Modeling the influence of the nozzle flow on diesel spray atomization under high pressure injection conditions

J. O. Soriano[1], M. Sommerfeld[2], A. Burkhardt[1] and U. Leuteritz[1]*

[1]Siemens VDO Automotive, Regensburg, Germany.
[2]Faculty of Engineering Science, Martin Luther University Halle-Wittenberg, Halle, Germany.

Abstract
This paper describes a further development of a cavitation-induced spray model in which the flow conditions in the nozzle are employed to characterize the spray. Some preliminary results for evaluation of the model performance are presented. The model demonstrates the transition from the flow inside the injection hole to the dense spray near the nozzle and is able to map the dominant influence of cavitation and turbulence on the three dimensional spray. As a first step, this study shows simulations of turbulent cavitating flows in real size nozzles. Then, the properties of the simulated flow at the nozzle exit are used as boundary conditions for the Lagrangian simulation of the spray.

Introduction

One of the most essential issues that influence diesel mixture formation in the combustion chamber is the flow inside the tip of the injector. This, together with the fuel pressure, the charge air motion and density, and the combustion bowl geometry determine the quality of the combustion process and if the engine meets the demanding requirements imposed on Diesel engines in the last few years.

One example of the influence of the flow inside the injector is the estimation of the point of impingement of the spray at the piston bowl. Under ideal conditions, this point of impingement could be defined as the point where the geometric axis of the hole reaches the piston bowl, assuming that the injection hole axis is the same as the spray axis. This means that the point of impingement is given by the geometrical included angle of the hole Ψ_{geo} (figure 1), assuming a symmetrical spray. But this definition bases only on the nozzle and piston geometry and not on the hydraulic conditions of the flow inside and outside the nozzle, which are known to play a role [1]. Measurements carried out at Siemens VDO Automotive AG on two real injection nozzles with different geometries indicate the influence of the injection hole geometry on the impulse distribution of the spray, and therefore on the expected point of impingement.

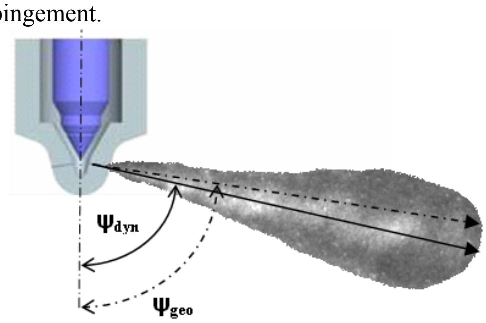

Figure 1: Included angle definition

Then, for designing purposes, it is necessary to know the dynamic and therefore the actual included angle of the spray Ψ_{dyn} (figure 1), which can be measured with a spray momentum test bench.
The development of a CFD tool capable to reproduce this hydraulic behaviour would be of a great importance for the design process of new and better injection systems.

Specific objectives

The objective of this paper is a preliminary validation of the performance of a cavitation and turbulence-induced break-up spray model by means of impulse measurements. The initial turbulent and cavitating conditions of the spray are obtained from in-nozzle flow simulations of two different real size nozzles.

Both studied nozzles are used in real engines and have very similar geometries, whose relevant parameters are summarized in Table 1.

	Nozzle A	**Nozzle B**
CF	1.5	0.7
HE (µm)	25	30
φ_1 (°)	78.84	85.63
φ_2 (°)	101.86	94.75

Table 1: Geometrical parameters of the studied nozzles

Here, CF (conicity factor) and HE (inlet rounding) are typical geometrical parameters of diesel injection nozzles [2], where:

$$CF = \frac{D_{in}[mm] - D_{out}[mm]}{L_{hole}[mm]} \cdot 100 \qquad (1)$$

φ_1 and φ_2 are other geometrical parameters that influence the nozzle flow, as described in figure 2. Length and exit diameter of the injection holes have very similar values. Both nozzles have 8 injection holes and show very low cavitation intensities.

