

Influence of in-nozzle flow on spray morphology

Andreas Schmid*¹, Chawki Habchi³, Julien Bohbot³, Beat von Rotz¹, Kai Herrmann²,
Rolf Bombach², German Weisser¹

¹Wärtsilä Switzerland Ltd., CH-8401 Winterthur, Switzerland

²Paul Scherrer Institute, CH-5232 Villigen PSI, Switzerland

³IFP Energies nouvelles, 1 et 4 avenue de Bois-Préau, F-92852 Rueil-Malmaison, France

*Corresponding author: andreas.schmid@wartsila.com

Abstract

The diesel combustion process is strongly dependent on the rate of introduction of fuel and hence the quality of the atomization process, which in turn is significantly influenced by effects caused by the geometry of the fuel injection equipment itself as well as the potential occurrence of cavitation. In particular, the injector geometry of large marine two-stroke diesel engines differs substantially from the configurations used in most other diesel engine applications, as the injector orifices are distributed in a highly non-symmetric fashion. In order to simulate, respectively experimentally assess the impact of key features of such orifice arrangements on spray morphology, a generic nozzle design has been introduced in [1]. This specimen consists of an elongated tip with two orifices: The first one for producing the spray to be observed, the second one significantly far downstream of the first and with a diameter corresponding to the area of the remaining four orifices in a typical production injector, in order to simulate the same flow behaviour up to of the spray orifice. It was extended for the purpose of the present study by adding an insert, thus mimicking the flow conditions inside more recent injector designs. Selected configuration variants corresponding to isolated variations of key design parameters were investigated experimentally as well as by means of CFD. For this purpose, the IFP-C3D code, which includes an advanced cavitation model, has been utilised. The results obtained in those simulations help understanding how fluid-dynamic effects occurring in the injector influence the propagation of sprays. In particular, the observed non-symmetric spray structure can be clearly attributed to high levels of flow inhomogeneity at the exit of the orifice, a feature that is even more pronounced with eccentric arrangements of the orifice and is also to some extent depending on the height of the flow channel leading to the injection hole. For eccentric orifices, the generation of a swirling flow inside the orifice seems to contribute to clearly wider spray angles and sprays from such orifices are moreover characterised by non-negligible deflection from their theoretical axis. It could also be shown that it is essential to properly account for the actual flow conditions inside the injector, as single-spray tests neglecting the flow through the other orifices of a multi-hole injector yielded distinctly different spray behaviour, which would lead to erroneous conclusions when applied the results as is to multi-orifice configurations.

Introduction

The diesel combustion process and emission formation are not only highly dependent on hydro-carbon oxidation kinetics but also strongly influenced by turbulence and two-phase flow effects. In particular, this is related to the rate of introduction of fuel and the quality of the atomization process, which in turn is significantly affected by the geometry of the fuel injection equipment itself as well as the potential occurrence of cavitation. The injection system design and its operational parameters are decisive factors with respect to these processes. Nozzle internal phenomena, their impact on the fuel flow characteristics at the orifice outlet, and the associated effects on the sub-sequent spray formation and combustion processes are far from being fully understood today. As a consequence, they are commonly not accounted for in the spray-related sub-models applied in CFD simulations. An improved understanding of the liquid break-up physics is highly required and the spray boundary conditions at the nozzle exit in particular are of supreme importance for spray modelling purposes.

In order to obtain such better understanding of those influencing factors, Wärtsilä Switzerland Ltd., PSI and the IFPEN worked together in a project focusing on the investigation of in nozzle flows in generic injector configurations. These nozzle designs allow the isolated study of the effect of key design parameters of actual large two-stroke engine injectors on fuel spray characteristics at relevant operating conditions. To this end, extensive computational fluid dynamic (CFD) investigations of the injection system internal flow have been performed, which help assessing the liquid conditions at the nozzle exit and, therefore, the boundary conditions for spray development. In parallel, the same generic injector designs have been applied on the Spray Combustion Chamber (SCC) test facility in order to characterize the associated spray behaviour by means of Mie-scattering measurements.

First investigations on the basis of generic injector configurations have been presented in [1]. Based on the results achieved during this first measurement campaign, a second series of nozzle designs has been devised. All those configurations were specified in order to allow the investigation (by means of both simulation and actual experiments) of the propagation of a single spray as a function of particular design features or parameters. In particular, five clearly distinct injector configurations have been selected from a large variety of possible options for investigating the impact of five key factors on the flow condition at the nozzle exit and the resulting spray characteristics.

Nozzle configurations

Figure 1 explains the injector design presented in the first measurement series [1], with the fuel exiting the observed orifice (diameter d) while the fuel amount of another four orifices is bypassed through an additional orifice with diameter D whereas the area of the bypass-orifice is four times the area of the observed (and partially bypassed) orifice.

The generic nozzle design has been extended by adding an insert in the nozzle bore. This introduced a channel of height s underneath the orifice as can occur in more recent injector designs [2]. Figure 1 also shows the flow volume used for the simulations in comparison to the drawings of the experimental setup. Cut A-A explains the tilt of the channel in case of eccentricity. This was made to ensure a constant channel height. The channel was tilted such that the crossing point between the surface of the nozzle bore and axis of the orifice was located in the middle of the channel.

