ILASS-Europe 2002 *Zaragoza 9-11 September 2002*

PRIMARY BREAK-UP MODEL FOR DIESEL JETS BASED ON LOCALLY RESOLVED FLOW FIELD IN THE INJECTION HOLE

E. v. Berg*, A. Alajbegovic, D. Greif*, A. Poredos*, R. Tatschl*, E. Winklhofer* and L.C. Ganippa*****

E-mail: Eberhard.von.Berg@avl.com

**AVL List GmbH, Hans-List-Platz 1, A-8020 Graz, Austria*

***AVLPowertrain Engineering Inc., 47519 Halyard Drive, Plymouth, MI 48170-2438, USA*

****Department of Thermo and Fluid Dynamics, Chalmers University of Technology, Sweden*

Abstract

A primary breakup model has been developed based on locally resolved properties of the cavitating nozzle flow. According to this model the turbulent length scale determines the droplet size. A mixed aerodynamic/turbulent approach is used to calculate the break-up rate. The droplets are released from the surface of a coherent liquid core calculated from a mass balance. The model delivers the initial droplet size and velocity distribution for the discrete droplet model. The characteristic feature of the results from the model is a remarkable asymmetry of the spray shape. The spray direction is deviated towards the direction where cavitation is more pronounced with largest spray angle in the plane of maximum cavitation. These results are confirmed by recent experimental findings. The model has been implemented into the FIRE CFD-code and applied for Diesel injection. Validation of the model is presently done using experimental data from AVL and Chalmers University showing both, cavitation and spray formation simultaneously.

Introduction

Reliable break-up models for Diesel jets are an important prerequisite for simulation of mixture formation in IC engines. Especially the initial stage of droplet formation near the injector is not yet fully understood. Different ideas on the break-up mechanism are under discussion, e.g., break-up due to cavitation and turbulence effects originating from nozzle flow or aerodynamic break-up from self-exciting interaction of surface elevations and flow field outside the nozzle. The objective of the new primary break-up model presented here is to deliver the initial size and velocity distribution of the spray droplets linking spray and mixture formation with injection parameters and cavitating nozzle flow features. This will support optimization of injectors as well as of the overall mixture formation process.

Previous work

Turbulence induced break-up has been studied first by Huh and Gosman [1] and later by Bianchi and Pelloni [2], who calculated mean droplet size and production rate from estimated average turbulence properties of the nozzle flow. At AVL this idea has been extended by using local nozzle flow properties as well as additional turbulent kinetic energy (TKE) sources from cavitation bubble collapse to calculate locally distributed break-up rates in the orifice cross section (Tatschl et. al. [3]). This model assumed homogeneous initial droplet sizes of approximately nozzle diameter. The two-dimensional resolution has been achieved by random sampling from the flow conditions in the orifice calculated from a 3D calculation of the cavitating flow in the injection hole. In the present work this model has been extended by using release of droplets from a coherent liquid core region and introduction of additional couplings between bubble dynamics and turbulence model.

Model features

The overall model uses detailed information from 3D turbulent cavitating nozzle flow simulations performed with the FIRE CFD code based on a two-fluid approach [4,5]. In the two-fluid model separate sets of conservation laws are solved for the liquid and the gas phase. Inter-phase momentum transfer is described by taking into account drag forces as well as turbulent dispersion force. Drag formulation is based on single bubble drag transformed to the total drag force for a computational cell via bubble number density. The turbulent dispersion is described relating turbulent dispersion force to continuous phase TKE and dispersed phase volume fraction gradient. Mass exchange is calculated from rate of change for total vapor bubble mass due to evaporation or condensation. Bubble number density N''' is modeled using a simple correlation describing decay of number density with increasing volume fraction due to bubble collisions or film formation. Bubble dynamics is calculated from a simplified Rayleigh-Plesset equation [4]. The set of equations used is given in Table 1 below. Fig. 1 shows a representative picture of flow properties in the nozzle orifice cross section calculated from the two-fluid model.