* Corresponding author: oscar.soriano@siemensvdo.com
Proceedings of the 21st ILASS-Europe Meeting 2007

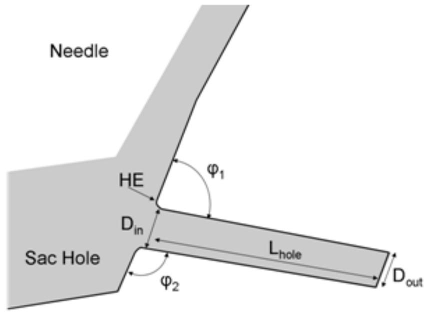

Figure 2: Scheme of nozzle tip. Geometrical parameters

Based on the results of a detailed investigation of the flow inside the injection holes of high pressure Diesel injectors [3], a model for cavitation and turbulence induced primary break-up of liquid jets was developed some years ago [4]. The model uses a cavitation and turbulent energy based approach for the evaluation of all necessary starting conditions for the calculation of secondary break-up like drop sizes, velocity components and spray angle. This model has been now enhanced to be applied for 3D nozzle flow simulations accounting not only for turbulence and cavitation intensities inside the injection hole, but also for the velocity distribution at the outlet of the hole, including secondary flows.

Experiments

TEST BENCH - The spray momentum test bench consists of a high pressure chamber, which is screwed at a rotating gear ring, as seen in figure 3. The injector is adapted from below at the rotation axis. The nozzle protrudes into the high pressure chamber, which can be turned around the injector with a step motor. To measure the side and included angle, a piezo sensor is moved on spherical coordinates perpendicular to the spray axis inside a high pressure vessel. Thereby it is possible to measure each spray hole.

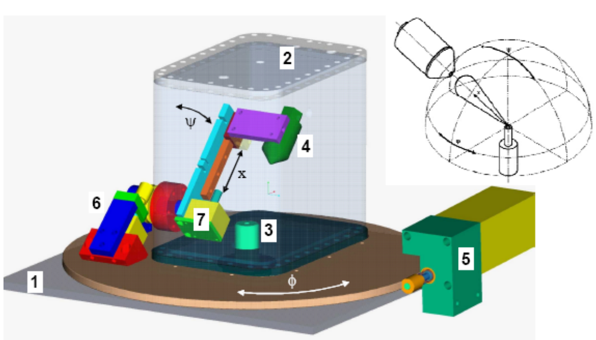

Figure 3: Setup of the spray momentum test bench

1	Basis plate	5	Side angle adjustment (Φ)
2	Pres. chamber	6	Included angle adjustment(ψ)
3	Injector fixation	7	Distance adjustment (x)
4	Sensor		

The interior of the chamber can be charged with nitrogen, which is pumped through a filter in a closed circuit. While measuring, the injector is triggered with a constant frequency by a test bench control unit. The rail pressure and the trigger time can be chosen arbitrarily. The back pressure can be adjusted up to 40 bar.

The spray momentum is the integration of force vs. injection time

$$\Delta\vec{p} = \int_{t_1}^{t_2} \vec{F} dt \tag{2}$$

Generally, the dynamic included angle is defined by the point of maximum spray momentum.

EXPERIMENTAL RESULTS - As a first step, measurements of the geometrical included angle Ψ_{geo} of the injection hole of the above mentioned nozzles were done. The results are shown as points on the left side of the diagram in figure 4. There is not a great difference between the reference and the measured values of Ψ_{geo}, but nevertheless this little deviation affects the accuracy of the CAD models used for simulations.

Then, spray momentum measurements were carried out for the different nozzles with a constant feeding pressure of 950 bar and variable vessel pressures, from 5 to 35 bar. The injection time was adjusted to 1ms. Measurements show that the maximum of spray momentum does not necessarily have the same direction as the geometrical axis of the injection hole. Actually, the impulse distribution of the sprays changes with the vessel pressure in which the spray penetrates. With increasing vessel pressures, the maximum of the spray impulse changes its direction, so that the real included angle of the spray increases.

Measurements also show dependence of the dynamic spray angle on the nozzle type, indicating different break-ups of the liquid jet because of the different nozzle geometries.