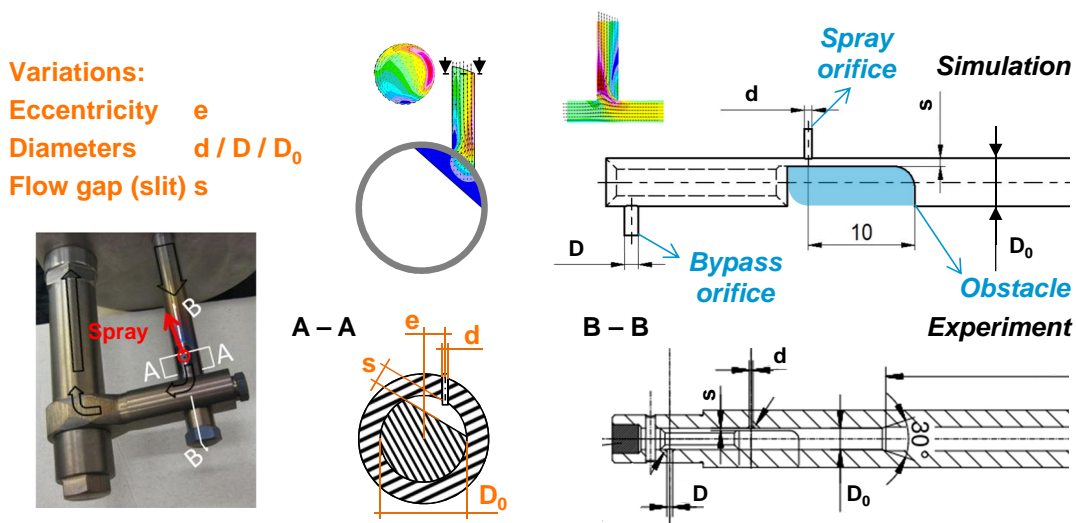


Figure 1. Schematic overview on generic nozzle design features, simulating the conditions of a single orifice within a serial injector

The geometric features of those five configurations – one reference geometry and four variations – are listed in Table 1: The injector configuration 2 (highlighted in blue) represents the reference injector while the four remaining configurations are the variations (highlighted in orange) in the nozzle eccentricity, channel height, nozzle bypass and turbulence generation. They are the configurations number 1, 4, 15 and 34, respectively, the apparently random numbers labelling the different injector configurations arise from a pre-selection of a large number of injector layouts. The four variations presented here were chosen to represent a balanced mixture of academic interest (#15 without bypass and #34 the swirl chamber) and industrial applications (#1 thin flow channel as can occur in more recent injector designs [2] or #4 with eccentricity [1] in combination with such thin channel). It is worthwhile noting that eccentricity is a characteristic feature of large two-stroke diesel injectors ([3], [4]).

Table 1. Injector configurations considered. Blue: reference injector. Variations are in orange: Introduction of eccentricity e , reduction of channel height s , comparison with single-hole nozzle and introduction of swirl.

Nozzle name	Nozzle number	d	e	s	D
[-]	[-]	[mm]	[%]	[mm]	[mm]
Reference	2	0.75	0	0.75	1.3
Eccentric	4	0.75	0.8	0.75	1.3
Thin channel	1	0.75	0	0.50	1.3
Without bypass	15	0.75	0	0.75	-
Swirl Chamber	34	0.75	0	0.75	0.5

Figure 2 shows the main elements of the generic injector design hardware for the particular configuration number 34, which involves a swirl-generation chamber underneath the orifice. The insert is fixed with a screw such that the swirl chamber is located in axis-symmetric position relative to the orifice.

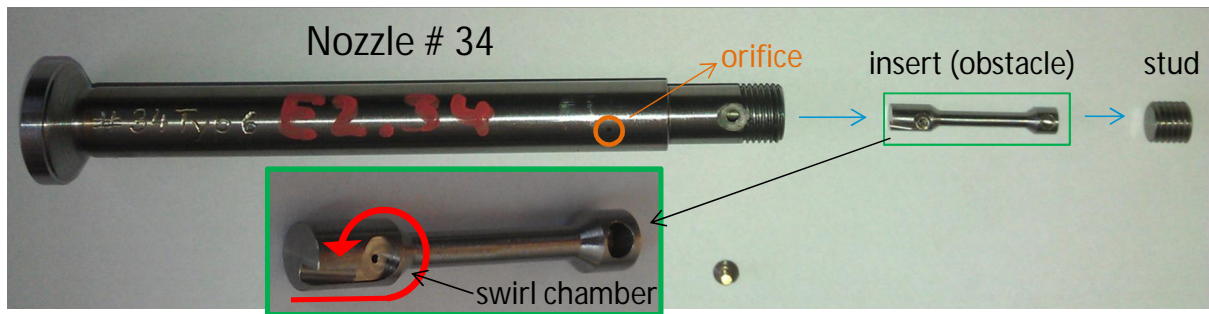


Figure 2. Nozzle configuration 34 with the insert to position the swirl chamber under the orifice.

Experimental setup

The investigations were performed making use of the Spray Combustion Chamber (SCC), an optically accessible constant volume chamber of dimensions representative of smaller two-stroke as well as larger four-stroke marine diesel engines ($\text{\O}500 \times 150 \text{mm}$) [5] and validated at conditions typical of those engines [6]. Figure 3 shows a schematic of the test facility, illustrating the working principle of the experimental setup: A pressure vessel equipped with fast opening valves feeds process gas via a heater, the so-called regenerator, into the SCC. Pressure and temperature levels as well as swirl in the spray combustion chamber are adjustable by changing accumulator pressure and/or duration of the blow down process and heater core temperature. By starting the injection with sufficient delay after the end of the blow-down process, due to the continuous decay of the swirl, quasi-quiescent conditions can also be realised.

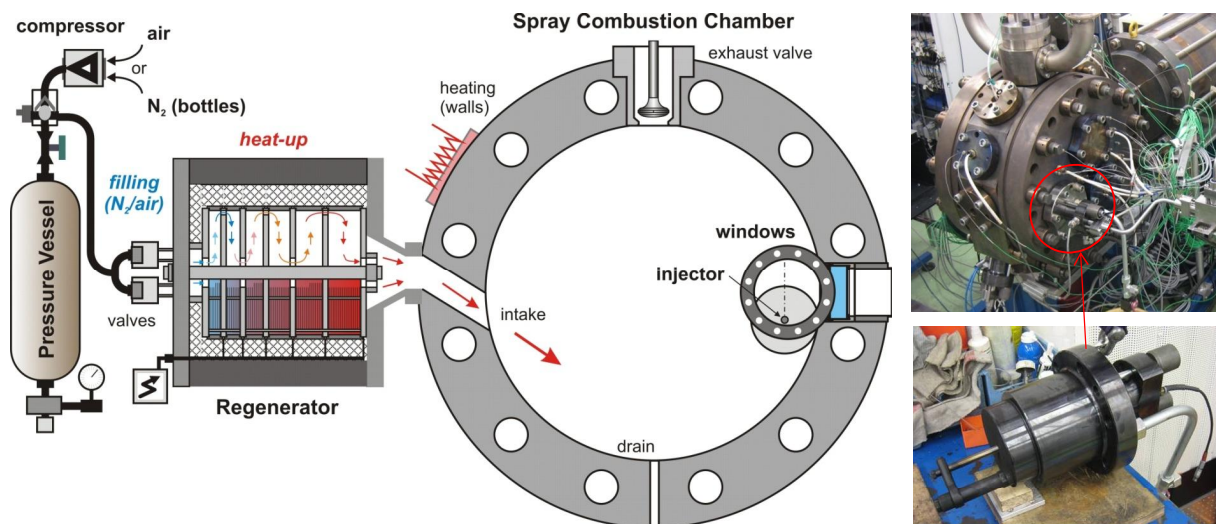


Figure 3. Spray Combustion Chamber schematics.

