HSDI Diesel Engines - Gas Exchange Optimization and the Impact on Reducing Emission

ABSTRACT

All future powertrain developments must consider the primary tasks of achieving the required emission levels and CO2-values, while still providing comfort, good drivability, high reliability and affordable costs. One method for improving fuel economy in passenger vehicles that is beginning to be examined is the incorporation of downsizing.

To achieve the levels of engine performance that will be required, increasing the boost pressure and/or rated speed is mandatory. When the boost pressure or the rated speed is increased, the result is an increase in the mass flow rate through the intake and exhaust ports and valves. Considering the impact of these changes, the port layout of the system has to be reanalyzed.

The primary purpose of this study is to examine these effects on the port layout on a modern diesel combustion system and to present a promising concept. The study included flow measurements, PIV measurements of intake flow, CFD simulations of the flow field during intake and results from the thermodynamic test bench. One of the key aspects that needed to be examined was to demonstrate flow quality's impact on combustion.

A new port concept was developed as a result of this study that utilizes a variable valve lift to provide a highly variable swirl. This design enables a homogenous flow field in the cylinder, with excellent flow coefficient. The design also allows an increase in volumetric efficiency combined with a reduction in flow losses.

INTRODUCTION

The prerequisites for advanced concepts of High Speed Direct Injection (HSDI) Diesel Engines require that due to the rapidly increasing price of crude oil they must improve fuel economy and the heat-trapping ability of carbon dioxide emissions. One method of reducing fuel consumption is through downsizing the displacement of the engine [1]. However, to maintain the same level of performance, the specific power and torque of the engine has to be increased. In addition, the load during the test cycle must also be increased.

No matter what from of exhaust gas after treatment technology is selected, a further reduction in the raw emissions of the engine is required. To achieve this target in combination with downsizing, the complete gas exchange system has to be optimized to meet the new boundary conditions, which in parallel would lead to lower fuel consumption.

If the engine is to meet the demands for increased performance, decreased pollution emissions and increased efficiency, the entire engine needs to be optimized. The layout of the intake ports is one important issue, because an increase in specific power will also increase the mass flow rate through the intake and exhaust ports. Reducing the gas exchange work can be accomplished through improving the efficiency of the turbocharger and also through improvements to the intake and exhaust ports that will reduce flow losses and increases the volumetric efficiency of the engine. Conversely, the charge motion has a significant influence on soot formation, soot oxidation and combustion velocity [3],[4]. Previous investigations have [5] shown that the swirl number is not a sufficient index. The uniformity of swirl greatly influences the combustion process and soot oxidation as well.

The following study highlights and analyzes a new concept for intake and exhaust port design that is designed to meet these future challenges.

CONCEPT AND LAYOUT OF INTAKE PORTS

DEFINITION OF OBJECTIVES AND LIMITATIONS

Advanced diesel engine combustion systems require the highest and coldest possible filling in order to solve the seemingly conflicting demand for lean mixture during burning, as well as low oxygen content to prevent NOx formation [2]. Furthermore, combustion must be supported through charge motion, which is realized through swirl motion in advanced direct injection diesel engines. The primary goal when designing the intake and exhaust port geometry is to achieve the lowest flow losses possible and ensure a defined and stable charge motion. However, the relationship between the resulting angular velocity and the relative velocity between injection spray and air is approximately linear to the mean piston when no additional measures are taken. If an adequate angular velocity of the filling has to be achieved for low engine speeds, the concept must offer the possibility to increase the swirl.

This results in the following objectives for the port design:

- Maximum flow in the intake and exhaust ports
- Adequate charge motion
- Variable charge motion

On the one hand, the degrees of freedom for the port configuration are limited due to design constraints, such as:

- Height of the cylinder head
- Space required for injector and glow plug
- Ensuring adequate cooling
- Achieving adequate stiffness/peak pressure capability of cylinder head

The concept introduced in the following is optimized to these demands and boundary conditions.

SWIRL GENERATION IN DIESEL ENGINES

The intake ports of DI diesel engines can be subdivided into three groups: Tangential ports, helical ports, and filling ports.[7] Tangential ports run tangentially at a relatively flat angle into the combustion chamber and thus generate a charge motion around the vertical axis of the cylinder. Tangential ports can be used to generate a relatively high swirl with moderate flow. Using two tangential ports in a 4-V engine is only feasible with turned valve arrangement.

Helical ports feature a geometry that lets part of the main air flow rotate around the valve stem and thus forces the majority of the drawn in gas mass into the combustion chamber in the direction of the desired swirl. For a parallel valve arrangement, it makes sense to combine a tangential port with a swirl duct.

Intake ports with the primary goal of representing the lowest flow resistance possible are referred to as filling ports. For this purpose, the port is guided as straight as possible from the top onto the valve.

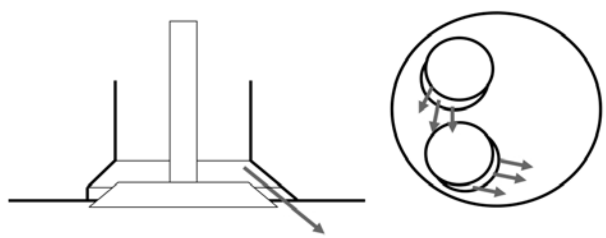

Figure 1: Seat Swirl Chamfer

In addition to the options described above for generating a swirl flow with the help of duct geometry, the flow of the air can be forced to rotate around the vertical axis by eccentrically machining the valve seat rings. One side is covered here while the valve seats are retracted so that the air at low valve lifts mainly flows in on the opposite side. This concept makes it possible to achieve extremely high swirl numbers with low valve lifts without reducing the flow for high valve lifts. Two additional aspects speak for the use of seat swirl chamfers: The chamfer is machined together with the valve seats and is thus subject to significantly reduced tolerances during production. The variation from cylinder to cylinder, or engine to engine in serial production, can be reduced considerably. Secondly, valve pockets in the piston can be eliminated due to the valve recess.