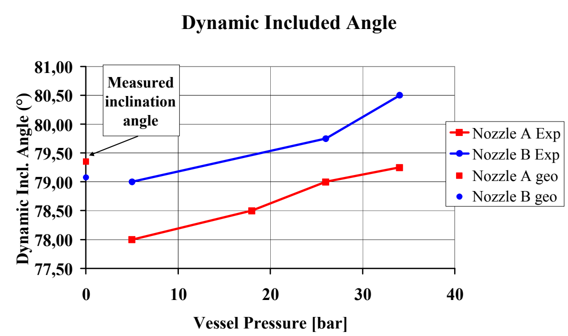

Figure 4: Dynamic included angle variation with vessel pressure

Model features

While the physics of the nozzle flow and the secondary break-up of the spray are well understood, there are still some open questions regarding the basic principles of the primary break-up of diesel sprays. A review of the literature shows that the primary break-up of the spray is the result of complex phenomena like aerodynamic interaction with the gas, internal jet turbulence fluctuation, non-axial kinetic energy and cavitation inside the nozzle. These atomization mechanisms in the liquid jet are responsible for the initial perturbations on the jet surface. Then, these waves grow according to Kelvin-Helmholtz instabilities until they detach as atomized droplets [5,6].

In this model, the above mentioned atomization mechanisms are assumed to be present in the liquid jet as break-up energy. Basing on the distribution of this break-up energy at the nozzle exit, properties of the injected droplets are calculated.

The break-up energy is calculated by means of a two-phase flow simulation of the nozzle. The results are then processed to obtain the size and velocities of the injected droplets. These droplets are then introduced into a rectangular mesh of hexagonal cells and a size of approx. 0.2 mm at the origin of the spray and 1mm in the far field. The probability for new particles to be injected at a certain position depends on the spatial resolution of the mass flow at the nozzle exit, so that a higher mass flow is represented by a higher number of particles. With this coupled approach, no initial value from empiric correlations needs to be used in the simulation chain.

This coupling method is convenient to deal with the atomization of non-axial symmetric liquid jets penetrating in dense gaseous ambient with very high velocities like the ones encountered in high pressure Diesel injection. Under these conditions a very fast disintegration of the coherent liquid flowing in the nozzle into drops of different sizes and shapes can be expected. This assumption of a very short intact core length can be found quite often in the literature [5,7]. Thus, the numerical transition of the Eulerian two-phase flow in the nozzle into the Lagrangian two-phase particle flow in the chamber should offer a good approach to the problem.

The break-up energy consists of the three following terms: flow-induced turbulent kinetic energy [5], cavitation induced turbulent kinetic energy [7] and non-axial kinetic energy [8]. While turbulent fluctuations near the surface of a jet cause instabilities which lead to break-up, it is considered here that the turbulence intensity in the core of the liquid column will not contribute to the atomization of the jet. The same approach is used when considering the instabilities caused when modeling the released energy of collapsing cavitation bubbles: only the induced instabilities near the surface are assumed to contribute to the break-up. As a third break-up mechanism it is considered in this study that the non-axial kinetic energy is also able to induce instabilities on the surface of the liquid, as investigations show [8].

Flow induced turbulent kinetic energy and non-axial kinetic energy are a direct output of nozzle flow simulation, while cavitation induced turbulent kinetic energy has to be calculated by resolving the cavitation bubble dynamics [4].

As a first step, the flow is divided in two zones depending on the liquid and gas contents. Cells with a volume fraction of vapour VF>0.3 belong to the cavitating zone (zone 2) and the rest to the liquid zone (zone 1), as seen in figure 5. Then, a cylindrical control volume for the energy conservation is defined, where the length is equal to the effective diameter of the liquid zone. After that, the break-up energy is evaluated for each zone as follows:

$$E_i = \eta_i \cdot \left(E_{cav,i} + E_{non-axial,i} + E_{kin,turb,i}\right) \tag{3}$$

Here, i=zone 1 or zone 2 and η represent the efficiency of the energy transformation, and has a value of 0.85 for this first evaluation of the model.