Figure 3 also illustrates the specific injector positioning and optical access arrangement options: The injector was positioned in the back cover, via an adapted dummy device (compare [1]), which was mounted instead of a window. The photographs in Figure 3 give an impression of the complete assembly and installation of this "injector in dummy" configuration, including the mounted injector, the nozzle with the eccentric single-hole and the fuel discharge system for the bypassed amount of fuel.

A Nd:YLF-Laser was used for illuminating the liquid phase of the spray. The scattered light was then captured with two high speed cameras operated at 20 kHz, which were located in front and on the side of the chamber. Figure 4 shows the optical setup, with the injector mounted on the back side of the SCC and the two cameras observing the front and the side of the spray. Camera one is flanked by the two illumination beams: With the help of a semi transparent mirror the laser light was separated into two part-beams which were then guided past both sides of the front camera. Using such setup resulted in a more homogenous illumination of the spray and therefore an easier detection of possible asymmetry in the spray pattern.

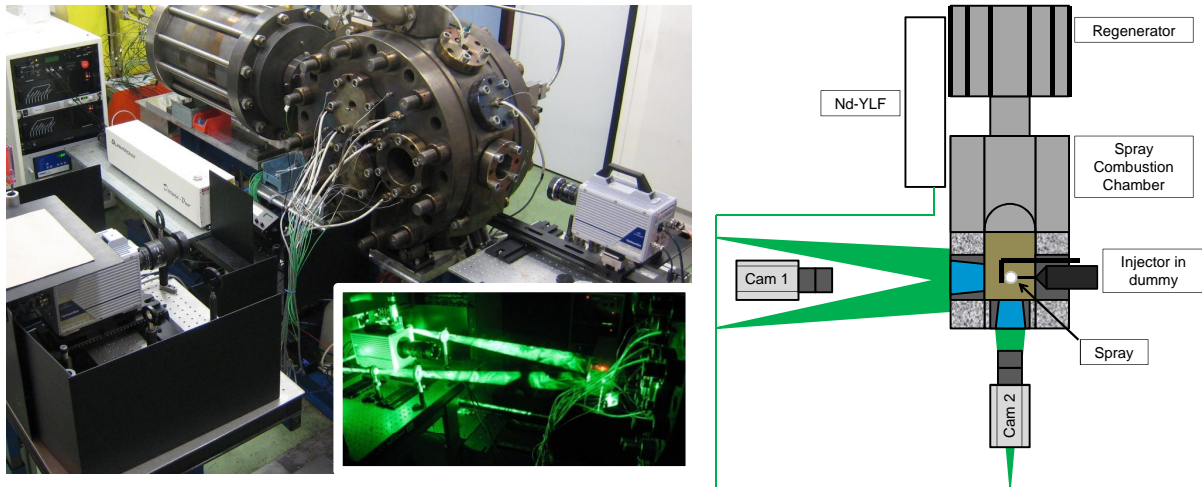


Figure 4. Optical setup at the Spray Combustion Chamber: Nd:Ylf illuminating the liquid phase in combination with two high speed cameras recording the scattered light

Simulation setup

Currently, most cavitation models are based either on a single-fluid mixture approach using a homogeneous equilibrium model (HEM) [7] or a simplified Rayleigh-Plesset (RP) bubble dynamics equation [8]. Both approaches ignore the thermodynamic processes happening at the liquid-gas interfaces during phase changes. In order to overcome this deficiency, IFPEN has developed a new cavitation model called GERM based on the equilibrium at liquid-gas interface of the Gibbs free energy of liquid and gas (IFP-C3D software [9], [10]), in a general framework of highly compressible, turbulent and multi-species two-fluid models [11]. The applicability of the GERM cavitation model has been verified using a typical diesel injector for which experimental data is available [12] and the results show a fairly good qualitative agreement between the numerical and the experimental cavitation regions.

The numerical investigations of the influence of cavitation on the in-nozzle flow have been carried out for the three different injector configurations 1, 2 and 4 according to Table 1. The layout depicted in Figure 5 presents the dimensions of the different components of the injector configuration. In particular, the discharge pipe has been simplified based on its estimated volume.

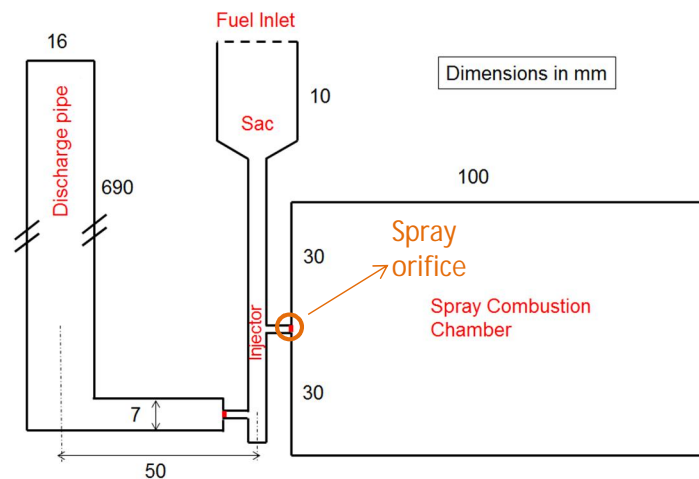


Figure 5. Dimensions of the basic computational domain applied for the simulation of the different injector configurations.