PORT CONCEPT

Since the design goal for the concept presented here is to achieve a high flow rate with moderate swirl, a parallel arrangement of the valves is preferred, since it not only promotes high flow, but also permits a central and vertical injector position when two overhead camshafts are used.

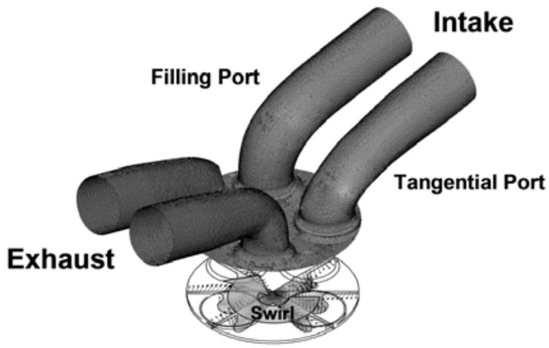

Figure 2: Port Concept

With parallel valve arrangement, it makes sense to design one port as a tangential port. Straight port geometry (Filling Port) was selected for the second one corresponding to the request for excellent flow, which in itself does not generate a directed charge motion [7]. Both intake valves were fitted with seat swirl chamfers that are used to adjust the integral swirl over the intake event to the desired level of approximately 1,3. The layout of the exhaust ports is optimized to reduce the flow resistance, leading to a reduction in gas exchange work and the share of residual gas. This, in turn, leads to lower fuel consumption and increased air efficiency.

VARIABLE CHARGE MOTION

In order to facilitate a swirl increase at low speeds with this configuration, the "state-of-the-art" solution would be to close the filling port with a swirl flap. This method is also observed in the following investigations in each case as a reference for the innovative method presented in the following.

The use of seat swirl chamfers on both intake valves ensures a very high charge motion at low valve lifts. The concept presented here makes use of this property by flattening the valve lift curve over the entire intake process to increase swirl. The swirl ratio plotted in Figure 3 and Figure 4 is based on measurements conducted at the FEV flow test bench. Swirl ratio is defined here as the ratio of tangential velocity to axial velocity.

$$Swirl\,Ratio = \frac{c_u\ (Tangential Velocity)}{c_a\ (axial Velocity)}$$

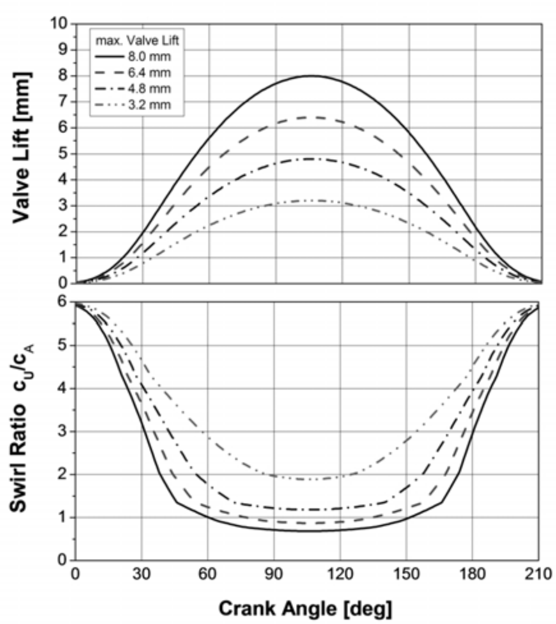

Figure 3: Principle of Variable Swirl

The upper part of Figure 3 shows the standard valve lift curve with a maximum valve lift of 8 mm as a continuous line. The other curves show scaled/reduced valve lift curves. The lower part shows the stationary swirl, which is dependent on valve lift. Thereby, the increase of swirl with the reduced valve lift curve is obvious. The figure describes the principle of variable swirl by varying valve lift. The impact of reduced valve lift on charge motion will be analyzed in the next chapter utilizing PIV measurements on a flow box and CFD calculations.

A reduction of the valve lift has, of course, a negative effect on the flow coefficient. However, in most cases the swirl increase is required at low speeds, and this is where the gas exchange cycle losses react with less sensitivity to a low flow rate. Advantages can be expected in comparison to the intake port shutoff, since a reduction of the valve lift by half for the most part does not lead to a reduction of the flow coefficient by half. On the contrary, closing the filling port reduces the flow coefficient by more than half. This is also confirmed in the following study with the help of simulations and measurement results.

CHARGE MOTION ANALYSIS

In this chapter, the charge motion is analyzed and assessed. For this purpose, flow, swirl, and PIV measurements made on a stationary flow test bench and CFD simulations are presented and analyzed.

FLOW MEASUREMENT

Figure 4 shows the results of the flow measurements performed on a stationary flow test rig in comparison to concepts without seat swirl chamfer. The flow coefficient α_K is defined as the quotient of the isentropic opening area related to the piston area.

