Virtual And Experimental Optimization Of Cylinder Head Swirl For Emission Reduction In Small Diesel Engine

Ashish Jain, Porpatham Ekambaram, Sukrut S Thipse

Abstract: Three dimensional CFD simulation approach involving Swirl and flow parameter is used for optimizing the cylinder head Swirl and Flow values for a single cylinder light duty LCV application diesel engine. Cylinder head Swirl of the baseline engine was 2.55 with flow coefficient value of 0.314. Engine out emissions for baseline engine were higher particularly NOx emission and is not able to meet the legislative emission limits. It is decided to reduce cylinder head swirl to counter this higher NOx emission also increase flow coefficient to take further benefit in terms of engine performance. CFD simulation approach is used to reduce Swirl in the intake port of Cylinder head with option of increasing the flow area. Inlet port core is extracted from cylinder head and scanned to generate baseline 3D model of intake port to carry out baseline CFD simulation. Three different Swirl options are proposed initially at interval of 0.2 swirl in simulation model to analyze the effect of Swirl reduction on engine performance and emission. Cylinder head intake port profile is modified on 2 different cylinder heads to generate new swirl and flow coefficient values based on simulation suggestions. These modified cylinder heads are used in engine for emission and performance improvement evaluation. The simulation approach has worked successfully for emission reduction as 18% NOx reduction is achieved with performance improvement of around 13% in other parameters.

Keywords: Cylinder head Swirl; Flow coefficient, CFD simulation, NOx emission, Intake Port Optimization

1. INTRODUCTION

Diesel engines are among the best available and reliable source of power for all domestic, large scale industrial and transportation applications in the world. Known for its strong, economical and robust operations under all conditions; however they are also known for high smoke and NOx emissions [1]. Size of these engines varies from small single cylinder engine used for commercial application to large engines used to propel ships. Reason for their wide application is high power density, fuel economy, economical and robustness under all operating conditions. The need for greater fuel economy, enhanced performance and drivability prospects of customers from engines are progressively increasing along with stringent emission norms from government side. The major issue is the efficiency of these engines. Attempts have been made and are still going on, to improve efficiency of these engines to attain the maximum possible efficiency [2]. Owing to the lack of throttling losses and higher compression ratio, diesel engines have low brake specific fuel consumption (BSFC) and thus low carbon dioxide (CO2) emissions. Furthermore, because of its ability to operate at low equivalence ratios, the diesel engine produces low carbon monoxide (CO) and hydrocarbon (HC) emissions. Unfortunately, the diesel engine suffers from relatively high nitrogen oxides (NOx) and particulate matter (PM) emissions. These engine out emission varies as per application and stringent norms imposed by government. Research is going on all around the world to reduce this pollution using latest technologies like combustion optimization, low temperature (LTC), HCCI, PCCI, supercharging, twin turbo, VVT/VVA, and alternative fuels. Diesel engine combustion appears to be the most prominent way to improve fuel efficiency for global CO2 emissions reduction. Combustion optimization is technological way for improvements, which have to be made throughout the engine, from incoming air and fuel to the outgoing exhaust gases as emissions. This process includes optimization of parameters affecting the combustion. These parameters are Fuel injection pump, injection rate and timing, fuel air mixing, intake port swirl, squish, cam timings, nozzle tip protrusion, piston bow shape, turbocharging, intercooling, EGR and design of peripheral components related to these parameters. Intake port swirl and flow coefficient is important parameter affecting fuel air mixing, combustion and consequently the performance and emission from the engine. In-cylinder fluid motion, flow structure and its strength with valve lift are important factors controlling the combustion process [3]. Flow stability, its regularity degrades at lower lift for low swirl cylinder heads [4, 5] Swirl is induced by good intake port design. Swirl and squish interaction is very complex phenomenon; higher swirl leads to higher turbulent flow field and kinetic energy at higher lifts were observed by Prasad [6]. Gafoor and Gupta [7] observed that with increase in swirl ratio the turbulence increases and hence results in better air-fuel mixing process, which further results in increase in the thermal efficiency and reduction in SFC and soot emission. NOx emission is increased slightly owing to better mixing and a faster combustion process. Higher swirl leads to enhance fuel air mixing and can provide benefits in soot oxidation and shorter combustion duration. It also hinder the combustion and prevent air fuel mixing due to higher angular velocity. Alberto et.al [8] studied effect of swirl on 4 cylinder CRDI variable geometry turbocharged engine. Effect of higher external EGR and variable swirl effect was analyzed on cylinder, liner wall and heat release rate. Same engine