Figure 6 shows different views and details of the mesh of the reference configuration 2. The meshes used in this study have a minimum cell refinement of 50 microns. The total number of cells is around 1.1 Million. The numerical time step Δt is computed automatically with a maximum value of 10^{-9} seconds. The simulation has been performed on 254 processors (Intel SandyBridge 3.5 GB RAM) of the ENER110 supercomputer of IFPEN resulting in about five days computational costs for 1 ms injection duration.

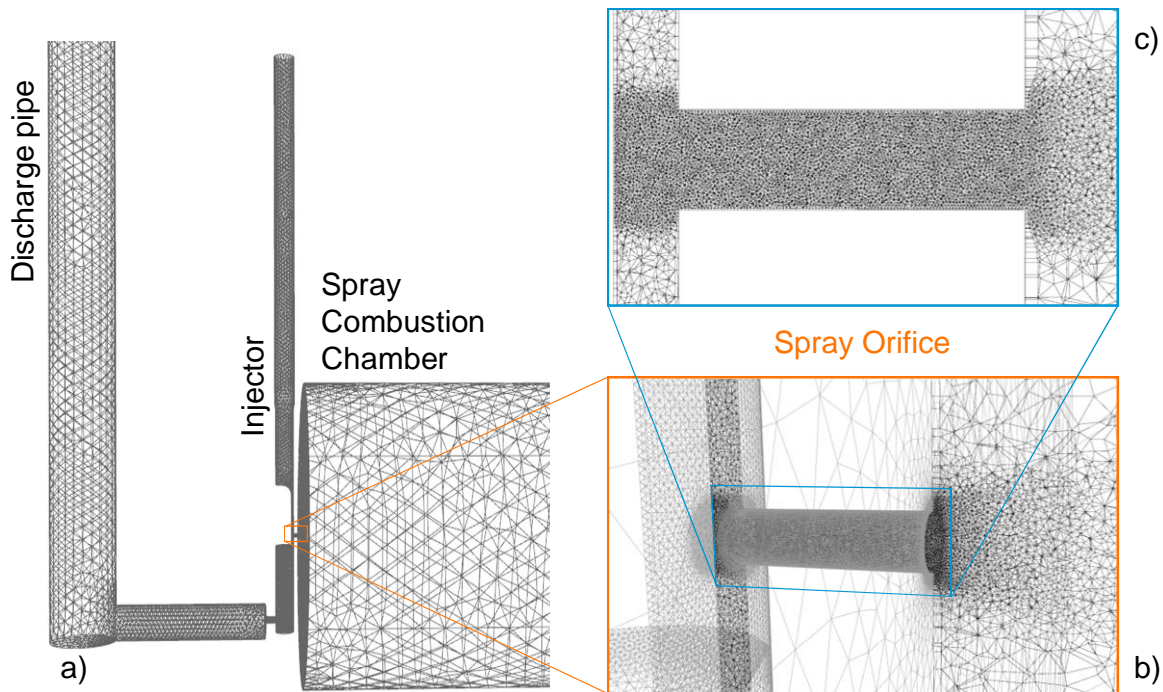


Figure 6. Mesh of the injector and the abutting volumes of the discharge pipe and the SCC a), magnified image of the mesh in and around the spray orifice b) and a cut through the latter c)

In order to perform the present cavitation simulations using similar pressure conditions as in the experiments, the injection pressure (700 bar) is specified at the fuel inlet (compare Figure 5). Moreover, this pressure value has been specified for the liquid fuel initially inside the nozzle. Initial conditions are furthermore required for the liquid volume fraction and the temperature by the IFP-C3D two-phase, highly compressible model. Figure 13 (b) shows the initial gas volume fraction distribution. Initial temperatures of both the liquid and the gas have been assumed at 400 K, corresponding to non-evaporating conditions and also in order to avoid liquid jet flash boiling in the combustion chamber. Future work will address this highly complicated process using higher temperature in the combustion chamber.

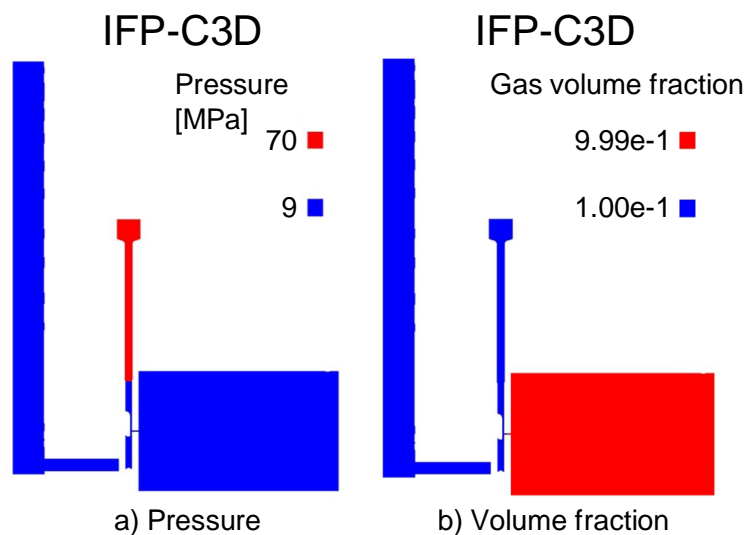


Figure 7. Initial conditions

Results

Experimental

The five nozzle-hole configurations listed in Table 1 and discussed earlier have been tested under non-evaporating, no-swirl conditions. The gas pressure was set to 4 MPa and 400 K resulting in a gas density of about 33 kg/m³, rail pressure was set to 80 MPa: The conditions in the SCC are hence largely similar to those in an engine running at medium load, except for swirl and temperature.

For the comparison of the different cases, the spray during the steady injection period has been analysed. Note that this approach was first verified and applied for investigating sprays from traditional injectors without insert and the results of those earlier investigations [1] serve as additional references in the discussion below. For the analysis of the results, the individual images acquired during the steady injection period have first been averaged for each experiment separately. Then, the mean images of the ten experiments of one series have been averaged again and those averaged images then serve as the basis of the following discussion.

The two images on the left side of Figure 8 show the spray of the reference case of the present measurement series (image a) compared with the reference case of earlier measurement series [1] without obstacle in the nozzle (image b). In both cases, the spray propagates without deflection from its nominal spray axis – defined by the orifice. However, in the case with obstacle, the overall spray angle is apparently increased.

