Oil & Gas Science and Technology – Rev. IFP, Vol. 54 (1999), No. 2, pp. 223-226

Simulation of Cavitating Flows in Diesel Injectors

M. von Dirke[1], A. Krautter[1*], J. Ostertag[1**], M. Mennicken[2] and C. Badock[2***]

1 Robert Bosch GmbH FV/FLP Postfach 106050 D-70546 Stuttgart – Germany
e-mail: vdirke@klarinette.flp.sh.bosh.de
2 Robert Bosch GmbH FV/SLE3 Postfach 106050 D-70049 Stuttgart – Germany

* now at Robert Bosch GmbH, K1/ESC3
** now at Universität Stuttgart, Institut für Strömungsmechanik
*** now at Robert Bosch GmbH, BaW/FVD-BK

Résumé — Simulation des écoulements en cavitation dans les injecteurs Diesel — Avec un nouveau modèle à deux fluides, il est possible d'effectuer des calculs tridimensionnels de mécanique des fluides numérique pour des écoulements en cavitation dans les composants hydrauliques des systèmes d'injection Diesel. Des géométries tests comprenant un clapet à bille, un nez à sac et un nez monotrou ont été utilisées pour vérifier si la méthode est applicable. Les calculs de fraction volumique moyenne montrent la distribution des zones de cavitation. Des calculs d'efforts ont mis en évidence un bon accord avec l'expérience. Ce nouvel outil permet d’améliorer la conception de nouveaux composants.

Mots-clés : cavitation, injecteurs, Diesel, modélisation.

***Abstract — Simulation of Cavitating Flows in Diesel Injectors** — With a new two fluid model it is possible to carry out three-dimensional CFD calculations of cavitating flows in hydraulic components of Diesel injection systems. As model geometries a ball valve, a sac-hole nozzle and a one-hole nozzle have been used to test the applicability of the method. Calculations of the ensemble averaged volume fraction show the distribution of cavitation zones. Force calculations were in good agreement with the experiment. With this new tool, the design of new components can be improved.*

Key words: cavitating flows, injectors, Diesel, modeling.

SYMBOLS

- $\mathbf{u}$ velocity vector
- p pressure
- α averaged volume fraction
- Γ mass exchange term
- τ_x turbulent fluctuations
- M momentum exchange term
- N bubble number density
- R bubble radius
- δ_{max} maximum radial shift.

INTRODUCTION

Cavitation, i.e. the “cold” vaporization of the liquid due to a drop in static pressure, has a major influence on the performance of Diesel injection systems. The collapse of cavitation zones may cause mechanical damage. On the other hand, it may also enhance spray breakup. For the further development of efficient injection systems, control of cavitation behavior is essential.

Cavitation is a very complex physical process that is not yet fully understood and still an active area of basic research. Nevertheless, recent developments in physical modeling and numerical methods have made it possible to simulate three-dimensional cavitating flows in components of injection systems.

In this paper, we present first results of steady-state three-dimensional calculations in model geometries.

1 TWO FLUID MODEL

In order to describe the cavitating flow a two-fluid model is used [1]. In the two fluid finite volume approach liquid and vapor phase are treated separately. For each phase, a transport equation for the ensemble averaged volume fraction α_i is used. The volume fraction can be interpreted as the probability of the phase i being in a volume element of the numerical grid. The two transport equations are coupled by interaction terms, which account for mass and momentum exchange. The momentum equation for one phase (the phase index has been omitted) can be written as [1]:

$$\frac{\partial(\alpha\rho\,\mathbf{u})}{\partial t} + \nabla\cdot(\alpha\rho\,\mathbf{uu}) = -\alpha\nabla p + \nabla\cdot\alpha\tau_x + M + \mathbf{u}\Gamma$$

This is a Navier-Stokes equation weighted by the volume fraction and including the exchange terms M and Γ. The momentum exchange term M is based one the drag law for a single sphere [1]. It is also necessary to include a k-ε based modeling of the tensor τ_x which is correlated with the turbulent fluctuations. Due to the mass exchange term Γ, the continuity equation becomes:

$$\frac{\partial(\alpha\rho)}{\partial t} + \nabla\cdot(\alpha\rho u) = \Gamma$$

In this approximation the mass exchange term is modeled with the help of the asymptotic Rayleigh equation, i.e. without the inertia term, which leads to:

$$\Gamma = N\rho\ 4\pi R^2\frac{\mathrm{d}R}{\mathrm{d}t}$$

with the bubble number density N as a model constant.

The two-fluid model emphasizes the macroscopic three-dimensional flow behavior. The microscopic cavitation model is hidden in the exchange terms. As a macroscopic quantity, the volume fraction contains no specific information one the shape or size of the cavitation entities in a volume cell. It is only assumed, that the momentum and mass exchange between the liquid and the vapor is similar to Rayleigh bubbles. It is obvious that this is a strong simplification of the cavitation process, justified mainly by computer constraints and the lack of a general theory of cavitation. More elaborated non-equilibrium theories describe only certain types of cavitation, like traveling bubbles or hydrofoil cavities, and have to simplify e.g. the nucleation process or the breakup of cavitation zones.