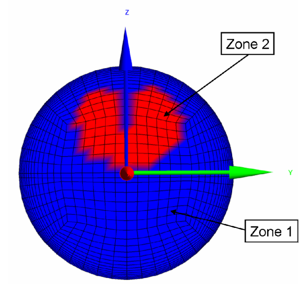

Figure 5: Typical distribution of zones at the nozzle exit

It is assumed that the total break-up energy in zone 2 turns into surface energy for the formation of the droplets and for their radial kinetic energy.

$$E_{\sigma,2} + E_{kin,d,2} = E_2 \tag{4}$$

$$n_2 = \frac{6 \cdot m_2}{\pi \cdot d_2^3 \cdot \rho_L} \tag{5}$$

$$E_{\sigma,2} = \sigma \cdot \pi \cdot d_2^2 \cdot n_2 \tag{6}$$

$$E_{kin,d,2} = \frac{1}{2}\frac{\pi}{6} \cdot d_2^2 \cdot \rho_L \cdot v_{rad,2}^2 \cdot n \tag{7}$$

For the liquid zone it is assumed that the total energy is present as turbulent fluctuations that act against the stabilizing surface forces [9]. Mass is split off the liquid zone until both forces are in equilibrium:

$$u_{turb} = \sqrt{\frac{2 \cdot E_1}{3 \cdot m_1}} \tag{8}$$

$$F_{turb} = \frac{1}{2}\rho_L \cdot u_{turb}^2 \cdot S \tag{9}$$

$$F_\sigma = 2\sigma \cdot (2 \cdot d_{eff}) \tag{10}$$

$$F_\sigma = C \cdot F_{turb} \tag{11}$$

Here, S is the surface of the liquid where the turbulent disrupting force is acting. In order to obtain plausible values for the diameter of the injected droplets, it is necessary to multiply the disrupting force by a coefficient C=$O(10^{-2})$ [9].

The remaining mass is transformed to a cylindrical droplet and its energy content is used to calculate its radial velocity. The split mass undergoes then the same calculation of eq 5 to eq 7.

Together with this primary break-up model, the CAB model is used for the simulation of the secondary break-up. The CAB model [10] is an enhancement of the well known Taylor analogy break-up model (TAB) proposed by O'Rourke and Amsden [11], where the break-up criterion is computed from the Taylor drop oscillator which models a droplet as a forced damped harmonic oscillator.

This combination should offer good predictions of the experimentally observed asymmetric sprays.

Nozzle flow simulation

The spray model uses detailed information from 3D turbulent cavitating nozzle flow simulations performed with the Ansys CFX CFD code based on a Volume of Fluid (VOF) method [12]. This model was chosen because it offers a good compromise between computational effort and results quality. The VOF-model assumes that phases share the same pressure and velocity. The flow is considered isothermal and the liquid and vapour phases incompressible, having constant density values. The flow is considered steady.

The simplified Rayleigh-Plesset model is employed for the mass transfer term between phases. This model bases on the growth of a single spherical bubble in an unbounded liquid domain [13].

The shear stress transport turbulence model is employed in the simulations [13], assuming the flow to be fully turbulent. For the purpose of this study, this model should yield enough accurate results, and allows us to work with steady state simulations as input for the spray model, even though cavitation is known to be a highly transient phenomenon. The injector is taken to be fully opened, with a needle lift of 250 μm.

Considering fully turbulent flow and steady state cavitation are simplifications which are expected to influence the accuracy of the results, but not the performance of the model.

Nozzle flow simulation results

As a first step, simulations are validated by comparing hydraulic flow measurements of the nozzles with the computed values. The nozzle hydraulic flow is the measured Diesel flow of a nozzle for a pressure difference of 100 bar in 30 seconds, as seen in Table2.

	Hydraulic Flow (ml/30s)	
	Nozzle A (CCP=0.99)	**Nozzle B** (CCP=0.98)
Measurement	322	323
Simulation	-0.45%	+1.87%

Table 2: Hydraulic flow measurements and simulations

Hydraulic flow simulations show a good agreement, for the uncertainty of the hydraulic flow test bench is of about 3%..

Secondly, simulations of the nozzle flow without cavitation model were carried out at 950 bar injection pressure. Results are depicted in figure 6, which shows a different distribution of forces at the exit of the injection holes. In the case of nozzle B, the distribution is symmetric, while in case of nozzle A, the main stress of the forces acts in direction of low dynamic angles. This is probably due to the different sac-hole geometry, which affect the in-flow conditions of the injection holes. Low values of φ_1 cause high detachment of the streamlines at the hole inlet, leading to a mass flow distribution weighted in the direction of negative values of Z, following the local coordinate system of figure 6.