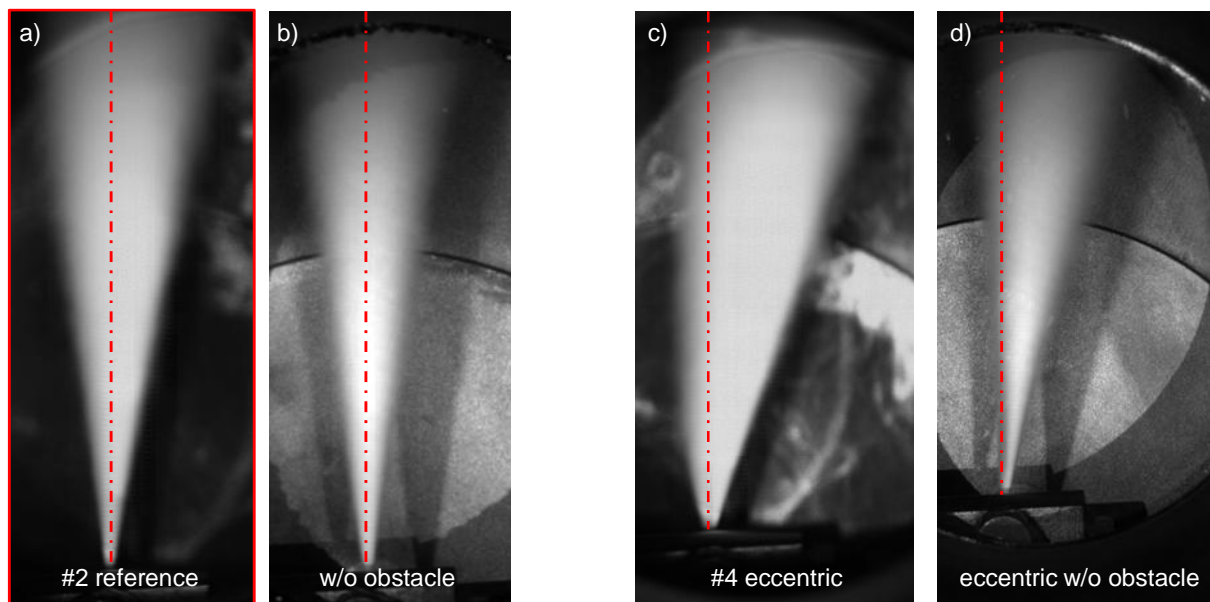


Figure 8. Comparison of the reference case with and without obstacle (left images) as well as the eccentric case with and without obstacle (right images)

The effect of nozzle-internal geometry features on spray characteristics is even more clearly visible when looking at the two cases shown on the right side of Figure 8. It shows again a comparison of two cases with and without insert; however with non-negligible eccentricity of the orifice (0.8) relative to the nozzle axis.

In case of an eccentric nozzle hole, the spray is significantly deflected against the direction of the eccentricity, which is the case for both sprays. Again, the presence of the insert in the flow passage underneath the orifice leads to an increased spray angle (comparing Figure 8 c to d), with the increase appearing to be even more pronounced than in the non-eccentric case. Moreover, the spray becomes clearly non-symmetric.

Figure 9 shows the averaged images for the five investigated cases. The reference case (nozzle #2) shows a symmetrical jet with a spray angle of about 20°. If the channel height s is reduced, as in nozzle #1, the spray is not influenced in lateral direction. The introduction of eccentricity on the other hand has a significant influence on the spray, as already discussed earlier.

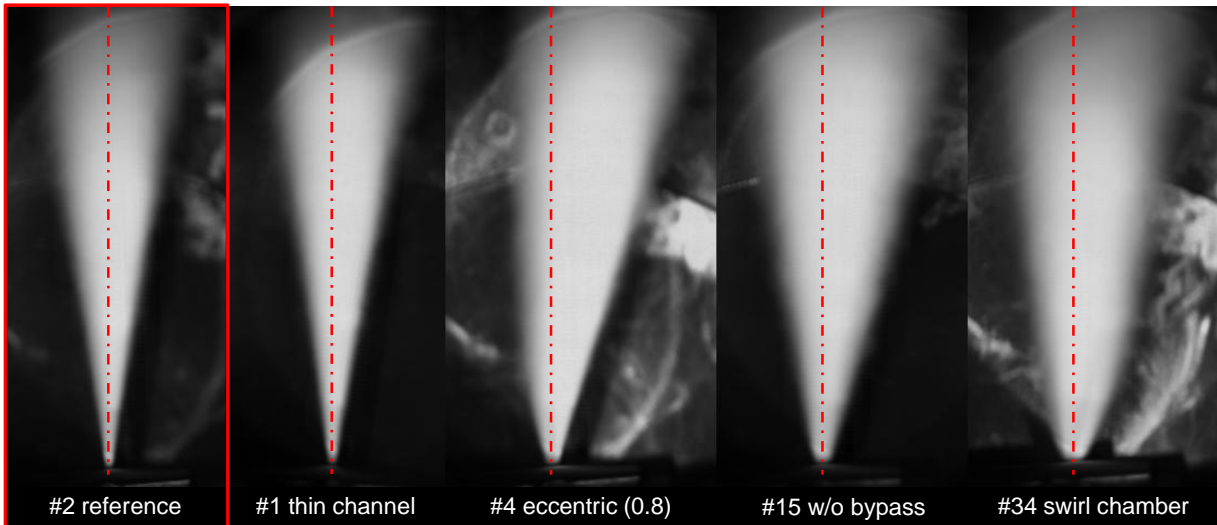


Figure 9. Spray images of the investigated nozzle designs, taken from the front and averaged during the steady phase.

In case of a nozzle without bypass, as in nozzle #15, the spray angle is increased compared to the reference case. It has to be stated here that the increase seen is not only the result of a drastically increased spray angle, but a combination of increased spray angle and an instable spray axis: From shot to shot, the spray axis of nozzle #15 stabilizes at one of two steady positions. Either the spray tilts 2° to the right or the left side of the theoretical spray axis. Statistically, the spray behaves such that it prefers the position on the right side, which is why the averaged image is pointing to the right.