• Ashish Jain*, Powertrain Engineering Department, Automotive Research Association of India, Pune, India 411038 Email: jainashish1985@yahoo.com
• Prof. Porpatham Ekambaram, Automotive Research Lab, Vellore Institute of Technology University, Vellore, India 632014. Email: porpatham.e@vit.ac.in
• Dr Sukrut S Thipse, Powertrain Engineering Department, Automotive Research Association of India, Pune, India, 411038 Email: thipse.edl@araiindia.com

USTR©2020
www.ijstr.org

519
configuration was used by Benjas et.al [9] to study the effect of swirl ratio on global energy balance of the engine. They observed that effect also depend on engine operating condition, heat transfer and heat release rate. Higher swirl lead to increase in heat release rate however this rate get affected in different ways. It had increased hear rejection to cooling system and higher heat transfer at intercooler. The present work is study of emission reduction in a light duty high speed commercial vehicle application diesel engine by optimizing the swirl ratio and flow characteristic with help of 3D CFD approach. The optimization work performed in actual cylinder head is based on simulation results. This paper is structured initially 3D computation model used for swirl optimization with a focus on modelling of swirl generation process and modification of port performance. The major constraint is that no material can be added in the cylinder head and main focus is to modify the port core geometry. This will not require any material addition on actual cylinder head, and same needs to be replicated in actual cylinder head, which is iterative process need skills and caution at the same time. The performance is validated on steady state swirl rig and this modified port performance data is used in 1D simulation model to check engine performance and emission. Finally these modified cylinder heads are used in the engine for validation.

2. SWIRL GENERATION AND MEASUREMENT METHODS

There are two basic approaches used to generate swirl during the induction process. For first approach, the flow is discharged into the cylinder tangentially towards the cylinder wall, where it is deflected sideways and downward in a swirling motion. In other words, swirl is largely generated within the intake port. The flow is forced to rotate about the valve axis before it enters the cylinder. The former type of motion is achieved by forcing the flow distribution around the circumference of the intake valve to be non-uniform, so that the intake flow has a substantial net angular momentum about the cylinder axis. Swirling flow can also be generated by valve masking or valve grooving so that air rotation about the valve axis can be generated and further it gets developed in the cylinder. For research engine this methodology is used to develop swirl. Intake Ports are also masked or shrouded in the valves region in research engines as adaptation is easy and advantage in terms of cost and times. The swirling flow is extremely difficult to examine and understand under actual engine operating condition due to its nature of flow. Hence steady state flow tests bench with paddle wheel and Impulse type measurement principle is used to analyzed and categorize the swirling flow happens through the intake port cavity of cylinder head or replica of cavity in form of flow box. Paddle wheel is mounted at a height of 1.75D from cylinder head bottom (where ‘D’ is Cylinder bore diameter). Air is forced to flow through the intake port cavity in the cylinder head with blowers sucking the air on the other end. As this air passes through port cavity, due to port geometry it comes with swirling motion into the cylinder and rotates the paddle wheel. This speed of the paddle wheel is measured with non-contact type measurement principle. Fig. 1 shows Steady State Swirl Measurement system with position of Sensors and Paddle wheel.

The flow coefficient \( \mu_s \) is calculated as the ratio of actual (measured) mass flow rate of air to theoretical mass flow rate of air.

\[
\mu_s = \frac{m_{\text{actual}}}{m_{\text{theo}}}
\]  
(1)

Theoretical mass flow rate is calculated at the valve seat reference cross sectional area \( A_v \) of minimum seat diameter \( d_v \) using formula.

\[
A_v = \frac{\pi d_v^2}{4}
\]  
(2)

The flow area \( A \) between valve and valve seat is expressed as \( A = s A_v \). Where, \( s \) is an obstruction coefficient depends on the geometric conditions and the valve lift.

