Evaluation of CI In-Cylinder Flow using optical and numerical techniques

R. Rezaei[1], S. Pischinger[1], P. Adomeit[2], J. Ewald[2]*
[1]Institute for Combustion Engines, RWTH Aachen University, Germany
[2]FEV Motorentechnik GmbH, Aachen, Germany

In order to evaluate different port concepts for modern Compression-Ignition engines, usually quantities as the swirl level and the flow coefficient are evaluated, which are measured on a stationary flow test bench. As additional criterion, in this work, the homogeneity of the swirl flow is introduced and defined quantitatively. Different valve lift strategies are evaluated using three-dimensional Particle Imaging Velocimetry in a stationary flow configuration and transient In-Cylinder CFD simulation using both the Reynolds Averaged Navier Stokes equation and the Large Eddy simulation approach.

Introduction

New concepts for High-Speed Direct Injection Compression Ignition (CI) engines are under development due to the increased awareness of the CO_2 emission impact on global climate change which goes hand in hand with the demand of further reduced fuel consumption as well as the aggravated emission legislation standards.

In order to meet theses requirements for future CI engines, not only the injection system has to be suitably defined. Also, for the chosen injection system, the optimal in-bowl swirl has to be generated. The magnitude of the swirl optimum, however, is furthermore dependent on the operating point and engine speed. Therefore, in order to provide the corresponding flexibility, a CI engine concept has been developed that features a variable valve lift and port deactivation concept. By means of this, the optimal trade-off between swirl level and in-cylinder fresh charge filling level can be found.

It has been shown that different valve lift strategies nominally lead to similar filling and swirl levels. However, differences in combustion behavior and engine-out emissions give rise to the assumption that local differences in the in-cylinder flow structure caused by different valve lift strategies have noticeable impact.

In this work, these flow structures were analyzed and quantitatively assessed using both optical and numerical techniques.

Engine: Variable Charge Motion concept

The intake port of this DI diesel engine consists of tangential and filling ports [1]. Tangential ports can be used to generate a relatively high swirl ratio while the filling ports, as the name already implies, provide a high flow coefficient. Additionally the intake charge flow can be directed by machining the valve seat rings to yield swirl chamfers. This concept enables the generation of extremely high swirl numbers with low valve lifts without reducing the flow for high valve lifts. The impact of different valve strategies on the combustion system using a single-cylinder engine is assessed [1]. The test engine had the following design parameter:

- Basic engine: FEV system engine
- Bore x stroke: 75 mm x 88.3 mm
- Injection system: BOSCH 2000 bar Piezo
- Compression ratio: 15.3

As is shown in Figure 1, a reduction of the valve lift provides the best potential for emission behavior by increasing the swirl ratio. The utilization of the increased homogeneous swirl by reducing the valve lift reduces smoke emission significantly without any impact on fuel consumption. A further reduction of valve lift leads to a noteworthy increase of gas exchange losses, which finally leads to increased fuel consumption without any advantage concerning soot emission.

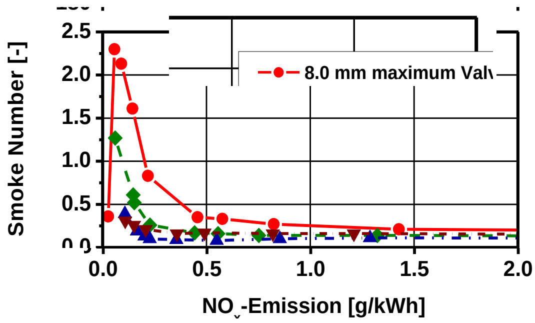

Figure 1: 1500 rpm, 6.8 bar emissions [6]

Increasing the swirl via reduced valve lift provides a slight improvement to the particulate air/fuel ratio trade-off from 8.0 mm to 3.2 mm valve lift. In Figure 2 in particular the NO_x/soot trade-off is shown in particular for two valve lift strategies, maximum lift of 4.8 mm vs. 8 mm max. valve lift with the filling port closed, which have the same swirl ratio. It was seen that the soot emissions with a closed filling port are considerably higher than for a lift of 4.8 mm. Therefore, the swirl level alone is insufficient to describe the in-cylinder flow. This was also found in [4].