The swirl chamber, as in nozzle #34 (Figure 9, right image) leads to a very broad spray. The swirl produced in the nozzle is transferred through the orifice and the vector components of the flow field pointing in tangential direction lead to a larger spray angle. Compared to the reference spray, the spray from the swirl chamber also shows a different spray shape. While for nozzles #2, #1 and #15 the shape is close to a triangle with more or less constant spray angle, nozzle #34 results in a shape which is very wide close to the nozzle, but with a change in the spray angle after a third of the total visible length. Note that such change of spray angle can also be observed in case of eccentricity, but here only on the left side, i.e. in the direction of the eccentricity.

Figure 1 shows the views from camera 2 on the side of the spray. Note that the dark zone in the middle of the spray is an artefact arising from the post processing of the images.

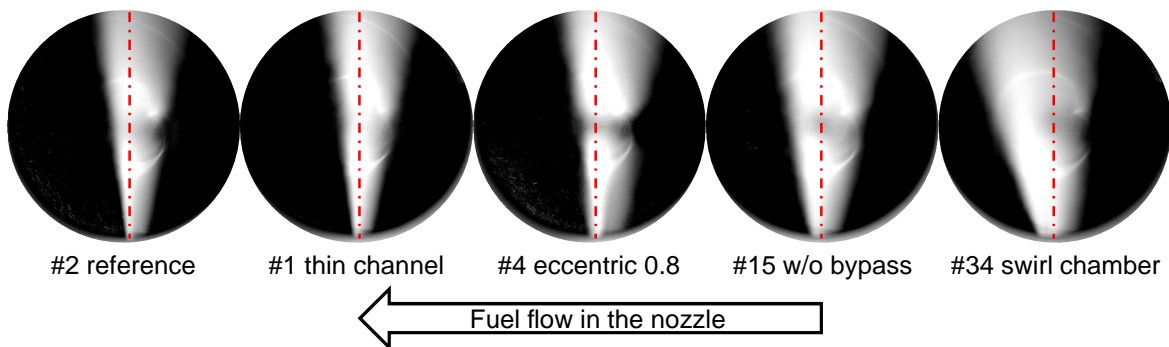


Figure 10. Spray images of the investigated nozzle designs, taken from the side and averaged during the steady phase.

Nevertheless the images allow assessing the characteristics of the spray in axial direction: For the reference case, the spray is leaning backwards, against the direction of the fuel flow in the nozzle. For the case with a reduced height of the flow channel (nozzle #1) the spray is slightly wider yet very similar. But if an eccentricity is introduced the spray

becomes wider, with a tendency against the flow direction in the nozzle. Interesting is the case without bypass, comparable to a single-hole nozzle: The spray becomes considerably wider than in the reference case and is also more symmetric, the leaning against the flow direction in the nozzle is less pronounced. For the swirl chamber, the flow becomes very wide, as was already observed from the front side. Moreover, it seems to be leaning more in the direction of the flow in the nozzle.

Simulations

A selection of the cases shown in the foregoing part has been investigated with help of CFD simulations. Due to limitations in time and capacity only three cases have been looked at up to now: Nozzle #1 “thin channel”, nozzle #2 “reference case” and nozzle #4 “eccentric case”.

Figure 11 shows the velocity distribution on a cut through the orifice, perpendicular to its axis, viewed from above.

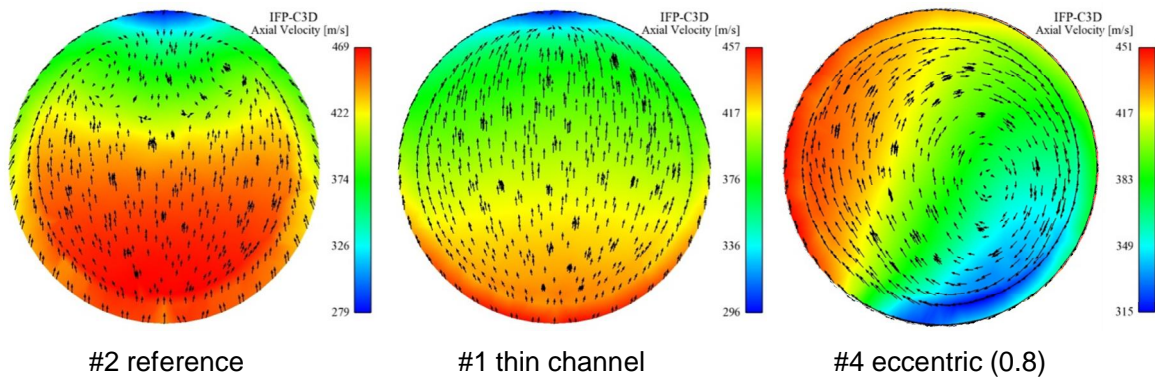


Figure 11. Exit section of the nozzle orifice: Nozzle exit velocity [m/s] and velocity vectors.

As can easily be seen, the two cases without eccentricity are characterised by a velocity distribution, which is symmetric with respect to the nozzle axis, also characterised by non-zero velocity components against the direction of the flow in the injector. If the orifice is in an eccentric position relative to the nozzle axis this symmetry pattern is no longer observed. The region with high axial velocity (pointing out of the plane) is moved to the side. Additionally we see a swirl pattern with its centre shifted to the region with lower velocities. A qualitative comparison of the averaged axial velocity component shows that with the thin channel, the exit velocity is clearly reduced compared to the reference case. For the eccentric case, a similar reduction of the averaged velocity can be observed, but not to the same extent as for the situation with reduced channel height.

The velocity distribution on a cut through the orifice along its axis is shown in Figure 12. Clearly visible are the geometric changes due to the different installation situations: The reduced channel height in nozzle #1 and the slightly longer orifice caused by the eccentric position of the orifice in nozzle #4.

