

Potential and Challenges of a 3000 Bar Common-Rail Injection System Considering Engine Behavior and Emission Level

2010-01-1131

Published
04/12/2010Johann A. Wloka, Sebastian Pflaum and Georg Wachtmeister
Technische Univ. München

Copyright © 2010 SAE International

ABSTRACT

The paper presents a study on the influence of increased rail pressure on the flow field of an injector nozzle at pressures of up to 3200 bar. Additionally the influence of nozzle conicity, HE-Rounding and nozzle-hole-diameter on the injector flow was investigated with CFD calculations. In these calculations the real behaviour of the nozzle movement was considered. The nozzles were tested on a special single cylinder research engine. The results showed high potential to fulfill EURO VI emission standards by raising the injection pressure and by optimizing the nozzle geometry, without exhaust aftertreatment measures.

INTRODUCTION

One important method to reduce emissions and simultaneously increase engine output is to optimize the injection process. Direct injection in diesel engines is currently standard and is becoming standard in gasoline engines.

In Diesel-Injection Technology one trend seems to be evident. Over the years injection pressure was increased which improved the atomization process for better combustion and emission characteristics. Additionally a lot of studies have been performed which show the influence of nozzle flow on the atomization process and engine emissions. Studies investigating the effect of nozzle conicity and HE-Rounding, showed that they greatly influence engine characteristics [1].

The injection process itself, especially flow distribution and pressure in the sac-hole, is influenced by the needle movement. To show these affects, a CFD-study was done comparing an original needle movement (1800 bar injection

pressure) with a modified for 3000 bar injection pressure. The modified needle movement was derived from a hydraulic calculation model which was developed especially for the used injectors in these studies.

To validate the CFD data, various nozzles were manufactured and tested on the single cylinder research engine. The exhaust emissions were measured, and the influence of the various nozzles at 3000 bar injection pressure was shown.

COMPUTATIONAL METHOD

NOZZLE INTERNAL FLOW CALCULATION

In order to predict diesel spray behaviour, and as a result the combustion process as well as emissions, the internal injector nozzle flow has to be considered. In this study an Eulerian multi-fluid model was used to calculate the cavitating flow in the nozzle. To calculate the cavitation phenomena, a reduced and linearized cavitation model was used, based on the Rayleigh-Plesset equation, [2]:

$$R\ddot{R} + \frac{3}{2}\dot{R}^2 = \frac{\Delta p}{\rho_c}$$

In this equation the terms of surface energy and viscosity are neglected as it is assumed that they have insignificant influence. The inertia term $R\ddot{R}$ is not evaluated in this study, because previous calculations showed that the difference in mass flow through the nozzle between the non-linear cavitation model (with inertia term) and the linear cavitation model (without inertia term) is quite small, but the linear cavitation model is numerically more stable, [1]. For each phase (liquid diesel and diesel vapor), mass conservation,

momentum conservation and turbulent kinetic energy (TKE) are solved separately.

The mass exchange through adjacent phase boundaries is calculated with the following equation,

$$\Gamma_c = \rho_d N''' 4\pi R^2 \sqrt{\frac{2}{3} \left(\frac{\Delta p}{\rho_c} \right)} = -\Gamma_d$$

where Γ is the source term for mass exchange, R the bubble diameter and N''' the bubble-density. The momentum exchange of both phases includes drag and turbulent dispersion forces and can be written as

$$M_c = C_D \frac{1}{8} \rho_c A_i''' |v_r| v_r + C_{TD} \rho_c k_c \alpha_d = -M_d$$

The drag coefficient C_D is a function of the bubble Reynolds-Number. A''' is the interfacial area density, which is a function of the bubble density N''' and is obtained from the cavitation mass exchange model. The definitions and mathematical descriptions of all coefficients are attached in abbreviations.

