Experimental and Numerical Analysis of Marine Diesel Engine Injection Sprays under Cold and Evaporative Conditions

C. Fink*[1], M. Frobenius[2], E. Meindl[2], H. Harndorf[1]
[1] Department of Piston Machines and Internal Combustion Engines, Rostock University
Albert-Einstein-Str. 2, 18059 Rostock, Germany
[2] AVL Deutschland GmbH, Am Pfanderling 70, 85778 Haimhausen, Germany

Abstract

In cooperation with several partners a project funded by the German government was purchased in order to investigate the emission reduction potential of modern Common-Rail injectors using different marine fuels. This includes the experimental and numerical analysis of injection sprays in an optically accessible high pressure/high temperature research chamber. Different methods like scatter light, PDA and a combined Schlieren/Scatter light-bypass technique are applied in order to quantify the spray parameters penetration length, cone angle, droplet size and velocity. The data is used for the validation and development of spray models for the simulation of diesel and heavy fuel oil injection sprays in a combustion chamber of a medium speed ship engine. In order to account for nozzle internal effects, a coupling between the nozzle internal flow and the spray simulation is done. By means of a special polymer moulding technique, real nozzle geometries were derived and considered in the simulation of the nozzle internal flow pattern. The obtained flow conditions are then used as input data for the spray simulation. A good correlation of the experimental and simulation results is observed. After a short introduction to marine diesel engine demands, the experimental setup, simulation strategy and settings as well as the results are presented and discussed. Throughout the work, different fuel conditions and properties needed to be considered.

Introduction

Approximately 95% of all ships larger than 100 BRT ships are propulsed and energized by medium and slow speed diesel engines [1]. By the introduction of the IMO-TIER I in year 2000 these engines have been faced to an emission limitation the first time. Before, this type of engines was mainly optimized regarding fuel consumption, resulting in efficiencies of up to 50% and more for large medium speed and low speed engines. This is, however, accompanied with very high NOx-emissions. Köhler [1] found that up to 18% of the global NOx-emissions may be caused by worldwide shipping. As a consequence, the IMO-TIER legislations currently only limit the NOx-emissions [2]. Figure 1, left shows the current and following stages of the IMO-TIER regulations versus rated engine speed. While TIER II may be achieved by engine internal measures, such as extended Miller cycle, improved turbocharging, charge air cooling and new flexible injection systems, it is today not at all known if TIER III is to be reached without exhaust gas aftertreatment systems. Even though NOx represents the dominant hazardous exhaust gas component during nominal engine load, soot emissions are of increasing significance in part load driving schemes, i.e. in costal waters, harbors and during maneuvering. Within several investigations, Pittermann et al. [3, 4] showed that this kind of emissions can drastically be reduced with the help of modern, fully flexible Common-Rail injection systems. However, the main constraints of applying high-end injection technology are reliability and durability during heavy fuel oil operation. Apart from a widely changing composition of heavy fuel oils (HFO) this is caused in a fairly high content of water, abrasive acting components and especially the high viscosity. These properties normally imply a very special treatment of heavy fuel oils, i.e. water and sediment separation and heating of the fuel until an acceptable viscosity is reached. The dependence of viscosity vs. temperature is presented in Figure 1, right for four different heavy fuel oils (HFO) and a standard diesel fuel (DFO). Against this background an associated research project founded by the German government was initiated with the objective to identify ways to significantly reduce marine engine exhaust gas emissions by applying newly developed heavy fuel Common-Rail injectors. Within this project, a detailed experimental and numerical analysis of the injection process, mixture formation, combustion and engine emission generation is done in close cooperation of the partners Caterpillar Motoren GmbH und Co. KG, L'Orange GmbH, WTZ Rosslau gGmbH, AVL Deutschland GmbH and the University of Rostock. The target of the investigation performed by AVL and Rostock University is (1) to analyze and understand the injec-

*Corresponding author

tion system behavior and fuel air mixture generation of large heavy fuel capable Common-Rail injection systems and (2) to implement, improve and verify spray atomization and mixture formation models.