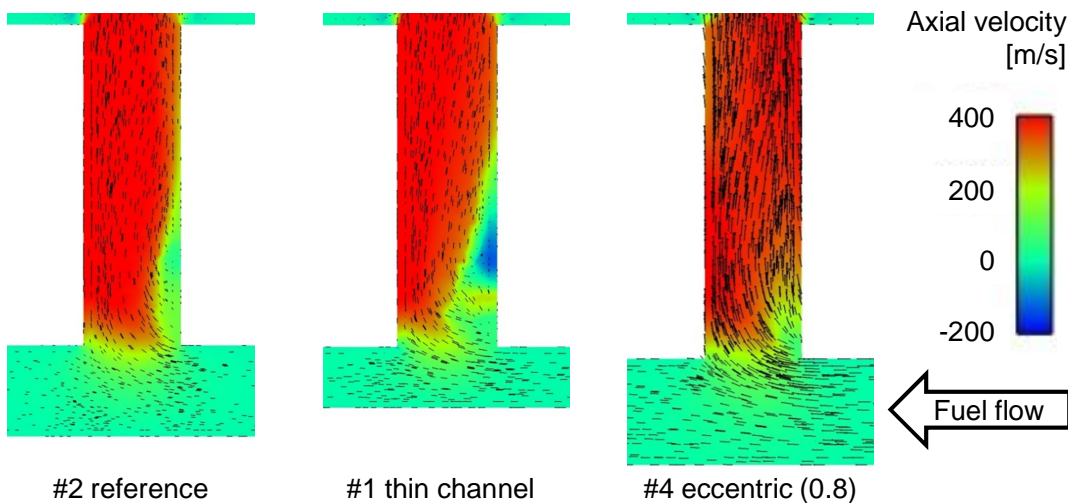


Figure 12. Cut through the orifice along its axis (similar to cut B-B in Figure 1)

These plots illustrate the different processes in the orifices leading to the velocity fields shown in Figure 11. Due to the velocity of the fuel, entering from the right side of the image section, passing through the nozzle underneath the orifice and entering it, a recirculation zone is formed, which is associated with lower axial velocities on the right side of the orifice. This is the case with all configurations; however, the size of this zone is clearly largest with the thin channel case. Obviously, the length of the orifice is not sufficient for achieving complete re-equilibration of the flow before the orifice exit, such that the velocity on the side of the recirculation zone is still lower at this point (as can be seen in the upper parts of the corresponding velocity distributions in Figure 11). In the eccentric orifice case, the recirculation zone size is smaller than in the other cases. At the same time, however, the flow pattern in the orifice seems to be more complex in general. In contrast to the other cases, the minimum velocity at the exit is now on the opposite side of the recirculation region, respectively on the bottom of the velocity distribution plot for this case in Figure 11.

Discussion

We can identify a few general features of the morphology of sprays originating from the orifices of such generic nozzle configurations modelling typical injector arrangements applied on large Diesel engines, which the detailed CFD simulations help to understand on the basis of the in-nozzle flow characteristics.

Sprays from non-eccentric orifices are propagating symmetrically with respect to the axis of the injector; however, when viewed from the side, a non-negligible deflection against the direction of the fuel flow inside the injector can be observed, and the spray is wider in this view. The velocity distributions at the orifice exit are also symmetric with respect to the injector axis; it is, however, also characterised by a significant velocity gradient between the side in direction of the flow inside the nozzle and the side downstream of the recirculation zone formed at the orifice inlet. One may assume that the non-zero velocity components in this direction are promoting the more pronounced expansion or leaning of the spray against the flow inside the injector.

In the case of non-eccentric orifices, the height of the channel in the injector seems to have only moderate impact on the spray – though, when comparing with the case without insert (hence with maximum possible channel height), there is a trend towards wider sprays with decreasing channel height. Also, when considering the side views, the lower channel height seems to be associated with a slightly larger spray angle, which would be in line with what one would expect on the basis of the velocity distribution at the orifice exit, where the non-orifice-axial velocity components appear to be slightly more pronounced than in the reference case. It is worthwhile noting that the rather large recirculation zone in this case, which is inevitably also associated with occurrence of cavitation, not only yields a reduced velocity uniformity at the orifice exit but also contributes to the higher pressure loss with this configuration, arising from the smaller cross-section of the channel. As a consequence, for the same boundary conditions at the injector inlet, the spray tip penetration is reduced compared to the reference case.

Using eccentric orifices changes the character of the spray significantly: It is getting wider, not only in the side view but specifically in the lateral direction. At the same time, it is strongly deflected, against the direction of the eccentricity. This latter effect can again be explained on the basis of the velocity distribution at the orifice exit, with two factors possibly contributing:

- Firstly, there is again a very pronounced variation of the (orifice-axial) velocity; however, with the maximum shifted (or rotated) to the side of the eccentricity. Hence, following the same line of argument as above, the gradient of the axial velocity should be associated with non-zero velocity components against the direction of the eccentricity, which could be expected to lead to a stronger expansion toward this side of the spray. An in-depth investigation with focus on cavitation effects can be found in [13].
- However, similar considerations should be valid in the perpendicular direction: The lower axial velocities on the side in direction of the flow inside the nozzle should, if this were the governing factor, also trigger a large angle to this side – the spray should be leaning towards this side instead of against the direction of the flow inside the injector. In fact though, the behaviour is very similar to the cases without eccentricity. This is not well understood and will require further investigations.
- Secondly, the flow at the orifice exit is characterised by a swirl-like pattern, specifically in the region of lower axial velocities. The effect of such swirling motion is clearly visible when looking at the results of the special case #34, with the integrated swirl chamber: The corresponding spray seems to exhibit some similarity to the characteristics of a hollow-cone spray generated by a pressure-swirl atomiser – the spray angle is considerably increased in all directions. In that particular case, the orientation of the spray in the side view is moreover shifted in the direction of the flow inside the injector, which is also not fully understood and might call for more thorough investigations, including CFD studies. Nonetheless, the results support the consideration that the creation of swirling motion inside the orifice tends to increase the width of the spray.

The special case of the injector without bypass underlines the importance of applying realistic flow conditions inside the injector for studying the characteristics of single sprays. Not only may random flow stabilisation effects as

observed with this particular setup and described above lead to erroneous interpretations of test results. But, well more importantly, the flow into the orifice may also be altered in such a way that any results obtained are actually far from representative. For the size of orifice considered here, the spray angle would be significantly overestimated when neglecting the fuel flow past the other orifices.

Conclusions and outlook

The work shows the results of investigations performed in the so called Spray Combustion Chamber at Wärtsilä Switzerland Ltd. in Winterthur. It is the logic consequence of earlier investigations [1] and uses the experimental setup introduced in that work to investigate phenomena of large injection systems. Situations which can occur due to the special geometric situation in large marine 2-stroke diesel engines have been modelled with the help of generic nozzles which allow the investigation of the occurring effects separated from other influences. The experiments were conducted under conditions close to those in real engines, applying a rail pressure of 80 MPa and a gas density of about 33 kg/m³.