The theoretical mass flow is defined as:

\[
m_{\text{theo}} = A_v \rho \sqrt{\frac{\Delta p}{\rho m}}
\]  
(3)

\[
\rho = \rho_v \left( \frac{P_{\text{in}}-\text{dp}}{P_{\text{in}}} \right)^{\frac{\gamma}{2}}
\]  
(4)

\[
\rho_m = \frac{1}{2} (\rho_v + \rho)
\]  
(5)

During testing, the thank pressure difference is maintained as 250 mmH2O for valve lifts up to 5 mm and 600 mmH2O for valve lifts lower than 5mm.

Using paddle wheel anemometer, the in cylinder charge rotation is calculated for several valve lifts. The paddle wheel speed is calculated by integrating the definition.

\[
\left(\frac{0.561}{2}, D\right) \leq r \leq \left(\frac{0.917}{2}, D\right)
\]  
(6)

Swirl ratio is evaluated from the ratio of the charge rotation speed of paddle wheel to the fictitious engine speed.

\[
SR = \frac{n_p}{n} = \frac{n_{\text{paddle}}}{n_{\text{motor}}}
\]  
(7)

Equating mean piston speed to mean axial velocity, we get the fictitious engine speed to be

\[
n_{\text{motor}} = \frac{v_{ax} \cdot S}{\rho \cdot A \cdot S}
\]  
(8)

Swirl ratio of the paddle wheel can be calculated as

\[
SR = \frac{n_{\text{paddle}}}{n_{\text{motor}}} = \frac{A \cdot S}{30 \cdot \rho \cdot n_{\text{paddle}}}
\]  
(9)

Mean swirl number is also obtained by integration over the crank angle (from TDC to BDC) considering valve lift and position speed. The mean swirl number is dependent upon the stroke and bore ratio.
3. EXPERIMENTAL TEST SETUP TEST SETUP

Baseline engine performance is measured on an engine dynamometer with AVL AMA emission measurement system. Detailed engine specification is mentioned in Table-1. Baseline engine emission measurement is found on higher side (Particularly NOx emission) and engine was meeting the legislative emission limit. It is decided to reduce this emission by reducing the swirl and increase in flow to get benefit in volumetric efficiency along with emission.

**Table: 1, Engine Specification**

<table>
<thead>
<tr>
<th>Type</th>
<th>Four stroke, Air cooled, single cylinder, CI engine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fuel</td>
<td>Diesel</td>
</tr>
<tr>
<td>Number of cylinders</td>
<td>One</td>
</tr>
<tr>
<td>Bore / Stroke Ratio</td>
<td>1.15</td>
</tr>
<tr>
<td>Volume</td>
<td>0.43 Ltr</td>
</tr>
<tr>
<td>Compression ratio</td>
<td>19:1</td>
</tr>
<tr>
<td>Rated Power</td>
<td>5.2 kW @ 3600 rpm</td>
</tr>
<tr>
<td>Rated Torque</td>
<td>18 Nm @ 2200-2400</td>
</tr>
<tr>
<td>Swirl Ratio</td>
<td>2.55</td>
</tr>
<tr>
<td>Flow Coefficient</td>
<td>0.314</td>
</tr>
<tr>
<td>Start of Injection</td>
<td>10±1° CAD BTDC</td>
</tr>
</tbody>
</table>

4. SWIRL MODELLING

In this section we will discuss about the Steady State flow testing and development of computational fluid dynamics simulation model for Intake port.

4.1 Simulation Strategy on Intake Port

Solving a fluid dynamics problem is actually solving a coupled system of nonlinear partial differential equations; however, there is no general analytical solution to these equations. Appropriate numerical solutions play a significant role in understanding the performance of fluid dynamics described by the mathematical equations. Inlet port core is extracted from cylinder head with rubber mould (Fig. 2) and 3D scanned to get the baseline 3D model of the port core. Baseline CFD model is built on AVL Fire software based on the digitized 3D model and calibrated with baseline cylinder head results on steady state swirl rig.