Nozzle flow data calculated with the two-fluid model are introduced into the break-up model on a polar auxiliary grid in the orifice cross section, which allows to determine the desired resolution. As a first step in the break-up calculation averaging is done along the perimeter of different ring sections on the auxiliary grid yielding a mean

Table 1.	Model equations for cavitating nozzle flow		
Mass Conservation	$\frac{\partial \alpha_k \rho_k}{\partial t} + \nabla \cdot \alpha_k \rho_k \mathbf{v}_k = \sum_{l=1, l\neq k}^{2} \Gamma_{kl}$, k = 1,2	Compatibility relation for volume fractions	$\sum_{k=1}^{2} \alpha_k = 1$ with k = 1: vap, k = 2: liq
Mass exchange	$\Gamma_{12} = N''' \frac{\partial m_{bubble}}{\partial t} = \rho_1 N''' 4\pi R_{bubble}^2 \frac{\partial R_{bubble}}{\partial t} = -\Gamma_{12}$	Simplified Rayleigh equation	$\frac{dR_{bubble}}{dt} = \sqrt{\frac{2\Delta p}{3\rho_2}}$
Momentum Conservation	$\frac{\partial \alpha_k \rho_k \mathbf{v}_k}{\partial t} + \nabla \cdot \alpha_k \rho_k \mathbf{v}_k \mathbf{v}_k = -\alpha_k \nabla p + \nabla \cdot \alpha_k (\tau_k + \mathbf{T}_k^t) + \sum_{l=1, l\neq k}^{2} \mathbf{M}_{kl} + \mathbf{v}_k \sum_{l=1, l\neq k}^{2} \Gamma_{kl}$		
Momentum exchange	Drag force: $M_{D,12} = c_D \frac{6\alpha_1 \rho_2}{2R} \lvert v_2 - v_1 \rvert (v_2 - v_1) = -M_{D,21}$	Turbulent dispersion force	$M_{T,12} = c_{T,12} \rho_2 k_2 \nabla \alpha_1 = -M_{T,21}$

a) b) c)

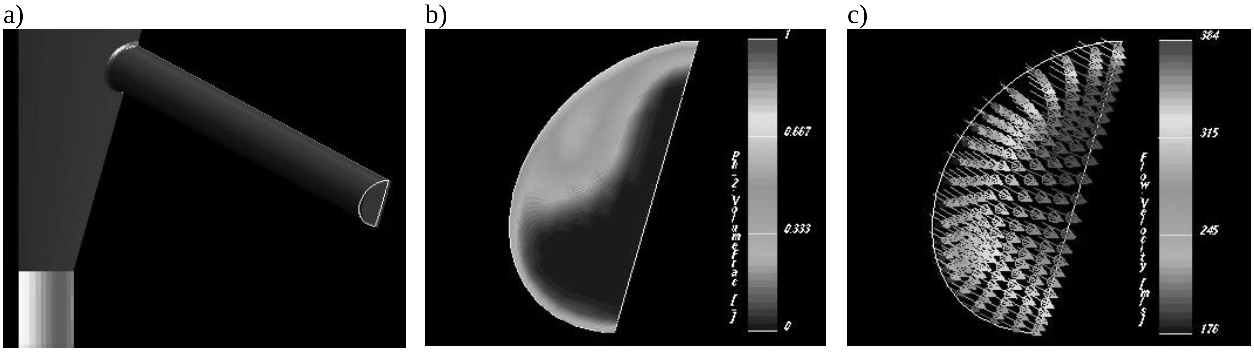

Figure 1. Results from cavitating nozzle flow calculation: a) Geometry, b) Vapor volume fraction, c) Liquid velocity at 87.5 MPa injection pressure.

break-up rate, which is dependent on nozzle hole radius. Thus the radial erosion of the liquid jet is determined from a mass balance and finally the coherent core length is calculated at the position where erosion is completed. To introduce the 2D structure of the flow the orifice data are projected to the core surface and the break-up model is applied locally at corresponding surface elements. The cell centers of the auxiliary grid projected onto the core surface also define the release positions for the initial droplet parcels. Figures 2a and 2b show the auxiliary grid as well as the release positions on the liquid core surface.