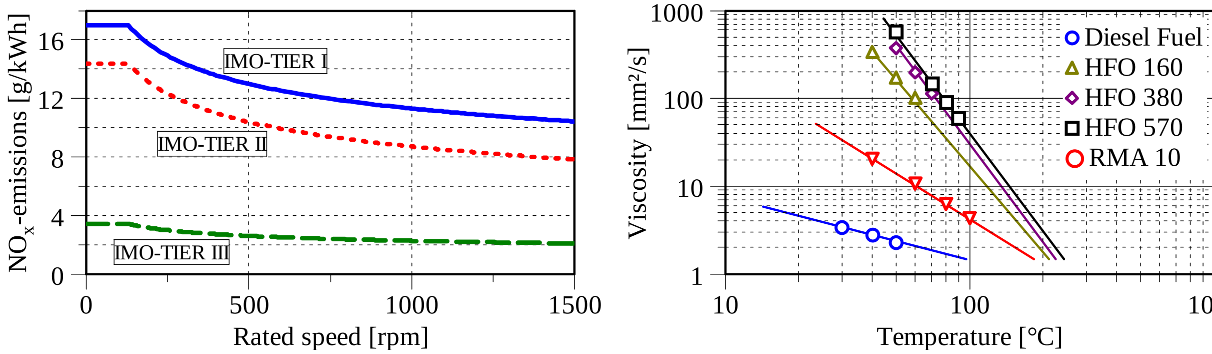

Figure 1: IMO-TIER emission limits (left); Viscosity vs. temperature for four HFOs and diesel fuel (right)

Experimental Test Set-up

For the experiments single circuit, heavy fuel capable Common Rail injectors have been used. These injectors are principally similar to those known as standard solenoid Common-Rail injectors of trucks and cars. However, some specific features by means of a separation of the solenoid from the hot fuel are implemented [5] affecting injector dynamics significantly. Table 1 presents characteristic technical data of the applied injector. In order to consider exactly the same mass flow rates at the injection nozzle outlet for the simulation as found for the experiments comprehensive measurements of the injection rate were done using a tailor made, heavy fuel capable injection rate analyzer (based on BOSCH-principle [6]). In previous work, [7] it was shown that depending on the load point there might be a strong influence of the injection rate and injected quantity on the fuel type and condition. An injection quantity reduction of up to 50% was observed when changing form a low viscosity to a high viscosity fuel condition or type, respectively. Another important input data for the simulation model concerns the nozzle hole geometry. Today, diesel injector nozzles are generally hydraulically eroded as to reduce a lifetime effect of drifting nominal injection quantities. The process of hydro eroding is typically flow rate controlled which means that geometrical parameters such as the nozzle hole inlet radius and the cone are not well defined and may vary in a certain range. In order to derive the real nozzle hole geometry a mould was created by using a special two component polymer technique. By extensive tests, performed by Buchholz [8], a very good reproducibility and reliability of this technique was shown. By means of a microscopic analysis correct contour data was obtained. Figure 2 presents a picture of the nozzle hole inlet seen through a microscope. Considering geometrical differences of each single nozzle hole inlet Buchholz [8] found by CFD calculations that the flow rate varies less than 0.44% between each nozzle hole.

Table 1: Technical data of injector

	Unit	Value
Max. Rail pressure	MPa	150
Allowable Viscosity	mm²/s	3 - 20
Max. fuel temp.	K	425
Number of nozzle holes		8
Nozzle type		sac
Hole diameter (outlet)	µm	375
Length to Diameter (L/D)		6.7

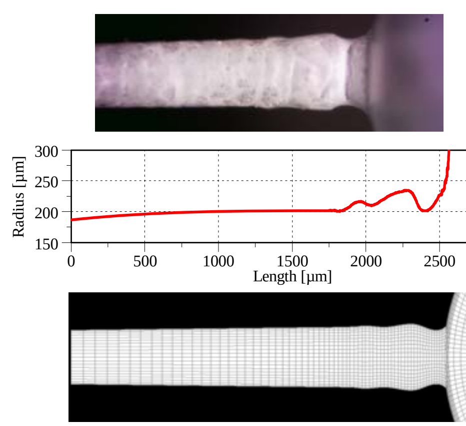

Figure 2: Microscopic view on inlet (top), contour of one nozzle hole (middle) and CFD-model (bottom)

