Numerical Simulation of Unsteady Nozzle Cavitation and Spray Formation under Diesel Engine Conditions

Mikhail Konstantinov & Claus Wagner[1]

German Aerospace Center (DLR)
Institute for Aerodynamics and Flow Technology,
Bunsenstr.10, 37073 Göttingen, Germany
Email: claus.wagner@dlr.de
Phone: +49 (0)551 709 2261
Fax: +49 (0)551 709 2404

Keywords
Diesel engine – Nozzle – Cavitation – Atomization – Spray

Abstract
The quality of atomization of liquid fuel is probably the most essential factor of the reducing emissions in Diesel engines. The atomization process depends among other things on the velocity of the fuel exiting the injection nozzle and on the extent of cavitation within the nozzle. Exit velocity modulations strongly influence the break-up processes, the spray penetration and also the droplet-droplet and spray-wall interactions.

Since the considered nozzles are extremely small and the fuel flow is rather fast, the experimental investigation of the above described processes faces lot of difficulties. On the other hand, numerical studies are difficult as well due problems associated with the modeling of cavitation. On of these problems is that the values of density in cavitating flows can change several orders of magnitude between neighboring cells within a computational mesh. The difficulty of resolving this practically discontinuous change in density can cause severe numerical oscillations in the calculated density field. Despite of these complications there have been some successes in the study of cavitation nozzles flow. Experiments with transparent nozzles (Chaves and Obermeier [1]) clearly demonstrate that even at a supply pressure of more than 200 bar cavitation may disappear for short times provided that the pressure increases rapidly. Conversely, cavitation extends more than expected when the supply pressure decreases fast enough. Moreover, the atomization of the liquid jets is heavily affected by the disappearance and the reoccurrence of cavitation.

For the numerical investigation of the processes described above we set up a numerical model based on a second order accurate finite volume method and simulated the unsteady atomization process of liquid fuel in Diesel engines. This model will be presented together with the obtained numerical results and a comparison with results of corresponding measurements. Further, the interdependency of transient nozzles flow and spray formation will be discussed. For that purpose the prediction of unsteady nozzle flow was performed based on two models needed to simulate the transient behavior of cavitation and two-phase atomization process. For the calculation of cavitation the model of compressible liquid with artificial barotropic equation of state [2, 3] is employed. Examples of different cases of transient flow through various 3D nozzle shapes (one- and multihole nozzles) and the corresponding spray development will be presented as well (Figs. 1-2). It will be shown, that the computed characteristics of the nozzle flow and spray structures are in good quantitative agreement with the experimental observations of the flow in high speed diesel injectors.

[1] Corresponding author: Claus Wagner, email: claus.wagner@dlr.de

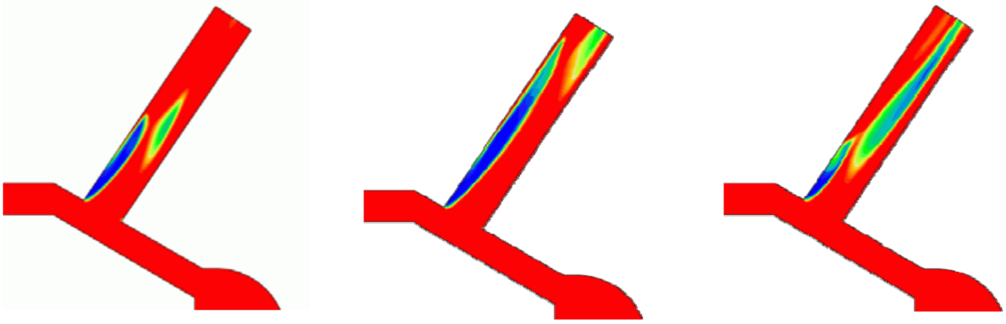

Fig. 1: Transient density behavior through nozzle cross-sections, where red color is liquid and blue is vapor. Intermediate colors are mixed.

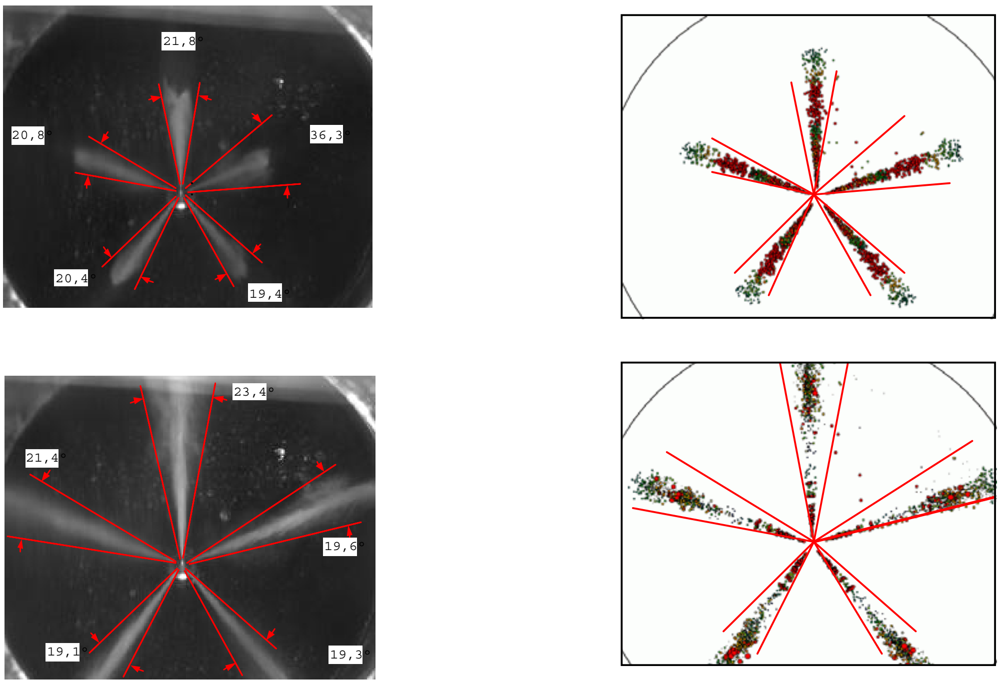

Fig. 2: Development of the structures of a Diesel spray injected in to the cylinder obtained experimentally (left) and numerically (right) for two points in time.

References

[1] Chaves, H. and Obermeier, F. (1996): Modelling the effect of modulations of the injection velocity on the structure of diesel sprays, SAE Paper No. 961126.
[2] Schmidt, D. P., Rutland, C. J. and Corradini, M. L. (1997): A numerical study of cavitating flow through various nozzle shapes. SAE Paper No 971597.
[3] Konstantinov, M. and Obermeier, F. (1999): Numerical simulation of transient nozzle flow, ZAMM **79**.