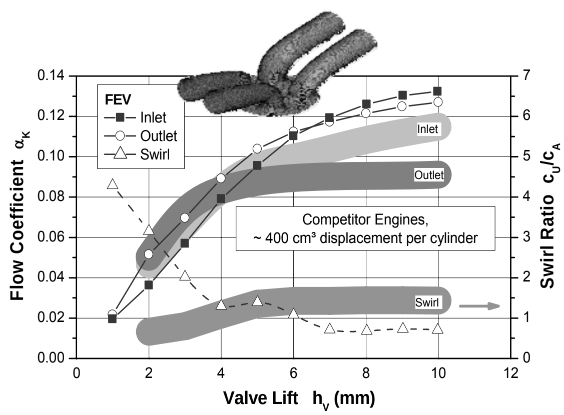

Figure 4: Flow Coefficient and Swirl Number

At low valve lifts a super-elevation of the swirl number as expected can be observed. When looking at the flow coefficient we can see that not only the intake ports but also the exhaust ports achieve excellent flow rates. To assess the flow characteristic of the ports, the small bore of the engine has to be considered. The bore defines the space for valves, injector and glow plug. Due to the fact that size of injector and glow plug is independent of bore, the remaining space for the valves is lower.

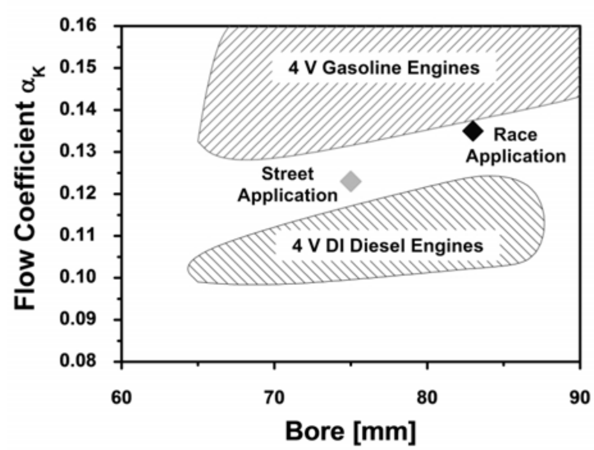

Figure 5: Flow coefficient over bore diameter

Figure 5 illustrates the FEV port concept in comparison to the scatterband of standard gasoline and diesel engines, which were measured at FEV Motorentechnik. Additionally, a high power application with a comparable port concept but a larger bore is shown. We can definitely see that the rates of both applications are clearly above the scatterband of diesel engines, and that the design goal of obtaining a very good flow rate has been achieved.

PIV MEASUREMENTS ON FLOW BENCH

The quality of the swirl generation by the intake port design is analysed by Particle Image Velocimetry (PIV) on a 3D-PIV flow test bench. The 3D-PIV flow test enables to measure the flow distribution in planar sections through the transparent liner tube. The results provide insight into the details of the swirl or tumble flow pattern as well as on secondary flow patterns. Furthermore, flow fluctuations can be recorded by sequences of repeated measurement of all velocity components in the PIV plane.

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

The first analysis examined the flow field in a horizontal section 40 mm below cylinder head. The flow fields are compared for two valve lifts, 1.6 mm with both intake parts active, and 3.2 mm with deactivation of the filling port. In both cases the intake flow rates are similar, but the resulting swirl flow patterns are strongly different. Due to the flow guidance of the swirl seat chamfers an intense and fairly homogeneous swirl flow pattern is obtained in the PIV plane at z = 37.5 mm. For the filling port deactivation, the swirl flow structure is less coherent, and the maximum swirl velocity components are observed downstream of the valve of the tangential port.

The flow pattern at a larger valve lift of 3.2 for dual port opening and 6.4 mm for filling port deactivation was examined next. The flow fields were compared in a horizontal section 75 mm below the cylinder head. For both cases the swirl is developed in this section, but it was also seen that the swirl intensity is higher in the case of dual ports with reduced valve lift. Since the intake flow rates are similar, the dual port activation with reduced valve lift configuration improves the trade-off between flow and swirl intensity. Additionally, for dual port activation with reduced lift, the velocity fluctuations are on a significant lower level, revealing that the swirl flow field is more stable than with filling port deactivation.

NUMERICAL STUDY OF GAS EXCHANGE AND TURBULENT FLOW STRUCTURE

Computational Fluid Dynamics (CFD) is nowadays a powerful tool for simulation and analyzing of the in-cylinder flow and combustion. It reduces the time and the costs for design and optimization of a combustion system. One of the most important parts of the in-cylinder modeling is gas exchange and flow modeling.

Reynolds-averaged Navier-Stokes (RANS) based turbulence models like k-ε, which are widely used in industry, are sometimes incapable to reproduce complex 3D turbulent flows, especially unsteady flows in internal combustion engines. The highest degree of accuracy for resolving complete turbulence is offered by the Direct Numerical Simulation (DNS) method. DNS represents the real solutions of the Navier-Stokes equations, meaning that all length- and time scales are resolved. Unfortunately, its industrial application is not possible in the foreseeable future.

Although the majority of the past numerical studies in internal combustion (IC) engines have employed Reynolds-Averaged Navier-Stokes (RANS) equations, it is well known that the LES model has better analysis capability for in-cylinder simulation in terms of sensitivity to design parameters like port- or bowl design. In the LES approach, the resolved part of turbulent velocity, which is larger than the filter size, is simulated without models, much like direct numerical simulation (DNS), and the unresolved part of turbulent velocity is modeled using the sub-grid scale (SGS) model.