At the release positions according to [2] the turbulent length scale is taken to determine the atomization length scale and also the droplet size, while for the break-up time scale a hybrid value calculated from a weighted mean of turbulent and aerodynamic time scales is used. The aerodynamic time scale is calculated assuming break-up due to Kelvin-Helmholtz instability at the liquid jet surface. From atomization length and time scales the local break-up rate is calculated according to the set of equations in Table 2. Thus, finally the model delivers the initial droplet size and velocity distribution at the liquid core surface (see Figures 2c and 2d) as well as the initial spray angle needed for initialization of the representative droplet parcels within the discrete droplet model.

Vapor formation from upstream cavitation in the injection hole changes the flow field and its turbulence. Especially, the disturbances from collapsing vapor bubbles increase the turbulence level in the liquid and modify turbulent length and time scales which are decisive for the break-up model. Similar as in [3] these effects are treated as

a) b) c) d)

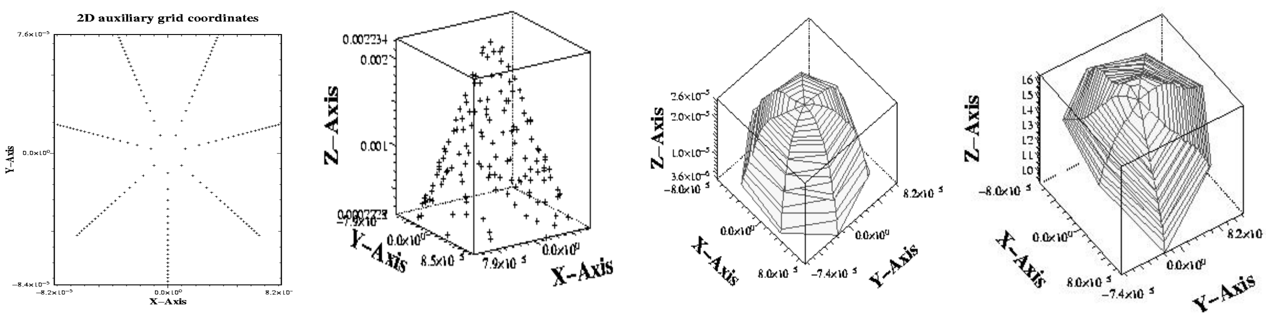

Figure 2. a) Auxiliary grid in nozzle orifice, b) Release positions [m] on liquid core surface c) Droplet sizes[m] in nozzle orifice d) Radial droplet velocities [m/s] in nozzle orifice

Table 2.	Model equations for break-up model				
Length scale:	$L_A = C_2 C_\mu \frac{k^{1.5}}{\varepsilon}$			Drop size:	$R_{drop} = L_A / 2$
Time scales:	Turbulent: $\tau_T = C_\mu \frac{k}{\varepsilon}$	Aerodynamic: $\tau_W = L_W \left[\frac{\rho_{amb}\rho_2 v_2^2}{(\rho_2 + \rho_{amb})} - \frac{\sigma}{(\rho_2 + \rho_{amb}) L_W} \right]^{-0.5}$			Mixed: $\tau_A = C_1 \cdot \tau_T + C_4 \cdot \tau_W$
Break-up rate:	$\frac{dR_{core}}{dt} = \frac{L_A}{\tau_A}$	Erosion rate:	$\frac{dR_{core}}{dz} = -\frac{\dot{M}(z)}{\rho_2 v_2}$	Relation with average break-up rate:	$\dot{M}(z) = \rho_2 \overline{\frac{dR_{core}}{dt}}(z)$

source terms in the 1D k-ε turbulence model coupled with the Rayleigh-Plesset equation governing bubble dynamics. Additional coupling terms considered in the new model are a correction-term p_E accounting for pressure fluctuations due to liquid turbulence, which enhances cavitation, and a mass transfer term, which may damp or enforce bubble dynamics. The most important model equations are given in Table 3. The solution for this ordinary differential equation system is gained using a solver from the ODEpack library for stiff non-linear systems.