Hybrid type meshing is used with number of cell ranging between 8, 00,000 to 10, 00,000 to get a converge solution. Cold flow steady state simulation approach with k-zeta-f turbulence model is used with 3500 iteration as convergence criteria. Whole iteration is carried out considering compressible flow of air inside the port. Boundary condition at intake end is taken as atmospheric air condition (100000 Pa) at port inlet and 97500 Pa (equivalent to 250 mm of water column) at liner end. The test setup is shown in the Fig.1. Three different swirl and flow values are considered initially to be achieved through CFD simulation approach and out of these three the lowest swirl and highest flow coefficient value cylinder head is used for engine optimisation.

4.2 Swirl Adaption on Cylinder Head

Based on 3D CFD analysis results of inlet ports, cylinder head geometry is modified with pneumatic grinder. This is an iterative process and requires skill to carry out experimental modification. Modified cylinder head port performance is measured after each machining iteration till the required swirl and flow is achieved on cylinder heads. Time taken to carry out this modification on single head is 4-6 days. This process is carried out for 2 level of swirl (1.9 and 2.1) on cylinder heads and same is used on experimental engine for performance improvements trails.

5. RESULTS AND DISCUSSIONS

In this section we will discuss the results of 3 Dimensional CFD, Modified Cylinder head results, and Emission Performance of engine with modified cylinder heads on engine test bed.

5.1 Swirl Modelling

Input .stl model and Meshed model of the inlet port core, mesh size, and number of cell is shown in Fig. 3. Fig. 4 shows the volume mesh model of intake port core. Fig. 5 shows the parameters defined in the simulation model and position of paddle wheel in which all sub-component of the assembly. In simulation and actual testing the position of paddle wheel remain same as 1.75D from Liner top face. In physical test inlet end is directly exposed to atmosphere; however, in case of simulation one hemisphere is used at inlet to get directions of velocities similar like test condition.

![Fig.2 Inlet Port Core Scanned from Cylinder Head model](image)

![Fig.3 Baseline Imported Model and Meshed Model of the Inlet Port](image)
Baseline 3D CFD simulation model is calibrated with testing results for the 7 different set of valve lifts and shows very good correlation in terms of swirl and flow. The deviation is shown in Table 2.

### Table 2, Comparison of Test Bench and Simulation Results for Baseline Intake Port

<table>
<thead>
<tr>
<th>Valve Lift</th>
<th>Swirl</th>
<th>Flow coefficient</th>
<th>% dev</th>
<th>Swirl</th>
<th>Flow coefficient</th>
<th>% dev</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exp.</td>
<td>1.914</td>
<td>0.43</td>
<td></td>
<td>1.859</td>
<td>0.45</td>
<td></td>
</tr>
<tr>
<td>CFD</td>
<td>1.873</td>
<td>1.793</td>
<td>4.462</td>
<td>0.410</td>
<td>0.42</td>
<td>-7.805</td>
</tr>
<tr>
<td>% dev</td>
<td>2.959</td>
<td>0.42</td>
<td>6.840</td>
<td>0.453</td>
<td>0.42</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>8</td>
<td>7</td>
<td></td>
<td>8</td>
<td>7</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>6</td>
<td>5</td>
<td>6.120</td>
<td>0.395</td>
<td>0.410</td>
<td>-3.797</td>
</tr>
<tr>
<td>6</td>
<td>5</td>
<td>4</td>
<td>4.180</td>
<td>0.318</td>
<td>0.328</td>
<td>-3.145</td>
</tr>
<tr>
<td>5</td>
<td>4</td>
<td>3</td>
<td>3.898</td>
<td>0.240</td>
<td>0.257</td>
<td>-7.083</td>
</tr>
<tr>
<td>4</td>
<td>3</td>
<td>2</td>
<td>11.565</td>
<td>0.219</td>
<td>0.230</td>
<td>-5.023</td>
</tr>
</tbody>
</table>