Based on the model of W. Willems [6], the Very Large Eddy Simulation (VLES) used in this study is distinguished from Large Eddy Simulations, which uses a constant filter length of 1 mm and also k-ε model for sub-grid scale modeling. This VLES model also uses the modeling of the law-of-the-wall from the RANS model.

In this study, the intake- and compression strokes are simulated in STAR-CD software with a VLES model instead of k-ε, in order to use the better analysis capability of this model in terms of predicting the non-homogeneity of the in-cylinder flow.

A complete mesh consisting of intake- and exhaust ports, piston and cylinder head is generated using a commercial mesh generating tool and imported into the STAR-CD software. The first numerical investigation is to study the effect of a swirl chamfer with different strategies in comparison with a case using tangential and helical port with 8 mm valve lift. Figure 6 shows how the swirl ratio for the new concept with a swirl chamfer can be increased by using different maximum valve lifts. For port deactivation, both ports have a maximum valve lift of 8.0 mm, but the filling port is closed. For the valve deactivation case, the valve of the filling port remains closed during the intake stroke. Using port- or valve deactivation can increase the swirl ratio in comparison to just using 8.0-8.0 mm lift for both valves and even the swirl ratio is about the case using 4.8-4.8 mm valve lift. But as experimental results show, the swirl ratio is not the dominant parameter for better combustion and emissions reduction.

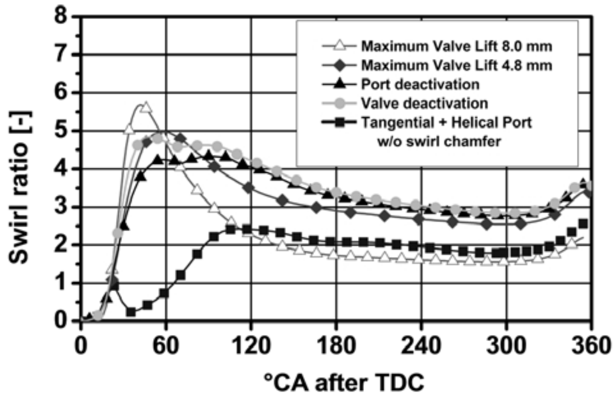

Figure 6: Swirl Ratio over CA

TURBULENT FLOW STRUCTURE AND ITS NON-HOMOGENEITY

A clear understanding of the generation and the spatial distribution of pre-TDC in-cylinder flow and its non-homogeneity is very important for the development of practical combustion systems. A major objective of this CFD study is to investigate the effect of different valve strategies like port- and valve deactivation on non-homogeneity of the in-cylinder flow field.

After start of injection, the interaction between the fuel jet and air plays the most important role for fuel evaporation, combustion and emission formation. Since the introduction of modern high pressure fuel injection systems with vertical injector position, it can be assumed, that the fuel introduction at each spray hole is nearly identical, so that, for optimal mixing, also the air side has to be as homogenous and regular as possible.

It is to be expected that all fuel jets should have the same opportunity of interaction with air. This means that the in-cylinder flow should ideally be symmetric. Otherwise in a non-homogeneous flow field, one jet would have very good air utilization while another jet would have poor air utilization. This leads to a high amount of soot emission formation in comparison with a homogeneous flow field, although both may have about the same swirl ratio.

In order to quantify the non-homogeneity of in-cylinder flow field, at different distances from cylinder head deck surface, cross-sections parallel to the cylinder head are considered. For each cross-section, there exist three velocity components, for each of which the mean value and RMS (Root Mean Square) of deviation are calculated. As Figure 7 shows, the considered cross-section is divided into concentric rings. For example, for the cross-section in Figure 7, five rings are considered. The radial distance between the considered rings is 4 mm in this study.

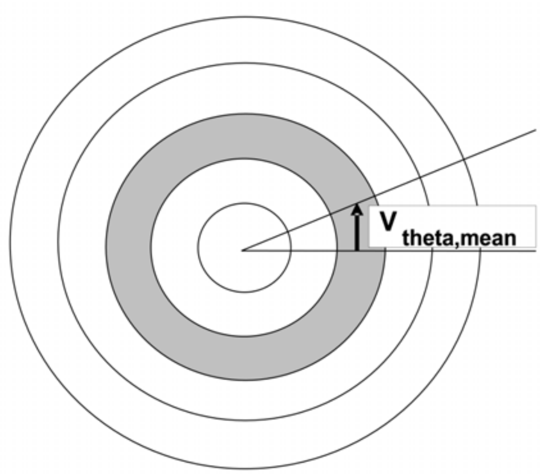

Figure 7: Top View of a Cut Section Considering Five Rings

For each ring, first a mean value of the velocity component is calculated. Equation 1 exemplarily shows the calculation of the tangential velocity on a ring.

$$v_{theta,mean} = \frac{\sum_{n=1}^{N_{cell}} m_n v_{theta,n}}{\sum_{n=1}^{N_{cell}} m_n} \qquad (1)$$

Where N_{cell} is the total number of cells on the ring. Equation 2 shows how for example the RMS of tangential velocity on a ring is calculated.

$$RMS_{V_{theta,ring}} = \sqrt{\frac{\sum_{n=1}^{N_{cell}} m_n (v_{theta,n} - v_{theta,mean})^2}{\sum_{n=1}^{N_{cell}} m_n}} \qquad (2)$$

Where m_n is mass; and $V_{theta,n}$ is the tangential velocity of cell number n.