2 EXAMPLES

As an example, the flows in three model geometries, a ball valve, a sac-hole nozzle and a one-hole nozzle have been calculated. A steady pressure difference has been used as a boundary condition and the needle lift has been kept constant.

2.1 Cavitation in a Ball Valve

A typical ball valve geometry is shown in Figure 1. The counter pressure was 10 bar and the inlet pressure was varied. Because of the rotational symmetry of the valve, a quasi two-dimensional grid (one layer sector) could be used.

The cavitation develops at the narrowest section of the valve seat. With higher inlet pressures the cavitation zone extends further downstream. The calculated force on the valve needle agrees very well with the experimental values as shown in Figure 2 (from [2]).

Calculated with a usual single-phase CFD code, the force is strongly underestimated due to the development of large zones with negative pressures.

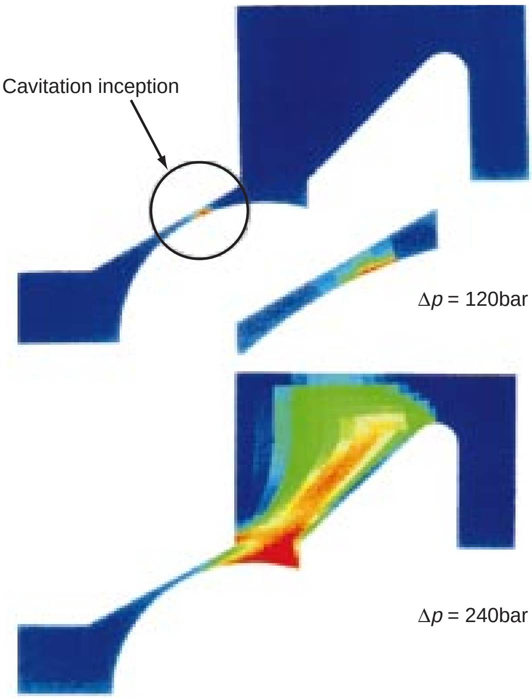

Figure 1

Cavitation in a model ball valve. Depicted is the volume fraction at different inlet pressures.

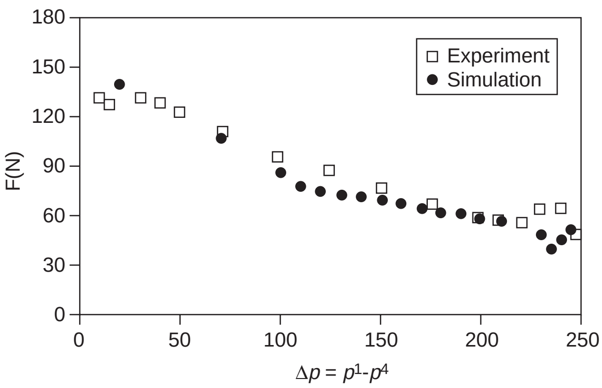

Figure 2

Cavitation in a model ball valve. Force on valve needle as a function of a steady pressure difference.

2.2 Cavitation in a Sac-Hole Nozzle

Figure 3 shows the cavitation zones in a sac-hole nozzle in a radial and axial cut through the spray hole.

The nozzle has four holes with different angles. A grid sector of 45° was used for the calculation. Inlet pressure was 1000 bar, outlet pressure 50 bar with a lift of 0.35 mm. Figure 3 shows the cavitation zones in the sac-hole nozzle for two different spray hole angles with 50° and 80°. Generally, the cavitation zone covers the upper half of the hole wall and extends deeper into the spray hole with a larger angle. In addition, it could be shown that adjacent spray holes with different angles influence each other.

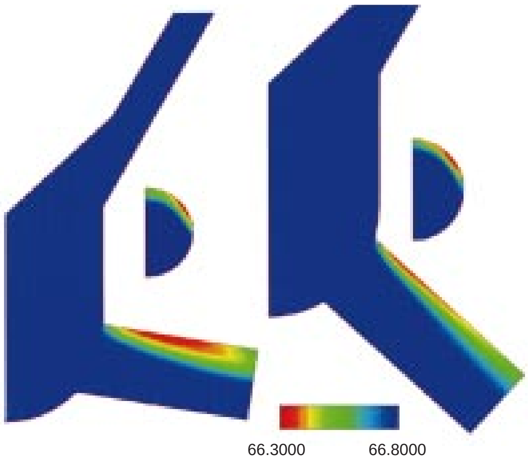

Figure 3

Cavitation in a sac-hole nozzle. Influence of spray-hole angle.

2.3 Cavitation in a One-Hole Nozzle

Experiments were carried out with a transparent one-hole nozzle as shown in Figure 4. In such small geometries, the development of cavitation films can be observed [3].