For the experimental investigation of the microscopic (droplet velocity and diameter) and macroscopic (penetration length and cone angle) spray parameters a high pressure/ high temperature chamber with optical access was used. The chamber is designed for a constant flow of air or nitrogen and temperatures and pressures of up to 875 K and 6 MPa. Optical access is enabled by three windows. The injector is mounted that way that one jet can be observed to a length of up to 130 mm. The other jets are deflected. By means of a scatter light setup measurements of spray penetration length and spray cone angle have been performed whereas droplet velocities and droplet diameters were measured using a standard three detector PDA system. A detailed description of the pressurised chamber and the optical measurement techniques can be found in [3, 8, 9].

While the measurements described before were carried out under cold conditions, further measurements have been done under hot conditions applying a modified combined Schlieren/Scatter light setup and a high speed camera. In general, this technique does allow the simultaneous observation of liquid and fuel gas phase penetration into the pressurised chamber. In contradiction to typical Schlieren/Scatter light setups a scatter light bypass is installed in order not to pass the scatter light through the Schlieren pin hole. Therewith more scatter light fractions arrive at the camera reducing blurring effects caused by the scatter light passing the ambient fuel gas phase. Aside from that considerably decreased light intensities are required to achieve sufficiently short exposure times of 3-5 µs. Figure 3 presents the principle of the combined Schlieren/Scatter light bypass method. To avoid fuel ignition a nitrogen atmosphere was applied during measurements at elevated temperatures.

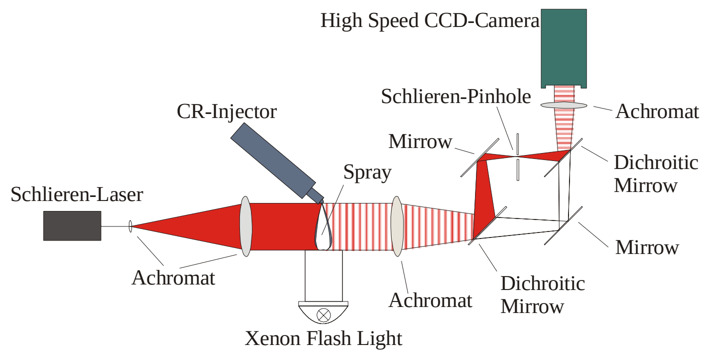

Figure 3: Combined Schlieren/Scatter light bypass technique

Numerical Model

As the droplet break up is significantly influenced by the nozzle flow, a coupled simulation approach considering the cavitation and turbulence within the nozzle, has been applied, figure 4. For the simulation of the cavitating multiphase flow in the nozzle, a non linear Rayleigh-Plesset based cavitation model [10], [11], has been used. For the simulation of the spray penetration the discrete droplet model (DDM) has been applied [11]. The primary break-up was modelled with the blob-injection model based on Bianch&Pelloni [12]. This model initializes the spray with a couple of large droplets of nozzle-diameter size. The near nozzle droplets represent the coherent liquid jet. From the primary break-up model the aerodynamic as well as the turbulent break-up is calculated dependent on flow turbulence and cavitation. Here, the influence of the cavitation bubble collaps is considered using an additional transport equation for the turbulent kinetic energy and the dissipation rate within the liquid core.