Finally as equation 3 shows, the separately calculated RMS values of the rings are averaged to determine the RMS value of the whole cross-section.

$$RMS_{V_{theta,cut-section}} = \frac{\sum_{r=1}^{N_{Ring}} A_r RMS_{V_{theta,ring}}}{\sum_{r=1}^{N_{Ring}} A_r} \qquad (3)$$

Where A_r is the area of each ring; and N_{Ring} is the total number of rings on the section plane.

This method for calculating the RMS value of each velocity component has been newly developed and used for this project. In order to find the order of magnitude and also to build-up a scatterband from the RMS values, different port and swirl chamfer geometries should be calculated using this method for comparison of the different schemes. These calculated values are not comparable to those measured from PIV in the stationary flow test bench.

The study showed and was additionally verified by the RMS tangential velocity, that the case with 4.8 mm valve lift is more homogeneous than those using port and valve deactivation.

Another velocity component is the axial velocity of the flow field which is plotted for the section plane in the middle of the bowl for three strategies at 30° and 10° bTDC respectively in Figure8.

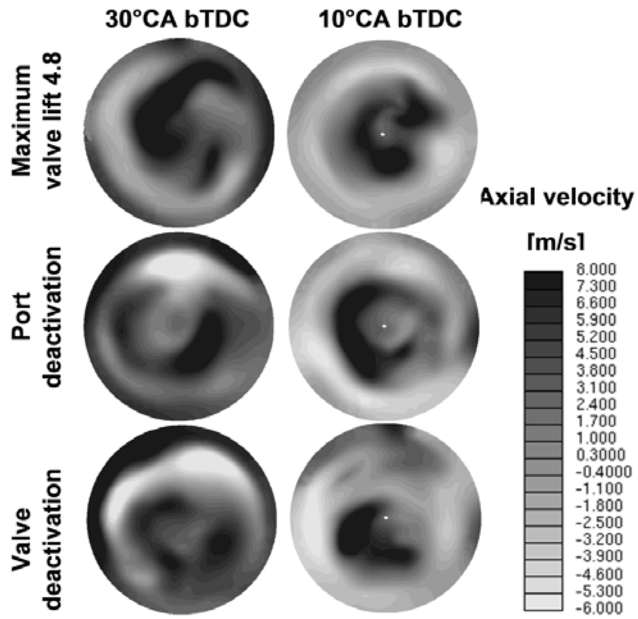

Figure 8: Axial Velocity in Middle of Bowl

The calculation of the RMS value of axial velocity verifies that the case with 4.8 mm valve lift is more homogeneous than those with port or valve deactivation, Figure9.

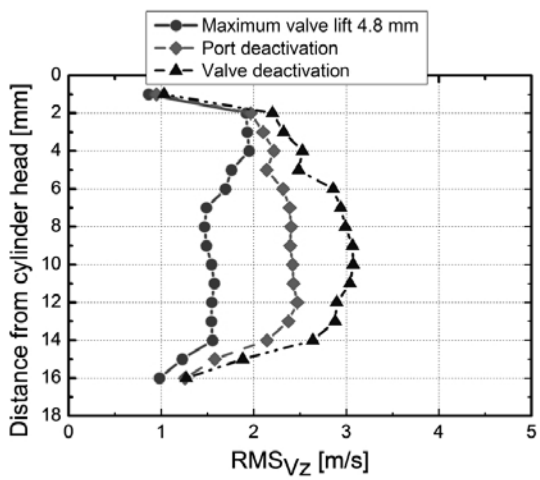

Figure 9: RMS Value of Axial Velocity at 10°CA bTDC

The study clearly showed that using port or valve deactivation strategies increase the non-homogeneity of the in-cylinder flow field in comparison with using maximum valve lift of 4.8 mm for both valves. This is also verified by the calculation of the RMS value of tangential and axial velocities. In fact using port or valve deactivation can increase the swirl ratio in the 4.8-4.8 mm case but it leads simultaneously to a non-homogeneous flow field, which has a great effect on the combustion.

IMPACT ON THE COMBUSTION SYSTEM

In this section, the impact of the flow quality on the combustion system is assessed. Tests made with a single-cylinder engine are used as a basis for the investigations.

TEST OBJECTIVE

The results presented here were obtained using a fully automated test bench. The engine was supercharged with an external boosting unit, which allowed setting the boost pressure independent of operation point and exhausting back pressure. The exhaust back pressure was set by a throttle in the exhaust system. The test engine used had the following properties:

- Basic engine: FEV system engine
- Bore x stroke: 75 mm x 88.3 mm
- Firing peak pressure: 250 bar
- Fuel injection system: HPCR 2000 bar Piezo
- Compression ratio: 15.3
- Nozzle: 8 x 153°
- Hydraulic Flow Rate: 620 cm³/min

The configuration of the cylinder head corresponds to the layout with retracted valves described in the previous information. This allows a piston design without any valve pockets.