This model is applied within the liquid core region for fluid elements until the liquid core surface is reached. If source terms are weak, this causes decay of turbulence and thus also of atomization strength along the core. If sources from bubble collapse are strong, the break-up can also be enforced. Due to the broad range of flow conditions varying locally over the orifice cross section and also with time during injection all possibilities may occur, as has been found by sample calculations with FIRE.

Table 3.	Model equations for turbulence and bubble dynamics model
Turbulence	$\dot{k} = S - \varepsilon, \quad \dot{\varepsilon} = \frac{c_2 \varepsilon (S - \varepsilon)}{k}$ with source term $S = \left| 2\pi c_B N'''R^2 \dot{R}(3\dot{R}^2 + 2R\ddot{R}) \right|$
Rayleigh equation	$\dot{R} = \frac{v}{\left(1 - \frac{\rho_1}{\rho_2} i_{e/c}\right)}, \quad \dot{v} = \frac{1}{R}\left[\frac{(p_{sat} - p + p_E)}{\rho_2} - 1.5v^2 - \frac{4\upsilon_2}{R} v - \frac{\dot{m}}{2\pi R^2 \rho_2} v \right]$ with $p_E = C_E \frac{2}{3} \rho_2 k_2$
Mass transfer	$\dot{m} = 4\pi R^2 \rho_1 \left| \dot{R} \right| i_{e/c}$ with $i_{e/c} = -1,0,1$ depending on condensation or evaporation

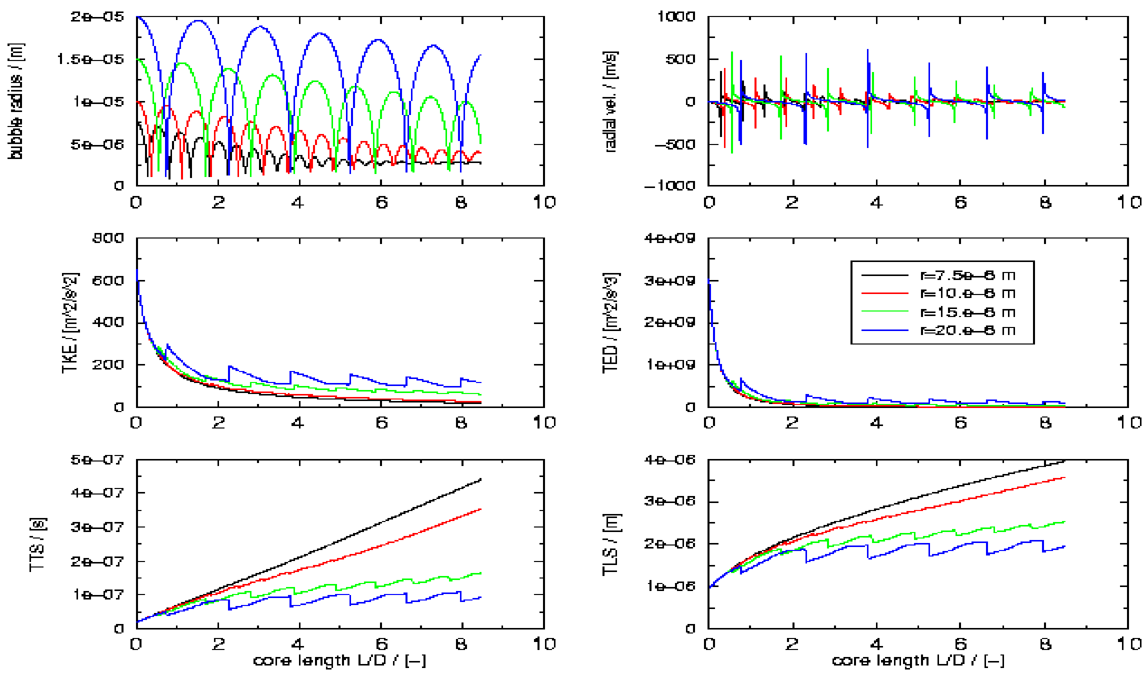

Figure 3. Results from bubble dynamics model coupled with 1D turbulence model for different bubble sizes : bubble radius and radial velocity, turbulent kinetic energy (TKE) and energy dissipation (TED) as well as turbulent time scale (TTS) and length scale (TLS) drawn versus core length normalized with nozzle diameter