The computational mesh for the nozzle-flow is shown in [Fig. 1](#). The calculation mesh covers only 1/16 and 1/20 of 8 and 10 hole nozzle domains, respectively. Buchholz performed some comparative calculations where he showed that this approach is applicable and therefore the calculation of the whole domain as unnecessary, [4]. In order to simulate the nozzle flow in the best manner, many geometric details have been implemented in the calculation mesh, [Fig. 1](#) right. To get the radius of curvature in the upper as well as lower part of the inlet of the nozzle hole, a special cast was done, [Fig. 1](#) left. The measured radius of curvature was then implemented in calculation mesh. The mesh consists of approx. 370'000 cells. The mesh is modular built to minimize the influence on the calculation result. Therefore the nozzle body and the sac volume are the same in each calculation. Only the nozzle, including the conicity and the radius of curvature was changed. The mesh is block-structured and the cell distribution of the nozzle mesh is held constant for each calculation.

<[figure 1](#) here>

NEEDLE-MOVEMENT AND EMISSION LOWERING STRATEGY

The first calculations were performed with a compressed needle curve of an original 1800 bar rail pressure needle curvature (original configuration). With the original configuration the injection duration takes approx. 1.5 ms, and the needle reaches maximum lift. According to measurements on the test rig at 3000 bar, the duration was shortened to

approx. 0.7 ms. Even though the original needle curve is compressed to a duration of 0.7 ms, the absolute needle displacement and characteristics stay the same. With this needle movement the characteristics in nozzle flow are not in accordance with reality because the needle movement influences the flow in the nozzle. For further calculations a more realistic needle curve was implemented. Due to the fact that no needle moving sensor was installed in the injector, the displacement curve has to be calculated. The calculation is done with a neural network based calculation tool which takes the mechanical and hydraulic side of the injector into account and also considers the solenoids influence, [5]. In [Fig. 3](#) the needle displacement curve for 3000 bar and a nozzle diameter of 172 μm is shown in contrast to the needle displacement for 1800 bar and 192 μm .

<[figure 2](#) here>

Both curves are shown over standardized time for better comparison. In the case of the needle movement with 1800 bar, one can see that the needle reaches maximum lift and remains open for a significant amount of time.

In terms of the 3000 bar configuration (stationary engine loads at 1200 rpm, 14.5 bar IMEP) the needle does not reach the maximum possible lift of 0.35 mm. It is evident to the eye that the needle opens and closes directly afterwards. There is no holding phase where the needle movement is stationary and thus the internal nozzle flow.

The initial calculations with the original needle displacement curve were compared with the calculations of the adapted and calculated needle curve for high pressures. [Figure 3](#) shows with constant injection pressure and nozzle diameter at 50°C standard time after SOI, the influence of the needle movement on the velocity.



Figure 3. Influence of needle movement on nozzle-velocity

Due to the results shown above, it is obvious that for further calculations the 3000 bar needle movement has to be considered.

At the start and at the end of the calculation the needle is closed. Due to the fact that the solver (AVL FIRE v2009, [3]) does not handle meshes with mesh size zero the initial needle gap was set to 17 μm .

The analyzed nozzles are VCO nozzles used in heavy duty truck engines. The original nozzle configuration and the calculated and analyzed new nozzle configurations are shown in Table 1.

Table 1. Details of manufactured nozzles

Nozzle - Number	Conicity - Faktor k	HE [%]	D _{out} [μm]	Hole Numbers
1 (Original)	1,5 (real 0)	8,8	198	8
2	1,75	13	176	8
3	2	13	175	8
4	2	25	172	8
5	2	25	151	10