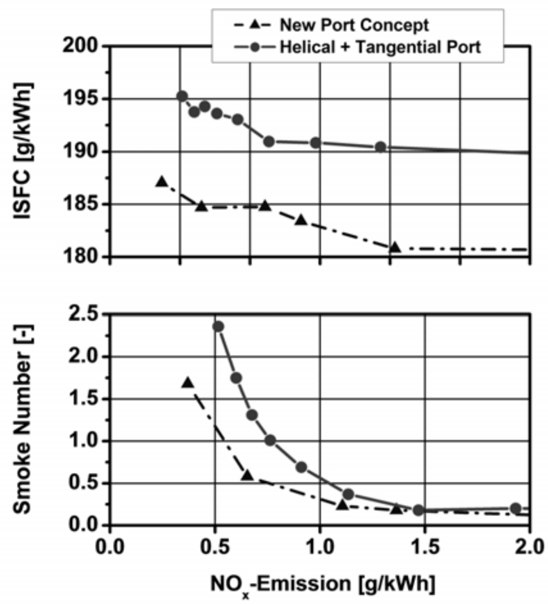

Figure 10: EGR-Variation, n = 2000 rpm, IMEP = 14.8 bar, Impact of New Port Concept

Figure 10 shows a comparison of EGR sweeps between the optimized port layout and a conventional cylinder head design at an engine speed of 2000 rpm and an IMEP of 14.8 bar. The investigations were conducted on the same engine, with the identical boundary conditions and calibration. The conventional engine has a port design with one helical and one tangential port, comparable integrated swirl number and no valve recess, but valve pockets in the piston. The compression ratio, the valve lift curves and the valve timing are identical. The results prove that the new concept is capable of improving emission behavior and reducing fuel consumption due to lower gas exchange loss, higher filling and improved combustion.

The following information shows the results and analysis of the different swirl levels for this significantly improved combustion behavior with the new port concept design.

The variable valve lift was achieved by replacing camshafts with different cam profiles than those designed in the concept phase. The four valve lift curves shown in **Figure 3** used different maximum valve lifts (8 mm, 6.4 mm, 4.8 mm, 3.2 mm), but identical event lengths.

The partial load investigations were performed in four load points:

- Engine speed 1500 rpm, IMEP 4.3 bar
- Engine speed 1500 rpm, IMEP 6.8 bar
- Engine speed 2280 rpm, IMEP 9.4 bar
- Engine speed 2400 rpm, IMEP 14.8 bar

The first three points are within the operating range of current diesel passenger cars during FTP 75 and NEDC; a fourth load point was selected to assess the potential of the system at high load, taking into account further downsizing and emission compliance in the whole engine map area. The curves shown in the individual load points are in each case variations of exhaust gas recirculation rate with identical injection and charging parameters.

	2280 rpm, 9.4 bar	2400 rpm, 14.8 bar	1500 rpm, 6.8 bar
Boost Pressure [mbar]	2275	2500	1500
Exhaust Pressure [mbar]	2375	2650	1600
Rail Pressure [bar]	970	1700	900
BOIMain [°CA BTDC]	0.6	3	5.5
BOIPilot [°CA BTDC]	14.6	35	16.5

The specific values shown relate always to the indicated power.

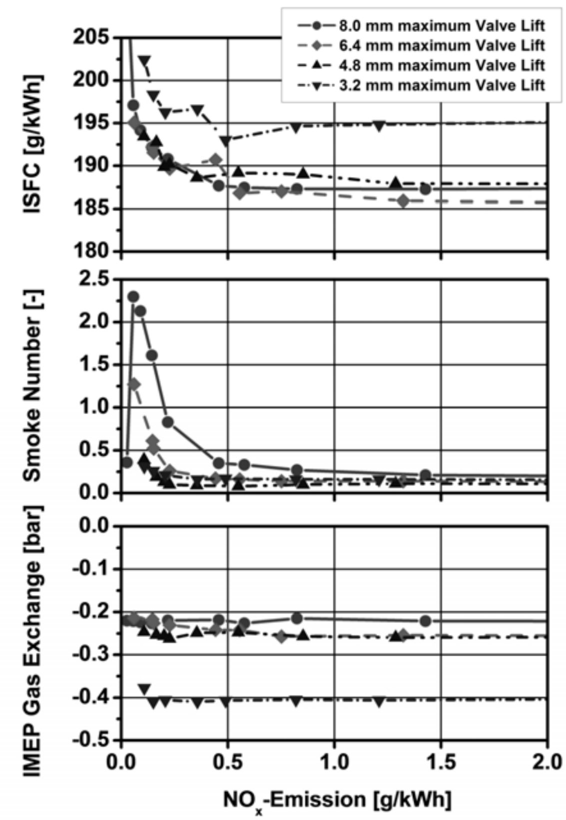

Figure 11: EGR-Variation, n = 1500 rpm, IMEP = 6.8 bar

Low engine speed leads to low charge motion. Thus an increase of swirl usually provides best potential to improve the emission behavior. As shown in Figure 11, the utilization of the increased homogeneous swirl by reducing the valve lift in the load point at 1500 rpm, IMEP 6.8 bar reduces the smoke emission significantly, with lowest smoke emission and without any impact on the fuel consumption for 4.8 mm valve lift. A further reduction of valve lift leads to noteworthy increased gas exchange losses, which finally leads to increased fuel consumption without any advantage concerning smoke emission. So in this load point a maximum valve lift of 4.8 mm is the best compromise.

The CO emissions and the smoke number over the indicated specific NO_x emissions for the load point 2400 rpm, 14.8 bar were also studied. As shown in the previous chapters, a reduction of the valve lift to 4.8 mm and closing the filling port leads to a comparable swirl increase

While the swirl increase through the valve lift reduction leads to considerably reduced CO emissions with minor smoke number improvement, the particulate emissions are rising significantly when using the "classical" intake port shutoff, whereas CO emissions are not increased.