Results

The model has been tested as stand-alone program to study basic effects and also as submodel implemented into the FIRE code. Results from the separate calculations without activating the source terms in the turbulence model show increase of turbulent length and time scales along the liquid core causing increasing primary break-up droplet sizes in downstream direction independent from bubble size. With turbulence sources the additional production of turbulence from bubble dynamics causes reduction of turbulent length scales. This is most pronounced for large bubbles. Since in the orifice cross section regions with and without bubbles are found, both effects may occur in coupled calculations. Typical results for bubble dynamics and their effects on turbulence are shown in Figure 3.

Results from FIRE have been calculated for an injector with linear increase of pressure up to 120 MPa and injection into 10 MPa atmosphere at 293 K with and without consideration of additional turbulence sources. The overall results for the shape of the spray show a remarkable asymmetry of the spray shape in both cases. The spray cone angle in the symmetry plane of the cavitating nozzle flow (defined by direction of injector and injection hole axis) is larger than in the perpendicular plane. Further the spray is dispersed towards the direction where cavitation is more pronounced (see Fig. 4). The overall distribution of droplet sizes and velocities is shown in Fig. 5 for cases with and without taking into account turbulence sources from bubble collapse. It is found that according to the effects of modified length scales the droplet size distribution is broader with inclusion of turbulence sources. A detailed analysis of the droplet size distribution in different cross sections below the nozzle shows larger drops on the side, which is opposite to the cavitation region within the nozzle, and smaller drops, i.e. better atomization, on the cavitation side.

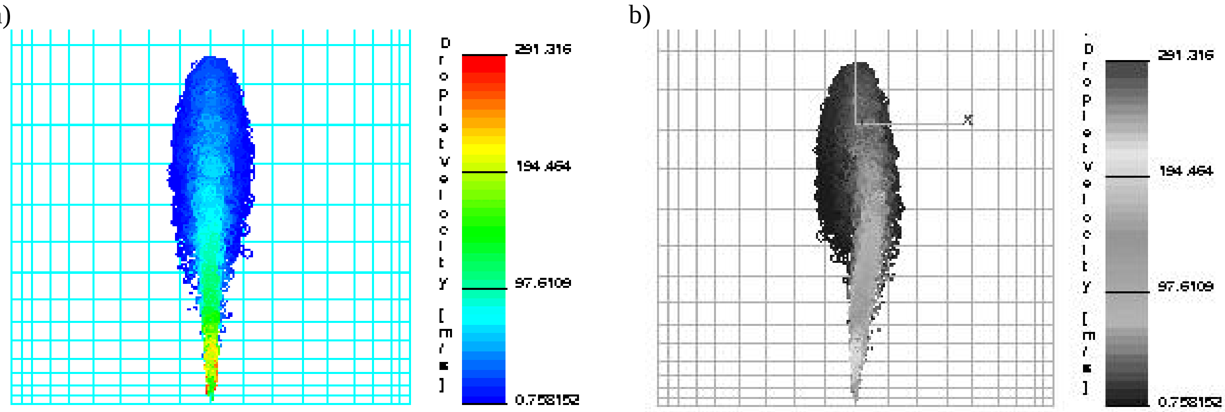

Figure 4. Appearance of spray cone under perpendicular views and droplet velocities: a) front view, b) side view