In parallel, extensive CFD simulations using an advanced cavitation model allowing the description of the two-phase, highly compressible flow inside the injection nozzle have been performed in order to enable a better understanding on how fluid-dynamic effects occurring in the injector influence the propagation of sprays. The results obtained indicate that injectors of this size are characterised by high levels of non-uniformity of the axial velocity at the orifice exit. As a consequence, the spray is not propagating symmetrically into the combustion chamber, but is rather characterised by different angles in the direction of the nozzle-internal flow and perpendicular to it. Moreover, the spray may actually be deflected from its theoretical (orifice) axis. Applying eccentric arrangements of orifices clearly enhances this pronounced three-dimensional spray structure: On the one hand, the velocity distribution at the orifice exit is substantially modified, which results in a strong deflection of the spray against the direction of the eccentricity. On the other hand, this creates a swirling flow pattern in the orifice, which in turn tends to lead to larger spray angles, as it has been demonstrated experimentally by investigating a special design with incorporated swirl chamber. It could also be shown that such effects are further enhanced with reduced channel height below the orifice inlet.

Not all the observed effects have been fully understood and further investigations, both on the experimental and the simulation side, are needed in order to establish a better insight into the governing phenomena and the (also quantitative) impact of key parameters of injector geometry and injection system operation. This includes also investigations into possible effects of ageing of the main components.

It could be clearly shown that it is essential to simulate the actual flow conditions inside the injector properly in order to achieve relevant results. Single-orifice experiments yielded distinctly different spray behaviour than those where the actual flow distribution in the injector had been taken into proper account.

Acknowledgements

The present work has been conducted as part of the HERCULES-C project within EC's 7th Framework Program, Contract SCP1-GA-2011-284354. Additional financial support by the Swiss Federal Government (SFOE Contract SI/500940, TP Nr 8100075, Project name "FlexFuel Combustion") is gratefully acknowledged. The fruitful collaboration with ETH Zürich is highly appreciated.

References

- [1] Schmid, A., von Rotz, B., Schulz, R., Herrmann, K., Weisser, G. and Bombach, R., "Influence of nozzle hole eccentricity on spray morphology", ILASS 2013, Chania, Greece, 1-4 September, 2013.
- [2] Spahni, M., Kyratos, A., de Jong, R., *The New X Generation Low-Speed Engines from Wärtsilä*, CIMAC Congress 2013, Shanghai, China
- [3] S. Hensel, K. Herrmann, R. Schulz and G. Weisser, "Numerical analysis and statistical de-scription of the primary breakup in fuel nozzles of large two stroke engines for the application in CFD engine simulations", COMODIA 2012, July 23-26, 2012, Fukuoka, Japan.
- [4] R. Schulz, S. Hensel, B. von Rotz, A. Schmid, K. Herrmann, and G. Weisser, "Development of Spray and Combustion Simulation Tools and Application to Large Two-Stroke Diesel Engine Combustion Systems," in Proceedings of the Council International des Machines a Combustion (CIMAC) Congress, Shanghai, no. NO. 259, 2013.
- [5] Herrmann, K., Schulz, R. and Weisser, G., "Development of a Reference Experiment for Large Diesel Engine Combustion System Optimization". 25th CIMAC World Congress on Combustion Engine Technology, May 2007, Vienna, Austria.
- [6] Herrmann, K., Kyratos, A., Schulz, R., Weisser, G., von Rotz, B., Schneider, B. and Boulouchos, K., "Validation and Initial Application of a Novel Spray Combustion Chamber Representative of Large Two-Stroke Diesel Engine Combustion Systems", ICLASS 2009, 11th Triennial International Annual Conference on Liquid Atomization and Spray Systems, July 2009, Vail, Colorado USA.

- [7] C. Habchi, N. Dumont, and O. Simonin, "Multidimensional Simulation of Cavitating Flows in Diesel Injectors by a Homogeneous Mixture Modeling Approach," *Atomization and Sprays*, vol. 18, no. 2, pp. 129–162, 2008.
- [8] E. Giannadakis, M. Gavaises, and C. Arcoumanis, "Modelling of cavitation in diesel injector nozzles," *JOURNAL OF FLUID MECHANICS*, vol. 616, pp. 153–193, DEC 10 2008.
- [9] J. Bohbot, N. Gillet, and A. Benkenida, "IFP-C3D: an Unstructured Parallel Solver for Reactive Compressible Gas Flow with Spray," *Oil & Gas Science and Technology*, vol. 64, no. 3, pp. 309–335, 2009.
- [10] A. Velghe, N. Gillet, and J. Bohbot, "A high efficiency parallel unstructured solver dedicated to internal combustion engine simulation," *Computers & Fluids*, vol. 45, no. 1, pp. 116 – 121, 2011, also in the Proceedings of the 22nd International Conference on Parallel Computational Fluid Dynamics (ParCFD 2010).
- [11] C. Habchi, "A Gibbs free Energy Relaxation Model for Cavitation Simulation in Diesel injectors", ILASS - Europe 2013, 25th European Conference on Liquid Atomization and Spray Systems, Chania, Greece, 1-4 September, 2013.
- [12] J. Blaisot, J. Daridon, C. Garsiand, P. Gastaldi, J. Grolier, S. Honnet, B. Loyer, P. Manuelli, L. Méès, I. Mokbel, B. Réveillé, R. Saliba, and M. Zellat, "NADIA_bio: New Advanced Diagnosis for Diesel Injection Analysis and bio fuels - A federative project around hydraulics of CR systems," in Proceedings of the SIA, INSA Rouen, France, June 5-6, 2012.
- [13] C. Habchi, N. Gillet, A. Velghe, J. Bohbot, A. Schmid, B. von Rotz, K. Herrmann, " On the role of Cavitation in Marine Large Diesel injector: Numerical investigation of nozzle orifices eccentricity ", ILASS - Europe 2014, 26th European Conference on Liquid Atomization and Spray Systems, Bremen, Germany, 8-10 September, 2014.