Figure 2 shows the indicated efficiency and the gas exchange losses for the three variants. The expected increase in gas exchange work for the variants with increased swirl can be observed. The measurement also confirms that the increase for the variant with the closed port is about twice as high.

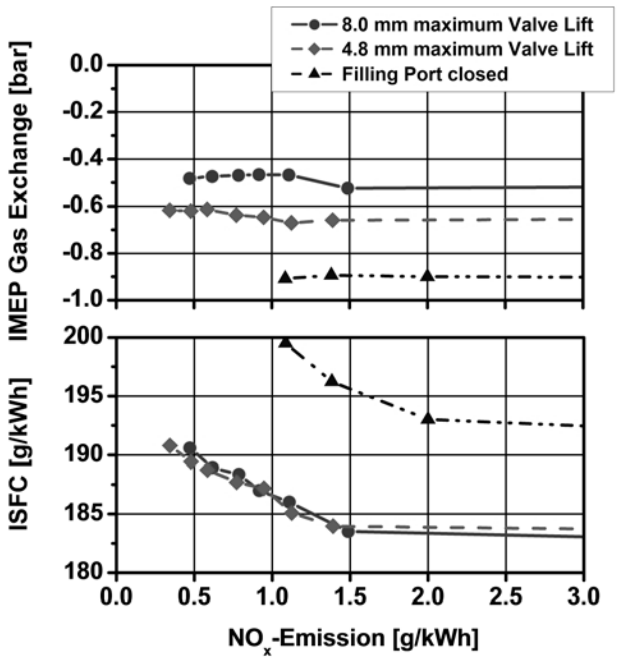

Figure 12: EGR-Variation n = 2400 rpm, IMEP = 14.8 bar

This can be used to explain, in part, the significant fuel consumption increase when the filling port is closed. However, Figure 13 shows that also the efficiency of the high-pressure cycle is considerably lower due to a delayed heat release.

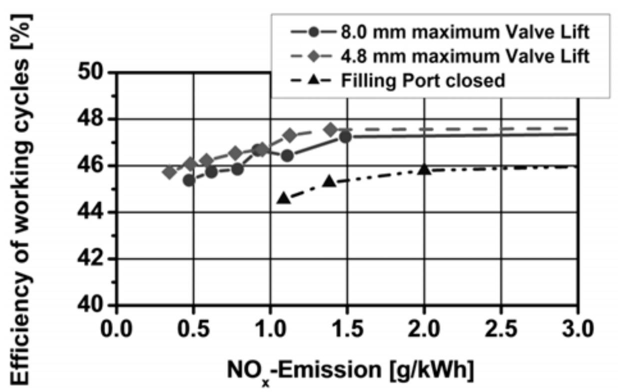

Figure 13: EGR-Variation n = 2400 rpm, IMEP = 14.8 bar

By contrast, the increased charge cycle loss can be compensated through a more effective combustion in the variant with reduced valve lift.

In order to rule out that the increase in smoke number during intake port shutoff is solely the result of low cylinder filling, Figure 14 shows the smoke emissions for the three variants plotted over air/fuel ratio. We can see here that the combustion system can generally convert the higher swirl into lower black smoke emissions. However, this is only true for the variant with reduced valve lift.

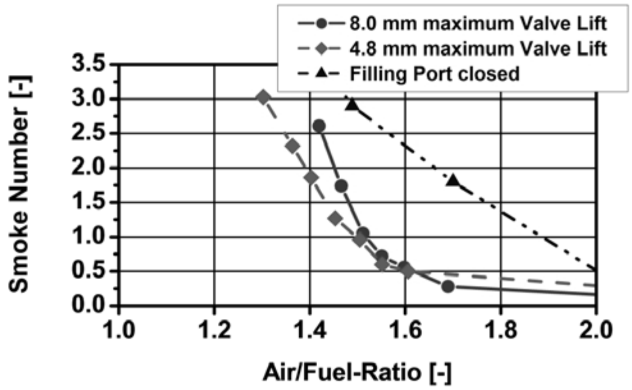

Figure14: EGR-Variation n = 2400 rpm, IMEP = 14.8 bar

When taking a look at load point 2280 rpm, IMEP 9.4 bar, the same effects can be observed as discussed and analyzed before. The particulate emissions for this load point also increase substantially when swirl is increased by closing the filling port.

Figure 15 represents two of the EGR sweeps that were plotted, based on the load point for 2280 rpm, IMEP 9.4 bar. It shows the smoke emissions for the reduced valve lift and intake port shutoff plotted over the nitrogen oxide emissions and the air/fuel Ratio. To be able to eliminate the influence of the different combustion chamber filling, additional measurements were made at this load point. The star represents a measurement with reduced valve lift, where the boost pressure was reduced (-75 mbar) until the filling corresponded to the value of the measurement with closed filling port. The exhaust back pressure was kept at a constant level. In this test point the load was not kept constant, but the injected fuel quantity was fit to that value, which was used with 4.8 mm maximum valve lift at same EGR rate.

This results in identical filling (mass and composition) for intake valve closing. Since the same fuel quantity was injected in both cases, the significant difference in smoke emissions can only be explained through the differences in the flow pattern discussed before.