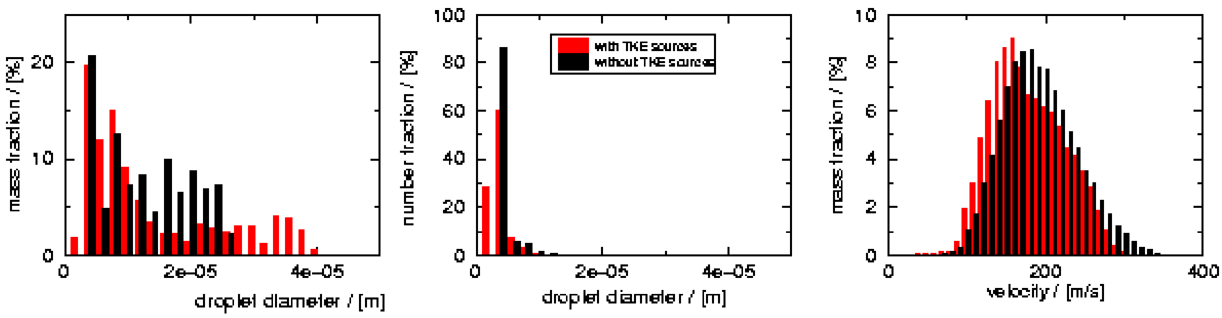

Figure 5. Example for droplet size and velocity distribution calculated with the new break-up model implemented in FIRE: comparison of model without and with additional turbulence source terms from bubble break-up

Results for the Sauter mean diameter of the spray show that the primary break-up model applied is not much influenced by the injection pressure, if core length is short and TKE sources from bubble collapse are weak. Otherwise the increase of turbulent length and time scales along the core introduces also a dependency on injection pressure. In any case the final size distribution is determined by further secondary break-up and/or collision processes. However, the initial size reduction due to primary break-up is expected to yield more realistic spray penetration behaviour compared to the method of 'blob injection' starting with initial droplet sizes of approximately nozzle diameter. But, further calculations and comparisons with experimental data are required to consolidate the results and to validate the model.

Experimental analysis of cavitation and spray formation

The cavitation structure and its influence on the spray pattern has been investigated at Chalmers University in transparent nozzle having a hole perpendicular to the nozzle axis using high speed camera and stroboscopic visualization. Observations revealed that during the inception stage, cavitation bubbles were seen in the vortices at the boundary layer shear flow and outside the separation zone of nozzle hole. Cavitation bubbles grew intensively in the shear layer and developed into cloud-like coherent structures when viewed from the side of the nozzle hole [7]. The instabilities of the shear layer and the re-entrant jets caused the coherent cloud cavitation structures to break off, which subsequently leads to shedding of the cavitation cloud. The shed cavitation is similar in shape to a horseshoe vortex. In all these conditions the jet appeared symmetric when viewed from front and side. As the flow was increased further, the cloud-like cavitation bubbles developed into a dense large-scale cavitation cloud extending downstream to the hole. Until this stage the spray appeared symmetric. When the flow was increased beyond this stage, a glossy sheet cavitation extended from the hole entrance to the hole exit, occupying a significant part of the hole on one side. This non-symmetric distribution of cavitation within the hole resulted in a jet, which atomized on the side where more cavitation was distributed and a non-atomizing jet on the side with less cavitation. Any further increase in flow resulted in a total sheet cavitation structure within the nozzle hole and the same asymmetric pattern of the jet was observed when viewed from the side.