The calculation of the new nozzle configuration was done by Bernoulli approximation which assumes that in the stationary flow field no or minimum cavitation will occur, [6,7]. To reach no cavitation in the nozzle the k- factor was set high, k = 2. The definition of the k-factor is shown below:

$$k = \frac{D_{\text{in}}[\mu\text{m}] - D_{\text{out}}[\mu\text{m}]}{10}$$

This definition is according to Blessing, [8]. Blessing showed that with high k-factors, geometrical induced cavitation in the nozzle holes can be inhibited. According to Pauer's research work, in which he showed that a positive k-factor increases the spray's momentum, [9], lies the reason why high k-factors are used. Consequently the air entrainment is higher and the mixing process faster. This is also possible to determine through Wakuris equation of the air-entrainment ratio, [11]:

$$\frac{G_a}{G_f} = \sqrt{t \cdot \tan \theta/2 \cdot \frac{1}{\rho_f \cdot d}} \cdot \sqrt[4]{\frac{\rho_a \cdot \Delta p}{C}}$$

The drawn in air mass into the spray can be expressed by the Naber and Sieber's equation, [10]:

$$\dot{m}_{\text{air}} \propto \sqrt{\rho_a \cdot \rho_f} \cdot d \cdot x \cdot v_f \cdot \tan \theta/2$$

A higher rail pressure and a smaller outlet diameter of the nozzle hole causes a higher air/fuel entrainment ratio (G_a/G_f) and hence better mixture formation. The inner nozzle flow is not regarded in this equation, though. Generally a higher conicity and a high HE-Rounding causes a higher spray momentum and therefore faster air/fuel mixture preparation. In this study, nozzles with high conicity and high HE-Roundings are used subsequently intensifying the air/fuel entrainment, thereby decreasing particulate emissions. When

regarding a LII-Image of a diesel spray by Crua [12], one can see where soot emissions are formed:

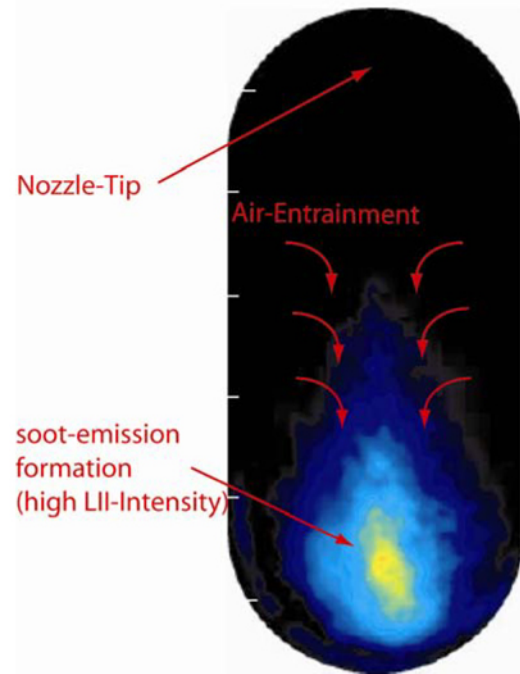


Fig. 4. LII-Image of diesel spray

These measures will, of course, negatively affect the NO_x emissions, but with a high EGR rate it is possible to hold the NO_x emissions on a constant level and mainly concentrate on reducing particulate emission with high pressure injection and modified nozzle configurations.

With a higher air entrainment into a diesel fuel spray the emission of soot can be reduced. Cavitation is a vital influential factor here, which on the one hand can improve mixture formation and its intensity; while on the other hand leads to an increase of the spray angle and reduces the sprays momentum. Measurements of spray momentum were carried out by [9] and he was able to show a strong dependency between the nozzle k-factor and the spray's momentum. Furthermore, it did not only affect the momentum but also the momentum gradient. Compared to a cylindrical nozzle (k = 0), a nozzle with k = 2 has a higher spray momentum, thus a higher mixing intensity, although cavitation, which is supposed to increase the mixture formation is suppressed. The higher mixture formations intensity is a result of a slight spray with high momentum. Not only can a high conicity suppress cavitation, but also a high degree of HE-Rounding. The HE-Rounding leads to a high radius of curvature in the upper entry to the nozzle hole. According to Acroumanis [13], the deflection of the flow is smoother and geometrical cavitation can be suppressed or sometimes even be totally avoided. The effect on the spray is the same like high positive

k-Factors, more momentum in the spray and subsequently increased mixing intensity.