Looking at only a single picture it is difficult to distinguish the cavitation films from geometry shadows, due to the varying optical thickness near the wall of the spray hole. Therefore, in [3] transient high-speed photography has been used. The simulation with an axial needle yields only small cavitation zones (Fig. 5). In small geometries, viscous effects dominate the cavitation development. Wall nuclei, adhesion and surface tension become more important for the mass exchange than in bubble cavitation. Generally, the two-fluid model overesti-mates the condensation rate in such situations. Despite its simplicity, the two-fluid model is very useful in studying the influence of geometric variations on cavitation zones [5].

In Figure 5 the needle has been shifted about $\delta = (2/3)\delta_{max}$. This leads to an asymmetric distribution of the cavitation zone, which is also seen in the experiment. Despite the drastic change in the volume fraction, the flow rate changes only about 5%. In this case, the system is stable and insensitive to an off-centered needle axis. Changes in the distribution of the volume fraction do not affect a global integral quantity like the flow rate.

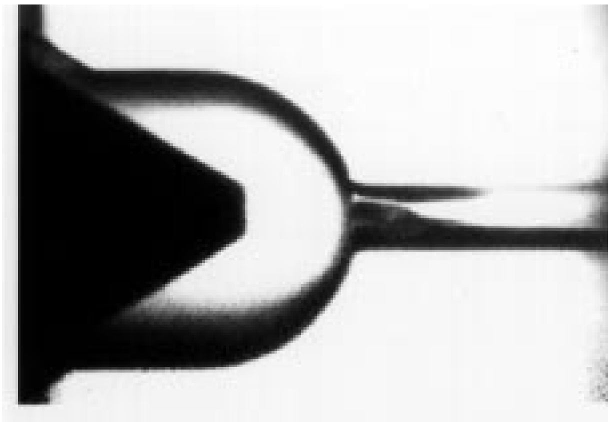

Figure 4

Cavitation in a one-hole nozzle visualized by a shadowgraphe. The spray-hole diameter is 0.2 mm, the needle lift 50 µm and the inlet pressure 25 bar.

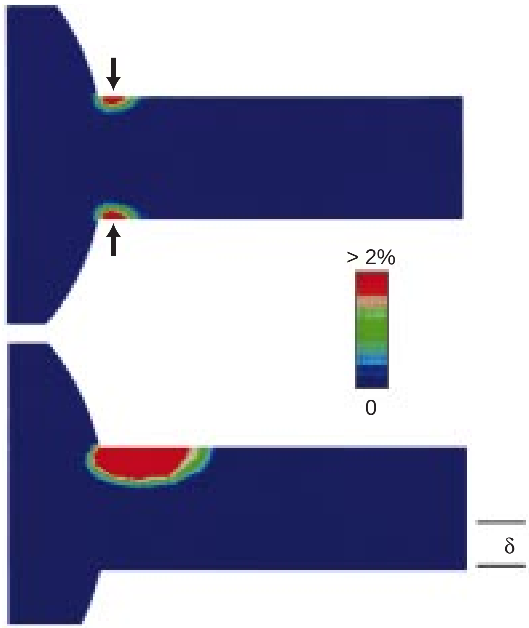

Figure 5

Cavitation in a one-hole nozzle. Influence of a radial shift of the needle on cavitation zones.

CONCLUSION

With a new two-fluid model, it is possible to carry out three-dimensional CFD calculations of cavitating flows in hydraulic components of Diesel injection systems. As model geometries a ball valve, a sac-hole nozzle and a one-hole nozzle have been used. Calculations of the ensemble averaged volume fraction show the distribution of cavitation zones. Force calculations were in good agreements with the experiment. Despite the inherent limitations and simplifications, the two-fluid method helps to improve the design of new components and can be employed to reduce development times. Further improvements have to include unsteady boundary conditions. The method should also be extended to deal with mesh movements and fluid-structure coupling.

ACKNOWLEDGMENT

This study is part of a common project with *AVL/Graz*. We would like to thank Dr. Reinhard Rieger, Dr. Herwig Grogger and Dr. Ales Alajbegovic from *AVL*. We would also like to thank our collegues at *Bosch*, Dr. Barbara Heine, Dr. Walter Egler and Peter Böhland.

REFERENCES

1 Grogger, H.A., Alajbegovic, A. (1998) Calculation of the Cavitating Flow in Venturi Geometries Using Two-Fluid Model. *Proceedings of FEDSM'98,* 5295 Washington.

2 Schautt, M. (1995) Messungen an Diesel-Reiheneinspritzpumpen. *Diplomarbeit Uni*, Stuttgart.

3 Krautter, A. (1996) Numerische Simulation der Zweiphasigen Innenströmung einer Dieseleinspritzdüse. *Diplomarbeit FH*, Heilbronn.

4 Badock, C., Wirth, R., Fath, A., Leipertz, A. (1998) Application of Laser Light Sheet Techniques for the Investigation of Cavitation Phenomena in Real Size Diesel Injection Nozzles. *ILASS-Europe,* Manchester.

5 Ostertag, J. (1997) Berechnung der Kavitierenden Innenströmung einer Einlochdüse. *Diplomarbeit Uni*, Stuttgart.

Final manuscript received in March 1999