The secondary droplet break-up is modelled with the WAVE Child Modell described from Liu&Reitz [13]. The droplet evaporation is modelled using the model from Abramzon&Sirignano [14] based on liquid film theory. This model has been extended by a multi-component approach suggested by Brenn et al. [15]. Additionally, a novel correction-function-approach developed by Frolov et al. [16] has been implemented to overcome various simplifications of the standard evaporation models by several correction functions. The standard spray evaporation models are based on several simplifying assumptions, e.g., steady behavior of the gas boundary layer surrounding the drop, no temperature profile inside drops, no deformation of drop shape and no internal circulation inside the drop. Based on analytical solutions for single drops and detailed numerical models for single drops various correction factors have been derived by Frolov et al [16] taking into account the above effects into standard spray models. The method of using correction factors matched with the exact solutions is decisive to gain a fast and robust implementation into the spray model,

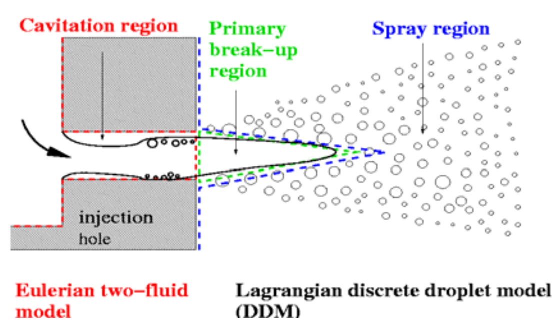

Figure 4: Coupling of nozzle flow with spray simulation

where the corrections have to be applied for thousands of droplet parcels in each time step.

Results and Discussion

Since very little differences in flow rate between the different nozzle holes were observed [8], only one nozzle hole has been simulated. As the flow in the nozzle hole is considered to be symmetrical, a half model as shown in figure 5 left has been used for the numerical investigations. Simulations have been performed from needle opening until needle closing for several injection timings and rail pressures representing an engine part load case. At this load, the needle does not reach its maximum opening. For higher rail pressures, the needle opens only to about 25% of the maximum lift. Measured needle lift curves and the measured static pressure before needle seat were used as boundary conditions for the CFD simulations. At the nozzle outlet, the chamber pressure was set as boundary condition. As fluids marine diesel oil (MDO, ρ = 832 kg/m^3 , μ = 3.4 mm^2/s) and heavy fuel oil RMG32 (ρ = 927 kg/m^3, μ = 89.5 mm^2/s at 25°C) have been used. For the simulation of the spray in the spray chamber, a fine mesh of about 120.000 cells has been used, see figure 5 right.

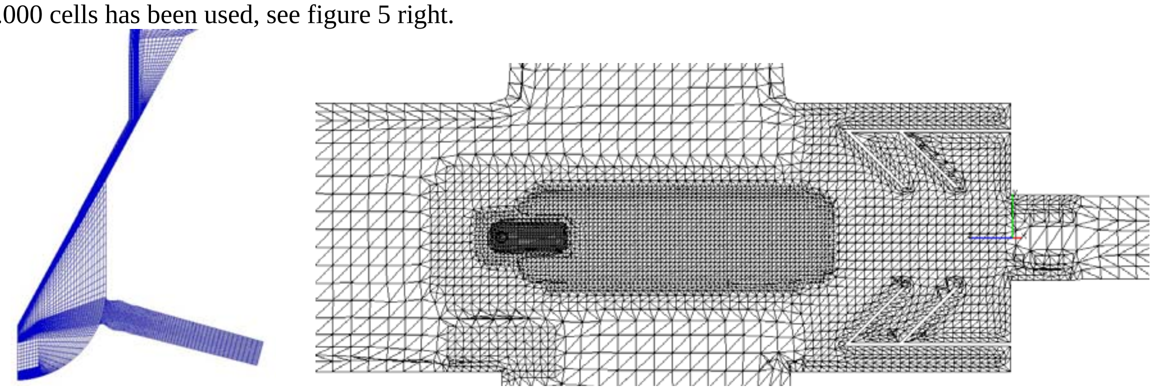

Figure 5: Nozzle geometry and computational mesh (left), detail of the computational mesh used for the spray simulations (right)