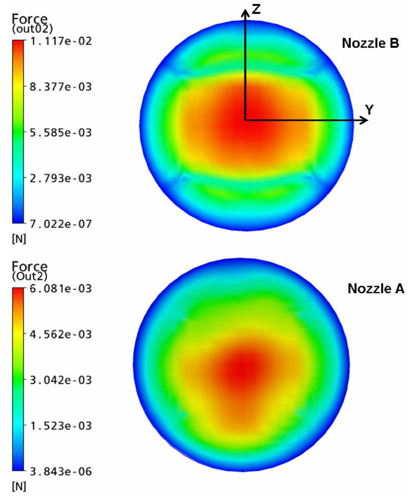

Figure 6: Flow force distribution at nozzle exit cross section. Single phase flow. 950 bar injection pressure

If cavitation is also taken into account, it would appear on the upper side of the injection hole, causing a contraction of the cross section of the flow. This contraction of the liquid area because of the vapour flow can only make the above mentioned effect stronger. In figure 7 cavitation appearance can be observed for nozzle A. Simulation show tubular cavitation films, which reach the nozzle exit.

Non-symmetric mass flow and velocity distributions are responsible for non-symmetric impulse distribution which can also be found in the spray, so that the geometrical included angle differs from the dynamic included angle. This is the reason for the offset between both experimental curves in the diagram of figure 4.

For the determination of the cavitation tendency, CCP (Critical Cavitation Point) measurements were carried out [2]. This is a non-destructive hydraulic measurement accounting for the relation of pressure to back pressure for the onset of choked flow. CCP values under 1 indicate cavitation. As seen in Table 2, measurements showed a light tendency of both nozzles to cavitate. Two-phase flow simulations of nozzle A confirmed this tendency. Despite the fact that nozzle B has a similar hole geometry, two-phase flow simulations show almost no cavitation for this nozzle. The reason for this behaviour can be found not only in the higher value of the angle φ_1 which leads to a lower detachment of the streamlines, but also in the value of the inlet rounding. Both effects tend to prevent cavitation. Inaccuracy of the turbulence model could also play a role in the underestimation of cavitation, due to wrong estimated pressure fluctuations. And finally, the assumption of perfectly smooth surfaces in the CAD models leads as well to underestimation of cavitation.

Anyway, regarding the purposes of this investigation, it is interesting to have a non-cavitating input for a better evaluation of the spray model.

Turbulent intensities and secondary flow conditions are similar for both nozzles.

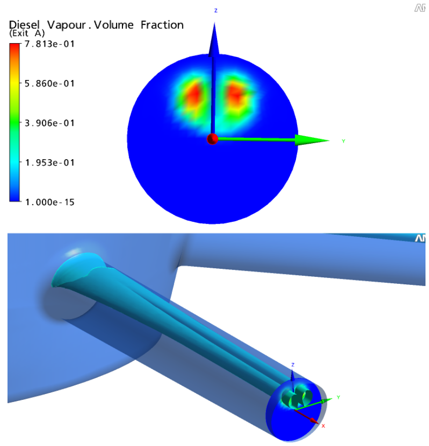

Figure 7: Cavitation appearance for nozzle A. Exit cross section & isosurface VF=0.3

Spray simulation

For the description of the spray penetrating the dense gas ambient two phases are considered: the dispersed phase (fluid droplets) and the gas phase (continuum). The gas phase is modeled with the Eulerian approach using the Navier Stokes equations, while the dispresed phase is tracked through the flow in a Lagrangian way. The tracking is carried out by forming a set of ordinary differential equations in time for each particle, consisting of equations for position and velocity [13].

As the concentration of the droplets in the fluid stream is high, two-way coupling is considered between the phases. This implies that the fluid affects the particle motion through the viscous drag and a difference in velocity between the particle and fluid, and conversely, there is a counteracting influence of the particle on the fluid flow due to the viscous drag. Droplet collision and coalescence are not considered. In addition to the drag forces, turbulent dispersion is also taken into account for these simulations. Although the gas-phase turbulence induced by this high-pressure-driven spray is anisotropic [14], an isotropic dispersion approach is used.