* Corresponding author: ewald_j@fev.de

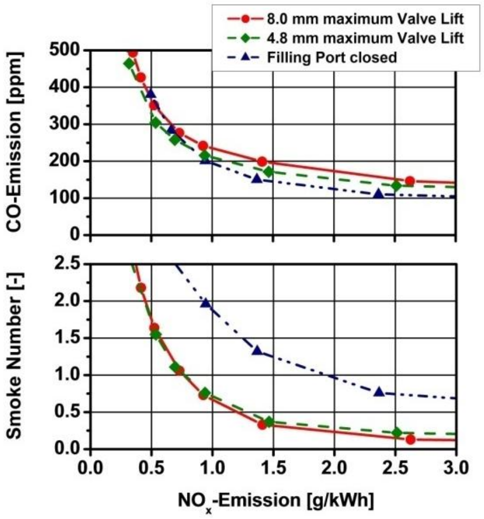

Figure 2: 2280 rpm 9.4 bar, emissions [1]

PIV measurements of stationary intake port flow

3D PIV stationary flow analysis of the new port design was performed for various valve lifts, and port deactivation strategies.

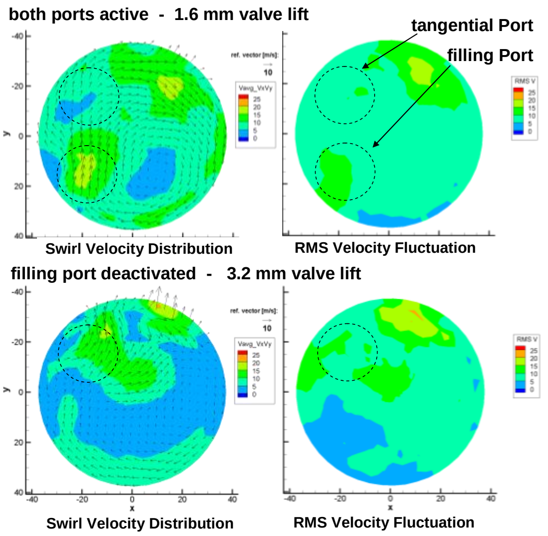

Figure 3: Charge motion analysis by 3D PIV without and with filling port deactivation at z = 75 mm

Figure 3 shows the flow field in a horizontal section 75 mm below the cylinder head. The average flow distribution is displayed as a vector field on the left of Figure 3 and the local distribution of the flow fluctuation intensity is displayed by the velocity RMS on the right side of the same figure. In both cases the intake flow rates are similar, but the resulting swirl flow patterns are strongly different. For the filling port deactivation, the swirl flow structure is less coherent, and fluctuation intensity is increased.

Table 1 compares the Root Mean Square (RMS) value of the measured tangential velocity and also the measured swirl ratio (C_U/C_A) at the PIV test bench.

	C_U/C_A	RMS_{Vtheta}	$\frac{RMS_{Vtheta}}{C_U/C_A}$
Filling port deactivated 3.2 mm	2.28	3.94	1.73
Both ports active 1.6 mm	5.66	3.79	0.67

Table 1: Results of swirl ratio and RMS of the tangential velocity.

As can be seen, the in-cylinder flow field generated when both ports are active is more homogeneous than the case with port deactivation.

Computational Setup

In this study, the commercial CFD software STAR-CD is used for the transient calculations of the intake and compression stroke with moving valves and piston. On the intake and exhaust port flange positions, pressure boundary conditions from GT-Power gas exchange calculations were employed. The calculation were performed from 360°CA to 720°CA. Two different turbulence models, the LES Smagorinsky [2] and also the k-ε model [3] are used for intake flow simulations.

Characterization of In-Cylinder Flow inhomogeneity

In order to quantify the in-homogeneity of the in-cylinder flow field different cut sections perpendicular to the cylinder axis are considered. Each of the considered cut sections is divided into concentric rings, shown in Figure 4.

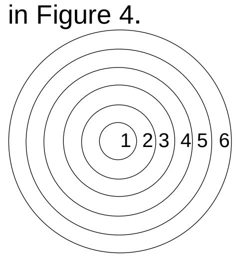

Figure 4: Top view of a cut section considering six rings

For each ring, first a mean value of the tangential velocity component is calculated. Then, for each of these rings the RMS of the tangential velocity is determined.

Simulation results using RANS and LES

The in-cylinder angular velocity is defined as angular momentum divided by the moment of inertia.

$$\omega_{in\text{-}cylinder} = \frac{\int_V \rho(vx - uy)\,dV}{\int_V \rho(x^2 + y^2)\,dV} \qquad (1)$$

The dimensionless swirl ratio for each operating point is then obtained according to

$$Swirl\ ratio = \left| \frac{\omega_{in-cylinder}}{\omega_{Engine}} \right| . \qquad (2)$$

Results of CFD simulations using the different valve lift and port strategies are shown in Figure 5. It can be seen that the in-cylinder swirl ratio can be increased to the same level either by reducing the maximum valve lift to 4.8mm or by port deactivation.