The maximum deviation for swirl value is 11% for lower lift values. Reason for this variation is lesser valve to seat clearance area at lower lifts, hence the resistance to flow will be higher making it difficult for air to flow. Even though the swirl values are lower for all CFD iterations while flow values are slightly higher, it is because of friction effect on actual casting as surface finish for 3D geometry is always better than actual casting surface. This surface finish leads to pressure drop in the flow path due to resistance and hence velocity will be increased as per conservation of energy principle. Comparison of velocity vectors and Planner section view for the Inlet port core at intermediate lift condition (6 mm) for Baseline port and Iteration-3 port is shown in Fig. 6 along with sectional flow pattern at various planes for modified port. The velocities for the baseline port and Iteration-3 port are 100m/s and 65m/s respectively. This clearly shows the reduction in Swirl ratio for the Iteration 3 ports. Fig. 7 shows the changes in the 3D geometry of inlet port core to achieve the results for Iteration-2. Two more iterations are carried out wherein the inlet port core profile is modified to reduce the swirl from existing level of 2.48 in simulation to new level of around 1.9 at an interval of 0.2 swirl. Results of all these iterations are shown in Table 3 wherein Swirl and Flow values of each iteration are compared. As we increase the flow area, the inlet port swirl values are get reduced as the resistance to the flow is decreased and hence pressure drop across port will also be lower. This effect can be clearly seen over the flow coefficient values, as it is increased with each iteration.

### Table 3: Comparison of Swirl and Flow Coefficient for Various Simulation Iterations

<table>
<thead>
<tr>
<th>Valve Lift</th>
<th>Baseline</th>
<th>Iteration -1</th>
<th>Iteration -2</th>
<th>Iteration -3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swirl</td>
<td>Flow</td>
<td>Swirl</td>
<td>Flow</td>
<td>Swirl</td>
</tr>
<tr>
<td>8</td>
<td>2.01</td>
<td>0.43</td>
<td>8</td>
<td>1.95</td>
</tr>
<tr>
<td>7</td>
<td>2.03</td>
<td>0.42</td>
<td>4</td>
<td>2.00</td>
</tr>
<tr>
<td>6</td>
<td>2.15</td>
<td>0.39</td>
<td>6</td>
<td>1.87</td>
</tr>
</tbody>
</table>

The parameters defined in the Intake Port Model for Simulation and Simulation Results for 6 mm are shown in Fig. 6.
The maximum increase in Mean flow coefficient is 11.8% between baseline and iteration 3 for 8 mm lift and overall it is 7% compared to baseline simulation model. Mean Swirl Ratio is also reduced by 25% compared to baseline simulation model. It will have a positive impact on engine's volumetric efficiency, emissions result and performance.

Reduction in swirl is further planned however it leads to more changes in the intake port profile, which is not possible as there will always some deviation between 3D model and actual casting due to manufacturing tolerances. Also wall thickness of the intake port profile will becomes very less locally and leakage can start from outside to the intake port, which can affect the swirl pattern and flow coefficient.

5.2 Adaptation of CFD Modification on Cylinder Head

3D simulation modification needs to be adapted on engine's cylinder head for reduction of swirl ratio and to increase flow coefficient. For the first few iterations extra care is taken during modification operation and quantity of material removed is less to get the correlation of material removal to swirl reduction. However after certain iteration and repetitive testing, we are able to achieve the desired results. Fig. 8 show the bottom view of the modified port profile for cylinder head. Extra material is also removed from inlet port helix region for reduction of swirl however there was hardly any improvement in flow coefficient for initial iteration-1. Fig 9 shows the Modified Cylinder heads for Iteration-2, in which baseline cylinder head was used for modification.

Similar operation is carried out on other cylinder heads for reduction of swirl by removing the material in port lip region and helix portion i.e., modification of port profile. As all iterations are carried out on actual cylinder heads, replicating the 3D model is not possible. However great improvements are observed in further iterations. Fig.9 shows the Comparison of modified Cylinder head with respect to original cylinder head. It is very clear that, due to material removal, port flow area is increased and resistance to flow is decreased. Table 4 shows the improvement in each iteration when tested on steady state swirl rig.