The particle source terms are generated for each particle as they are tracked through the flow. Particle sources are applied in the control volume that the particle is in during the time step.

Spray Simulation Results and Discussion

As explained above, the spray momentum test bench measures the force acting on a piezo pressure sensor, and integrates it in time to obtain the maximum of the momentum distribution. The Lagrangian simulation of the spray offers us the possibility to evaluate the force distribution, to integrate them and to calculate its maximal value as well. Therefore, a direct quantitative comparison is possible.

Considering an injection pressure of 950 bar, the spray was simulated for three different chamber pressures: 5, 20, 35 bar. Results are given in figure 8.

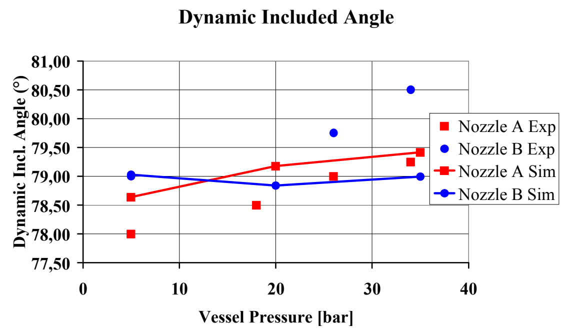

Figure 8: Dynamic included angle variation with vessel pressure. Simulation results vs measurements

Spray simulations of both nozzles show different impulse distribution at different chamber pressures. The dynamic angle for nozzle A increases with the chamber pressure. This is due to the influence of cavitation bubble collapses on the break-up process. The break-up of zone 2 depends strongly on the released energy of the collapsing bubbles. The higher the chamber pressure, the more break-up energy is released for the same amount of mass, and the bigger are the radial velocities of the droplets created in zone 2. This will cause the momentum distribution to move to higher dynamic angles (positive Z values in the local coordinate system). Contrary to the measurements, simulations of the flow inside nozzle B show almost no cavitation, and therefore the angle remains constant.

Although the slope of the results curve for nozzle A is reasonably well predicted there is an offset between measured and simulated dynamic included angles, especially at low vessel pressures. This is possibly due to a combination of effects acting in the same direction. First of all, it is likely that cavitation is under predicted. The simplified used CAD models might not be enough accurate, so that simulations fail to calculate the real turbulent and cavitation conditions of the flow. The more cavitation in the upper wall of the nozzle, the more asymmetric is the flow in the opposite direction, leading to lower dynamic angles. Secondly, numerical issues could also play a role. The modeling of an almost pure liquid column with a Lagrangian approach and without a collision and coalescence model might underestimate the impulse distribution in the direction of low dynamic angles. It is supposed that the asymmetric impulse distribution in the nozzle is retained in the spray for a certain distance along the axis, which might be proportional to the length of the liquid intact core, which again is inversely proportional to the vessel pressure. Therefore, the applicability of the Lagrangian approach for low vessel pressures is lower than for high pressures. In addition to that, a high collision rate can be expected near the nozzle outlet as here a very dense spray is observed, leading to bigger droplets [15] and a more pronounced impulse distribution in the above mentioned direction.

Other reasons for this behaviour can be the well known mesh dependency of the Lagrangian approach regarding momentum transfer between phases and the need of more validation in order to further adjust model constants.

For nozzle B the simulated dynamic spray angle remains constant due to the lack of cavitation in the input data. There is no promoted break-up on the upper side of the hole and therefore there is no positive slope for the curve. Thus, the impulse distribution remains symmetric, as it was found at the nozzle exit (figure 6). Although the lack of cavitation in this case is very useful to evaluate the performance of the model, the agreement with the measurement is rather poor. This indicates clearly the strong dependence of the spray model on the accuracy of the input data. In the case of nozzle B, better cavitation and CAD models are needed.

Conclusions and future work

The performance of a cavitation and turbulence-induced primary break-up model based on energy conservation concepts has been evaluated by means of experimental results. It has been shown that the model is able to map the influence of nozzle flow on the impulse distribution of the spray in the combustion chamber. This can help to get a better estimation of the wall impingement point or to better calculate the spatial distribution of the fuel-air mixture in the combustion chamber.