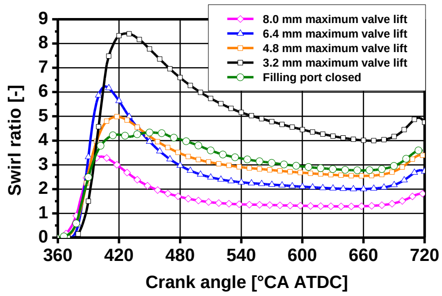

Figure 5: Swirl ratio over crank angle as calculated by RANS CFD simulations.

Inhomogeneity of in-cylinder flow

Figure 6 compares the cut sections of the tangential velocity field of each strategy at the middle of the piston bowl which is simulated using the k-ε model in STAR-CD at 2280 rpm.

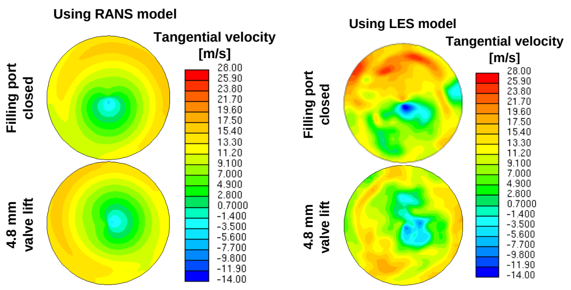

Figure 6: Cut section of the tangential velocity field in the middle of the bowl using the RANS model (left) and the LES model (right) at -30°CA ATDC

As from the RANS simulation, there is almost no difference between the two cases in terms of predicting the in-homogeneity of the flow field.

A cut section of the tangential velocity field in the middle of the bowl is shown also in Figure 6 for the same valve lift strategies using the LES model. The LES turbulence model captures turbulent flow structures while in RANS only the mean flow is resolved.

The RMS values of the tangential velocity from RANS and LES are compared for both valve strategies in Figure 7.

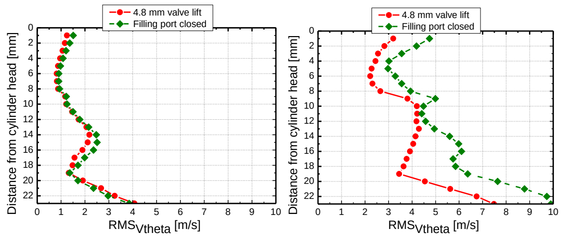

Figure 7: RMS value of tangential velocity using RANS(left graph) and LES(right graph)

As can be seen, differences in in-cylinder flow field between these valve strategies can be distinguished using the LES turbulence model rather than the k-ε model. The investigations with the LES model show that the 4.8 mm valve lift produces more homogeneous swirl than port deactivation. This is also in agreement with experimental investigations at the 3D PIV flow test.

Summary and conclusions

Differences in emission behavior for different valve lifts and with and without deactivated filling port were observed for a single-cylinder engine. Measurements on a stationary flow bench and CFD calculation both assessed the swirl level for the different concepts. While in particular a maximum valve lift of 4.8 mm for both intake ports produces the same swirl level as the maximum valve lift of 8 mm with the filling port deactivated, the soot emissions are significantly different and higher for the latter configuration

Therefore it was argued that next to the swirl ratio, another important parameter, describing these discrepancies, is required. An approach to evaluate the in-homogeneity of in-cylinder flow by means of PIV and CFD simulation was developed and presented. While in CFD, the RANS approach could not show visible differences in the in-homogeneity of in-cylinder flow, the LES approach in CFD and the 3D-PIV method showed differences between two cases in a way that the in-homogeneity in the case filling port closed is higher than a maximum valve lift of 4.8 mm for both valves.

This work presented here has been submitted to the SAE ICE conference in September 2009.

References

[1] D. Adolph, R. Rezaei, S. Pischinger, P. Adomeit, T. Körfer, A. Kolbeck, M. Lamping, D. Tatur, D. Tomazic, Gas exchange optimization and the effect on Emission reduction for HSDI diesel engines, SAE Paper 2009-01-0653 (2009).

[2] J. Smagorinsky, General Circulation experiments with the primitive equations I. The basic experiment, Monthly Weather Review, **91**(3),99-164 (1963).

[3] B.E. Launder, D.B. Spalding. The numerical computation of turbulent flow. Computer Methods in Applied Mechanics and Engineering 3,269-289 (1972)

[4] P. Adomeit, S. Pischinger, M. Becker, H. Rohs, A. Greis, G. Grünefeld, Potential Soot and CO Reduction for HSDI Diesel Combustion Systems, SAE Paper 2006-01-1417 (2006)