![Baseline Port](image1)

![Modified Port](image2)

### TABLE 4: Comparison of Swirl and Flow Coefficient for Various Modification Iterations

<table>
<thead>
<tr>
<th>Valve Lift</th>
<th>Baseline</th>
<th>Iteration -1</th>
<th>Iteration -2</th>
<th>Iteration -3</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>2.158</td>
<td>0.451</td>
<td>1.883</td>
<td>0.451</td>
</tr>
<tr>
<td>7</td>
<td>2.261</td>
<td>0.438</td>
<td>2.073</td>
<td>0.451</td>
</tr>
<tr>
<td>6</td>
<td>2.053</td>
<td>0.410</td>
<td>1.876</td>
<td>0.424</td>
</tr>
<tr>
<td>5</td>
<td>1.762</td>
<td>0.379</td>
<td>1.770</td>
<td>0.363</td>
</tr>
<tr>
<td>4</td>
<td>1.738</td>
<td>0.318</td>
<td>1.738</td>
<td>0.310</td>
</tr>
<tr>
<td>3</td>
<td>0.833</td>
<td>0.240</td>
<td>0.796</td>
<td>0.240</td>
</tr>
<tr>
<td>2</td>
<td>0.328</td>
<td>0.230</td>
<td>0.210</td>
<td>0.219</td>
</tr>
<tr>
<td>Mean</td>
<td>2.55</td>
<td>0.314</td>
<td>2.26</td>
<td>0.314</td>
</tr>
</tbody>
</table>

It is observed from the above table and Fig.10 that baseline tested Cylinder head has highest swirl while Cylinder head modified after Iteration-3 has the lowest Swirl Ratio. Iteration
2 and 3 have better flow coefficient compared to baseline cylinder head (Fig.11). Reason is very clear, lower resistance to the flow path leads to lower pressure drop across port and higher flow area leads to lower velocity and momentum of air molecules. Iteration 3 has 11% higher flow coefficient and 25% lesser swirl at 8mm valve lift. The overall gain in Flow coefficient is 6% compared to baseline cylinder head while Swirl reduction is 25%. This gain in flow coefficient can give way for higher fuel injection and thus power output can also be increased.

5.3 Engine Optimization Results
Modified cylinder heads are mounted on the engine and performance is measured on engine test bed keeping all other engine operating conditions same as per baseline testing to measure improvement. Fig. 12 to 16 show the improvement in Power, Torque, BSFC, Volumetric efficiency and NOx emission throughout the speed range and this improvement is varying from 1% to 20% for different parameter at different rpm range.
for performance improvement. These improvements are not only limited to any particular speed range thus giving an extra degree of freedom for further improvement in swirl and flow coefficient.

6. CONCLUSION

Based on the 3D CFD modelling and optimization approach, cylinder head modification is carried out based on CFD. Modified Cylinder head is used in engine and testing has been carried out. In summary improvement in overall engine performance is 13% (Power and torque) while emission is reduced by 18%. This trends shows the important of Swirl for the reduction of NOx emission and performance improvements. These can be further used with other engine operating parameters to further reduce the emission. Following conclusions are drawn.

1. 3D CFD model of baseline inlet port and steady state swirl rig result of the cylinder head are calibrated for Swirl and Flow coefficient with correlation band of 8%.
2. Inlet port Swirl Ratio is reduced by 27% and Flow coefficient is increased by 6% on 3D model of inlet port core with help of CFD simulation.
3. Inlet port core on actual cylinder head is modified with help of Pneumatic machining.
4. Inlet port Swirl ratio is reduced by 24% and Flow is increased by 7% on actual cylinder heads when compared with baseline cylinder heads.
5. Engine with modified cylinder head shown the good trend of performance improvement in performance as we as emission reduction. Performance is improved by 13% overall and NOx emission is reduced by 18% compared to baseline engine.

7. REFERENCES

[8] Alberto Broatch, Pablo Olmeda, Antonio García, Josep

![Figure 14: Comparison of Volumetric Efficiency Improvement](image1)

![Figure 15: Comparison of NOx Improvement](image2)