Simulation results for both nozzles show a good model performance, taken into account the dependency of the results on the nozzle flow input and the limitations of the Lagrangian approach for dense sprays. For nozzle A, the model is able to predict the influence of the asymmetric impulse distribution at the nozzle exit and the influence of cavitation on the break-up of the spray. But the agreement with measurements for low vessel pressures results has to be improved. In the case of nozzle B the momentum distribution at the outlet is almost symmetric. But for this nozzle, the lack of calculated flow cavitation as spray input explains the poor agreement of the curve at high vessel pressures, where the effect of cavitation is stronger.

Another conclusion to be drawn from this study is the question if the CAD models of very little geometries like a Diesel injection nozzle are enough reliable. CAD models used in simulations are simplifications of the real geometries found in real nozzles. Surface roughness and little variations of the CF and HE values can highly affect the cavitating and turbulent conditions of the nozzle, leading to a different break-up of the spray.

Next steps to be taken for a better simulation of the whole injection process are planed. First of all, a more exact calculation of the nozzle flow is needed. Transient instead of steady state nozzle simulations together with a more precise turbulence model (DES) should help regarding this issue. Measurements of other variables like spray velocities, Sauter mean diameter and spray penetration and angle are planned for a better evaluation of the performance of the break-up model. Finally, droplet coalescence and collision will be taken into account. This will be presented in the future.

Acknowledgements

The authors would like to thank Dr. Chaves from University Freiberg and Dr. Kull for their comments and support.

References

[1] Kampmann, S., Dittus, B., Mattes, P., *The Influence of Hydro Grinding at VCO Nozzles on the Mixture Preparation in a DI Diesel Engine*, SAE 960867, 1996

[2] Kull, E., *Einfluss der Geometrie des Spritzlochs von Dieseleinspritzdüsen auf das Einspritz-verhalten*, Dissertation. BEV, Erlangen, 2003

[3] Busch, R., *Untersuchung von Kavitationsphänomenen in Dieseleinspritzdüsen*; Dissertation, Universität Hannover, 2001

[4] Baumgarten, C., *Modellierung des Kavitationseinflusses auf den primären Strahlzerfall bei der Hochdruck Dieseleinspritzung*, Dissertation, VDI Verlag Re.12, Nr.543, 2003

[5] Reitz, R.D., Bracco, F.V., *Mechanism of Atomization of a Liquid Jet*, The Physics of Fluid, Vol. 25, No. 10, S. 1730-1742, 1982

[6] Schneider, B.M., *Experimentelle Untersuchungen zur Spraystruktur in Transienten, Verdampfenden und nicht Verdampfenden Brennstoffstrahlen unter Hochdruck*, Dissert., ETH Zürich Nr. 15004 , 2003

[7] Hiroyasu, H., Arai, M., *Structures of Fuel Sprays in Diesel Engines*, SAE 900475, 1990

[8] Cousin, J.; *Siemens VDO Internal Workshop*. 2007

[9] Arcoumanis, C., Gavaises, M., *Linking Nozzle Flow with Spray Characteristics in a Diesel Fuel Injection System*, Atomization and Sprays, vol. 8, pp.307-347, 1998

[10] Tanner, F.X., A *Cascade Atomization and Drop Breakup Model for the Simulations of High-Pressure Liquid Jets*, SAE 2003-01-1044, 2003

[11] O'Rourke, P.J., Amsden, A.A., *The TAB Method for Numerical Calculation of Spray Droplet Breakup*, SAE 872089, 1987

[12] Bauer, W., Iben, U., Voß, M., *Simulation of Cavitating Flow in Injection Systems*, VDI Berichte, Nr. 1846, pp. 1029-1042, 2004

[13] Ansys LTD. *CFX11.0 User Manual*, 2006.

[14] Bianchi, G.M., Pelloni, P., *Modelling the Diesel Fuel Spray Breakup using a Hybrid Model*, SAE 1999-01-0226, 1999

[15] Rüger, M., Hohmann, S., Sommerfeld, M. and Kohnen, G.: *Euler/Lagrange calculations of turbulent sprays: The effect of droplet collisions and coalescence*. Atomization and Sprays, Vol. 10, 47-81 (2000)