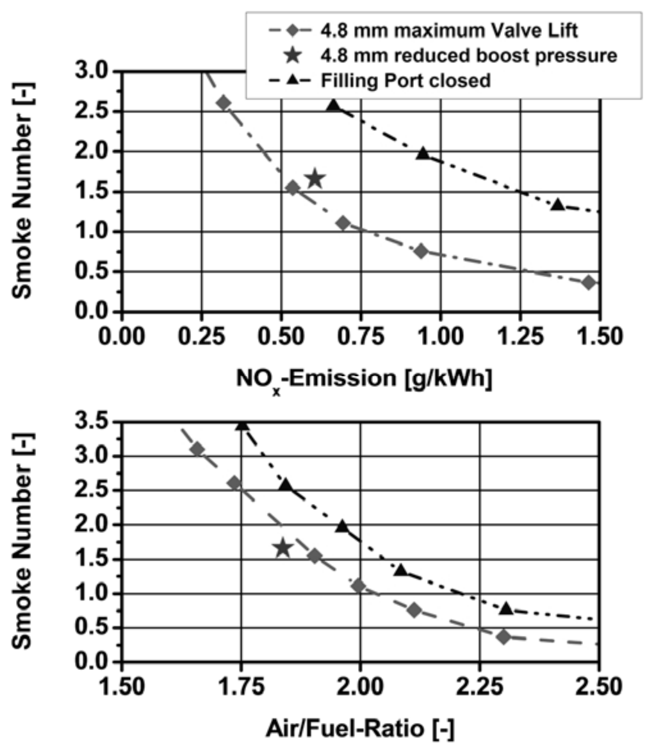

Figure15: EGR Variation n = 2280 rpm, IMEP = 9.4 bar

CONCLUSION

This study introduces a new port concept that shows great promise for direct injection diesel engines. The stationary flow test rig experiments illustrated the excellent flow rates, particularly when considering the small bore, for the cylinder head with tangential, filling port and seat swirl chamfers.

It was proven through stationary PIV measurements that it is possible to create a very stable and homogeneous flow pattern, with the assistance of seat swirl chamfers. Improved results were also noted when reducing the maximum valve lift as opposed to closing the filling port. Specifically, a reduced valve lift produces improved homogeneity of the charge motion even after compression, based on the subsequent flow patterns provided by CFD simulations.

Soot emissions deteriorated significantly with a less homogenous flow pattern, as shown in the results from the stationary thermodynamic test bench. Reducing the valve lift to increase the swirl provides a slight improvement of the particulate air/fuel ratio trade-off. Using a closed filling port produced considerably higher particulate emissions, even considering that the swirl number is similar to the configuration with reduced swirl.

OUTLOOK

Further studies are being conducted to obtain a better understanding of the complex processes during compression and combustion. An optically accessible version of the engine is being tested at the Institute for Combustion Engines to conduct LII measurements, analyze the soot formation and oxidation in cooperation with the LTFD Aachen. Additionally, PIV measurements are also being conducted on an optically accessible engine to visualize the flow pattern during the intake and compression processes.

ACKNOWLEDGEMENTS

The authors would like to thank their colleagues at FEV Engine Technology GmbH as well as the employees of the Institute for Combustion Engines, RWTH Aachen University, who have not been mentioned here, for their valuable services rendered in their individual technical disciplines, which formed the basis for preparing this article.

The described work has been performed within the project "CO_2-optimierter rußfreier Dieselmotor", supported by the Bundesministerium für Wirtschaft und Technologie (BMWi). The authors like to thank Mr. P.-J. Heinzelmann, BMWi, and Dr. B. Koonen, TÜV Rheinland Group, for supporting the project

REFERENCES

1. Körfer, Th.; Lamping, M.; Kolbeck, A.; Genz, T. Pischinger, S.; Busch, H.; Adolph, D.: The Potential of Downsizing Diesel Engines - Considering Performance and Emissions Challenges IMechE, Internal Combustion Engines, London 11. - 12.12.2007
2. Lamping, M.; Kolbeck, A.; Körfer, Th.; Adolph, D.; Busch, H.; Pischinger, S.: Modernes Dieselbrennverfahren zur Darstellung günstigster Motorrohemissionen bei verbessertem Verbrauchsverhalten, MTZ, 69.Jahrgang Seite 42, Vieweg Verlag, Stuttgart, Deutschland, 2008
3. Fuchs, T.R.; Rutland, C.J.: Intake Flow Effects on Combustion and Emissions in a Diesel Engine, Technical Report 980508, SAE, 1998
4. Miles, Paul C.: The Influence of Swirl on HSDI Diesel Combustion at Moderate Speed and Load, Technical Report 2001-01-1829, SAE, 2001
5. Adomeit, Philipp; Pischinger S.; Becker, M.; Rohs, H.; Greis, A.; Grünefeld, G.: Potential Soot and CO Reduction for HSDI Diesel Combustion Systems, Technical Report 2006-01-1417, SAE, 2006
6. Willems, W., Numerische Simulation turbulenter Scherströmungen mit einem Zwei-Skalen Turbulenzmodell, *Ph. D. thesis*, RWTH Aachen University, 1996.
7. Hermann, Hans Otto: Grundsatzuntersuchungen an einem 4-Ventil direkteinspritzenden PKW-Dieselmotor, Ph. D. thesis, RWTH Aachen University, 1995.

CONTACT

Dipl.-Ing. Dirk Adolph
adolph@vka.rwth-aachen.de
Dr.-Ing. Matthias Lamping
lamping@fev.de