The investigated injections correspond to a 25% load point at an associated engine. During engine run-up or loaded acceleration, this load point shows the highest soot emissions. It is examined, how different rail pressures affect the nozzle flow and spray formation. A rail pressure of 60, 100 and 140 MPa is investigated with a fuel volume of 500 mg per stroke for each rail pressure. Due to the different rail pressure, the needle opening and needle lift curves are different. With lower rail pressure, the injection duration is longer and the maximum needle lift is higher. For all cases, the limit of maximum needle lift is not reached. During the first passage of needle opening (0.1 ms), flow is characterized by high transients and cavitation. After this first period a cavitation zone occurs at the bottom side of the nozzle for the investigated rail pressures, see figure 6 left. The simulations show that the fluid flows into the nozzle hole from the bottom of the sack hole. This is caused by the high flow velocities in the needle seat area during needle opening. At higher needle lifts, the flow and the cavitation zone switches to the upper side of the nozzle, see figure 6 right. The flow velocity in the needle seat area decreases and the flow turns its direction more or less directly to the nozzle hole.

The numerical investigations were carried out for Diesel fuel oil (DFO) and heated heavy fuel oil (HFO). The fluid properties were taken from the FIRE property database [11]. For both simulations, identical needle lift curves were used, as the measurement of the needle lift curves with HFO was difficult. The flow and cavitation behaviour is similar; due to the higher viscosity of HFO the switching of the flow conditions in the nozzle holes was delayed.

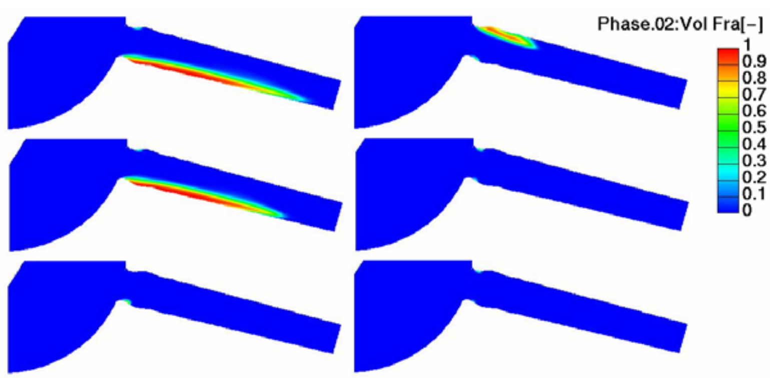

Figure 6: Nozzle cavitation at 0.15 mm needle lift (left) and 0.3 mm needle lift (right) for a rail pressure of 140 MPa (top), 100 MPa (middle) and 60 MPa (bottom)

The flow velocity, phase fraction as well as turbulent kinetic energy and dissipation rate at the nozzle exit during the nozzle flow simulation are stored in a file and are then used as the input for the primary break-up model during the spray simulation, as explained in figure 4. The start of injection was calculated from the needle lift curves and the signals from the injection rate analyzer. Figure 7 shows the comparison of the simulated spray droplets with pictures taken with the ICCD camera. The comparison is done at different timesteps for rail pressures of 60, 100 and 140 MPa. The higher rail pressure increases the momentum on the droplets and the spray penetration despite the shorter injection duration. The influence of the rail pressure on the spray angle is small. This shows that the volume increase with higher rail pressure is mainly reached by a higher spray penetration.

Figure 7 (right) compares the calculated spray penetration length for different rail pressures at 1.4 MPa chamber pressure with the measured curves. The simulations were performed for Diesel fuel oil (DFO). For the primary and secondary break-up, identical settings were used. A higher rail pressure leads to a higher penetration length over the whole injection period. The simulation results match well with the measurement.