The momentum of a spray also influences the combustion process of the spray itself. A slight spray with high momentum will combust further away from the nozzle tip. With a nozzle which has high cavitation and subsequently lower spray momentum, the combustion process will take place closer to the nozzle tip. Due to geometrical circumstances around the nozzle tip area less air is available. Combustion with less air will automatically be a source for soot, which is not intended to be formed. Therefore the decision was made to examine nozzles with suppressed cavitation and improve the mixing formation due to higher spray momentum.

Another reason for using high k-factor nozzles with high HE-Rounding is that the spray pattern becomes narrower allowing the design of nozzles with more than 8 bores (original version) as no negative interaction between the sprays will occur and the mixing is enhanced additionally.

The disadvantage of high mixing intensities is the “hard” combustion process which will lead to high NO_x emissions. Yet, a trade-off is available. The strategy chosen here for NO_x reduction was to use an extremely high EGR rate, what allows to concentrate the nozzle design only for low soot emissions. Based upon these considerations nozzles with a higher k-factor and high HE-Roundings were analyzed in this study.

In the figure below the calculated vapor distribution of different nozzle versions from [table 1](#) are compared. Cavitation often occurs on needle opening and closing, because turbulence is induced into the system and the pressure and velocity distribution in the nozzle enhance the cavitation phenomena. As shown in [figure 2](#) for such high pressures like 3000 bar and higher the needle opens to a certain lift and then closes immediately. There is no stationary phase in which the nozzle flow can establish a stationary distribution. For the reasons described earlier cavitation should be prevented, even in the needle opening- and closing phase, especially for injection pressures of 3000 bar. With a higher degree of HE-Rounding - which increases the upper and lower radius of curvature - a higher k-factor and smaller nozzle diameters, the cavitation in the opening phase can totally be avoided. In the needle closing phase it was not possible to avoid cavitation but it is possible to reduce cavitation consequently.

<[figure 5](#) here>

SINGLE CYLINDER RESEARCH ENGINE

Validating the simulated nozzle flow experimentally is a challenge. The validation tests are performed on a special single cylinder research engine. This diesel engine was designed and built at the department for internal combustion engines (LVK) for the main purpose of research work. The engine specifications are summarized in [Table 2](#).

Table 2. Technical Data of the Single Cylinder Research Engine

	Value	Device
Number of cylinders	1	-
Swept volume	1750	cm ³
bore	120	mm
stroke	155	mm
Compression ratio	17	-
Max. cylinder pressure	300	bar
Max. boost pressure	10	bar
EGR-rate	0 - 60	%
Common rail pressure	3200	bar

The single cylinder research engine, allowing no interaction between different cylinders, is equipped with four Lanchester balance shafts, that reduce vertical engine vibrations to a minimum. Horizontal vibrations caused by the dynamic engine torque are almost zeroed by inclined, stiff, rubber engine mounts. The mounts are specifically allocated so that the research engine's centre of rotation is directly where the combustion chamber is, which is vital for later observations when the engine is equipped with an optical accessible combustion chamber.

Beside these mechanical features the engine also has an externally cooled exhaust gas recirculation (EGR) system ([Figure 6](#)). Using the Venturi effect together with two check valves at the mixing point utilizing exhaust pressure pulsation, maximum EGR rates of about 60% are possible. This system shows a positively strong effect on NO_x emission.

To provide enough oxygen partial pressure with the high EGR rates the engine is supercharged with an external compressor. This allows the boost pressure to be controlled exactly between atmospheric pressure and 10 bar. A controllable exhaust throttle substitutes the turbocharger turbine which also sets it's degree of efficiency, accordingly. This system strongly affects particulate emission of the combustion process.

<[figure 6](#) here>