changming, Xu Sichuan, Zuo Chaofeng, Li Chuanyou, "Multi-valve intake port parametric design and performance optimization of the horizontal diesel engine", *ISSN 1392 - 1207. MECHANIKA. 2011.* December 2011. 17(6): 643-648

[2] Javad Kheyrollahi, Mojtaba Keshavarz, S.A Jazayeri, Mohsen Pourfallah, "Optimization of Intake port shape in a DI diesel engine Using CFD Flow Simulation", *INTERNATIONAL COUNCIL ON COMBUSTION ENGINES- CIMAC Congress,* DESA, K.N.T.U University of technology, Iran, Bergen,2010.

[3] David Rathnaraj and Thathapudi Michael Narendra, "Studies on variable swirl intake system for diesel engine using computational fluid dynamics", UDC: 5.621.43.041.6:532.517.4:66.011BIBLID: 0354-9836, 12 (2008), 1, 25-32 DOI: 10.2298/TSCI0801025J, March 2008.

[4] Michele Battistoni, Angelo Cancellieri and Francesco Mariani, "Steady and Transient Fluid Dynamic Analysis of the Tumble and Swirl Evolution on a 4V Engine with Independent intake Valves Actuation", *SAE paper 2008-01-2392,* University of Perugia, Oct 2008.

[5] Bertrand fasolo, Anne-marie doisy, Alain dupont, "Combustion system optimisation of a new 2 Litre Diesel engine for EURO-IV", *SAE paper 2005-01-0652*, Renault Power Train Engineering, 2005.

[6] Ravidra devi, Priyank saxena Veera rajendra, "Pressure reduction in intake system of a turbocharged intercooled DI Diesel engine using CFD methodology", *SAE paper 2004-01-1874*, 2004.

[7] M.C.Bates, "Knowledge based model for multi valve diesel engine inlet port design", *SAE paper 2002-01-1747*,Ricrdo consulting Engineering Ltd, M.R.Helical University of Brighton, 2002.

[8] Harishchandra Jagtap, Chavan Vinayak and Ravindra Koli, "Intake System Design Approach for Turbocharged MPFI SI Engine", *SAE paper 2011-26-0088*, Mahindra & Mahindra Ltd, India, January 2001.

[9] John B. Heywood, "Text book on Internal Combustion Engine Fundamentals", *McGraw-Hill* Education private limited, PP-326-365, 1997

[10] Frank M. White, "Fluid Mechanics", *McGraw-Hill* Education private limited, Second Edition, 1986.