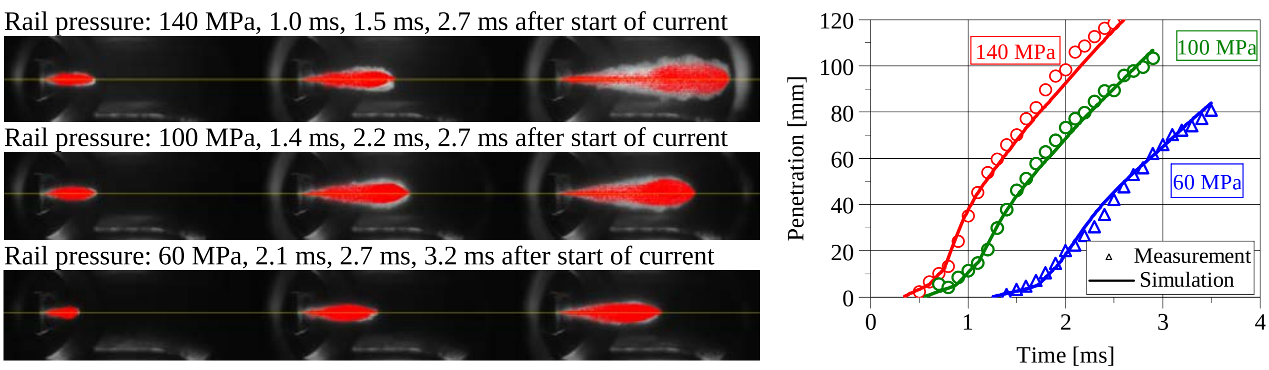

Figure 7: Calculated and measured spray (left); droplet penetration for 60 MPa, 100 MPa and 140 MPa rail pressure and DFO cold chamber (right)

Figure 8 gives an example of measurement results under hot conditions using an inert nitrogen atmosphere. On the left, a spray picture taken with combined Schlieren/Scatter light-bypass technique is shown at a chamber temperature of 800 K. From the pictures, the gaseous and liquid penetration length can be measured. The right side of figure 8 compares measured and liquid penetration of liquid and gaseous phase. The liquid droplets reach a constant maximum penetration of about 80 mm for all rail pressures, which is also shown by the CFD simulations.

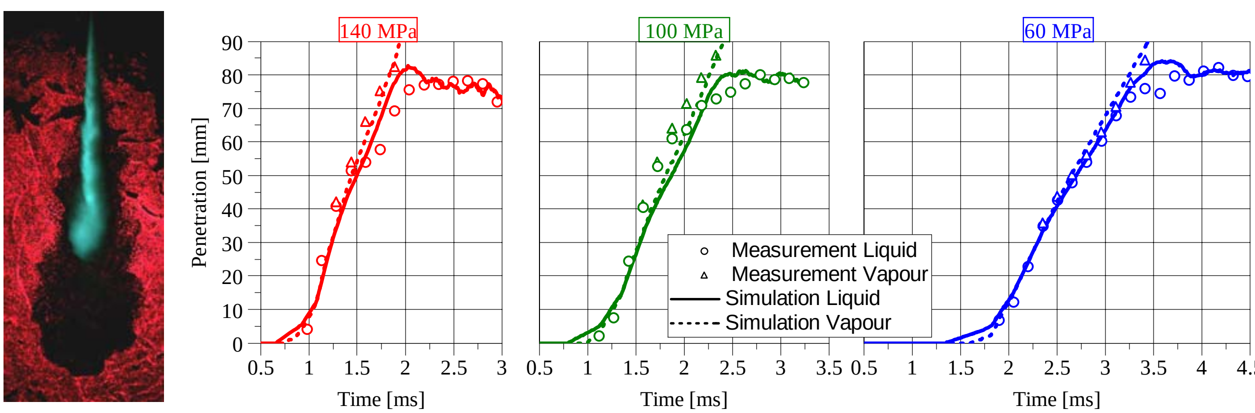

Figure 8: Picture taken with combined Schlieren/Scatter light bypass technique (left); Calculated vs. measured liquid and gaseous penetration for 60 MPa, 100 MPa and 140 MPa rail pressure, chamber gas conditions 800 K and 3 MPa

Conclusions

Measurements of fuel injection sprays of modern, single circuit Common-Rail injectors for large marine diesel engines have been performed in an especially adapted high pressure chamber. These measurements were used to (1) analyse the dependence of spray parameters on boundary conditions and (2) to validate injection spray simulation models with respect to large injector nozzles and different fuels. For the simulation, a coupled approach of a nozzle internal simulation considering real geometries and a subsequent spray calculation was done. Measurements and calculations clearly show similar results and dependencies. Regarding the usage of heavy fuel oil, good correlation of experiment and simulation was found. Applying the presented coupled simulation approach, a further investigation of the impact of the wide range of marine fuel properties on spray characteristics can now be done. For hot chamber conditions, a combined Schlieren/Scatter light-bypass technique has been applied. The results of the comparison under hot conditions show good coincidence of measured and calculated spray penetration, both of the gaseous and liquid phase. In the future, further measurements and simulations at elevated temperatures have to be done in order to validate results and models in a wide parameter range for different marine fuels.

Acknowledgements

The authors wish to acknowledge the funding of the research project EMI-MINI and EMI-MINI II, “Emission Reduced Ship Propulsion Systems” by the German Federal Ministry of Economics and Technology (BMWi).

References

[1] Köhler, H.: “NO_x-Emissionen aus der zivilen Schifffahrt“, MTZ 12/2003, S. 1040 ff
[2] International Maritime Organisation, MP/CONF. 3/35, Annex “… Emission of Nitrogen Oxides …”, 1997
[3] Buchholz, B.; Pittermann, R.; Niendorf, M.; “Measures to Reduce Smoke and Particulate Emissions from Marine Diesel Engines using Compact Common Rail Injectors”, CIMAC-Conference, 2007
[4] Frobenius, M.; Pittermann, R.: “Untersuchung der Wirksamkeit von Rußminderungsmaßnahmen bei mittelschnelllaufenden Schiffsdieselmotoren mittels optischer Methoden und CFD“; 8. Internationales Symposium für Verbrennungsdiagnostik, Baden-Baden, 2008
[5] Schmidt, R.; Schneider, H.: “Diesel-Einspritzung – Status und Trends der Zukunft”, Schiff & Hafen, 09/08
[6] Bosch, W: “Der Einspritzgesetzindikator, …“; Motortechnische Zeitschrift, MTZ 07/1964
[7] Fink, C.; Buchholz, B.; Niendorf, M.; Sadlowski, T.; Harndorf, H.: “Einfluss der Kraftstoffqualität auf das Betriebsverhalten moderner schweröltauglicher CR-Injektoren für Schiffsdieselmotoren“, MTZ-Heavy-Duty Conference, Bonn, Sept. 2008
[8] Buchholz, B.: “Analysis of Injection Sprays from Heavy Fuel Oil Common-Rail Injectors for Medium-Speed Diesel Engines”, Dissertation, Universität Rostock, 2008
[9] Fink, C.; Buchholz, B.; Niendorf, M.; Harndorf, H.; “Injection spray analyses from medium speed engines using marine fuels“, ILASS EUROPE 2008
[10] Plesset, M.S.; Zwick, S.A.: “The growth of a vapour bubble in superheated liquids”, Journal of Applied Physics 25 (1954), S. 493-398
[11] FIRE Manual “Eulerian Multiphase Flow”, 2008, AVL List GmbH
[12] Bianchi, G.M. and Pelloni, P. “Modelling the Diesel Fuel Spray Break-up by Using a Hybrid Model”, SAE Paper 1999-01-0226, (1999).
[13] Liu,A.B.;Reitz,R.D. "Modeling the Effects of Drop Drag and Break-up on Fuel Sprays", SAE 930072, (1993).
[14] Abramzon, B. and Sirignano, W. A. "Droplet Vaporization Model for Spray Combustion Calculations", AIAA 26th Aerospace Sciences Meeting, 1988.
[15] Brenn, G., Deviprasath, L.J. and Durst, F. “Computations and Experiments on the Evaporation of Multi-Component Droplets”, Proc.9th Int.Conf.Liquid Atomiz.Spray Syst. (ICLASS), Sorrento (Italy), July 2003
[16] Frolov, S.M., Frolov, F.S. and Basara, B. “Simple Model of Transient Drop Vaporization”, Journal of Russian Laser Research, Volume 27, Number 6, 2006