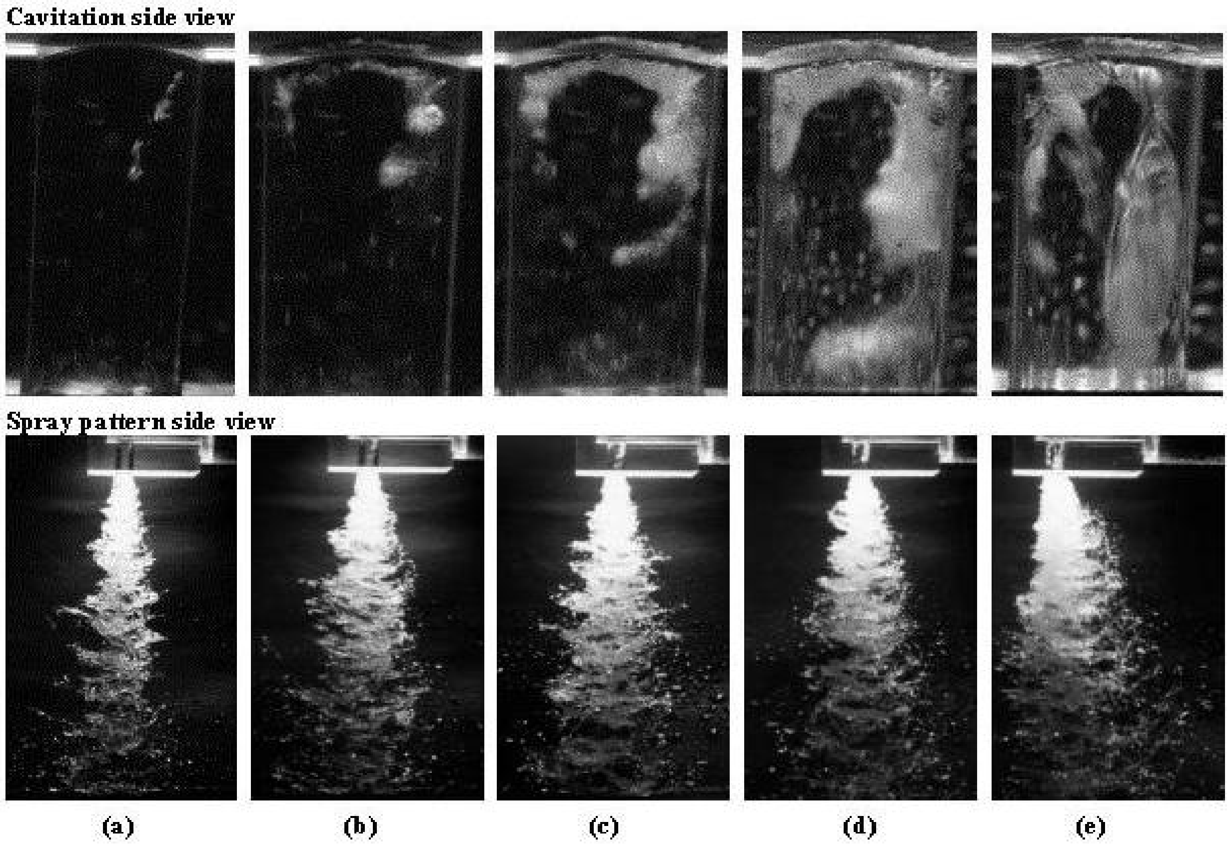

Figure 6. The dispersion of the spray under different cavitating conditions

The jet appeared to be symmetric when viewed from side under condition (a) to (d) in Figure 6, while in stage (e) a clear asymmetry was observed in the jet pattern where cavitation was distributed more to the right side of the hole compared to its left side. Transition from symmetric to asymmetric jet occurs due to the unsymmetrical distribution of cavitation within the hole of the nozzle.

Model validation

The results on asymmetric spray shape are confirmed by recent experimental findings at AVL and at Chalmers University. The AVL experiments are performed in real size geometry and under realistic injection conditions, however, in 2D planar nozzles for easy optical access [6]. The experimental analysis at Chalmers University [7] is done using an upscaled injector model in 3D with approximately 5-mm injection hole diameter operated with water at Reynolds numbers of 55000 to 80000. Both experiments clearly show the phenomenon of increased spray angle at the side with increased cavitation inside the nozzle (see Fig. 7a and b). Quantitative validation of the model is presently done for the 3D case in cooperation with Chalmers University. This has been started by doing calculations with the FIRE8 two-fluid model for the cavitating nozzle flow to deliver the flow field properties in the nozzle orifice as starting conditions for the primary break-up model. First results (see Fig. 7c) already show the basic features of the experimental vapor distribution, however, with vapor production still somewhat underpredicted. Next steps will be to apply the break-up model for these conditions and adjust model parameters by comparison with experimental findings.

a) b) c)

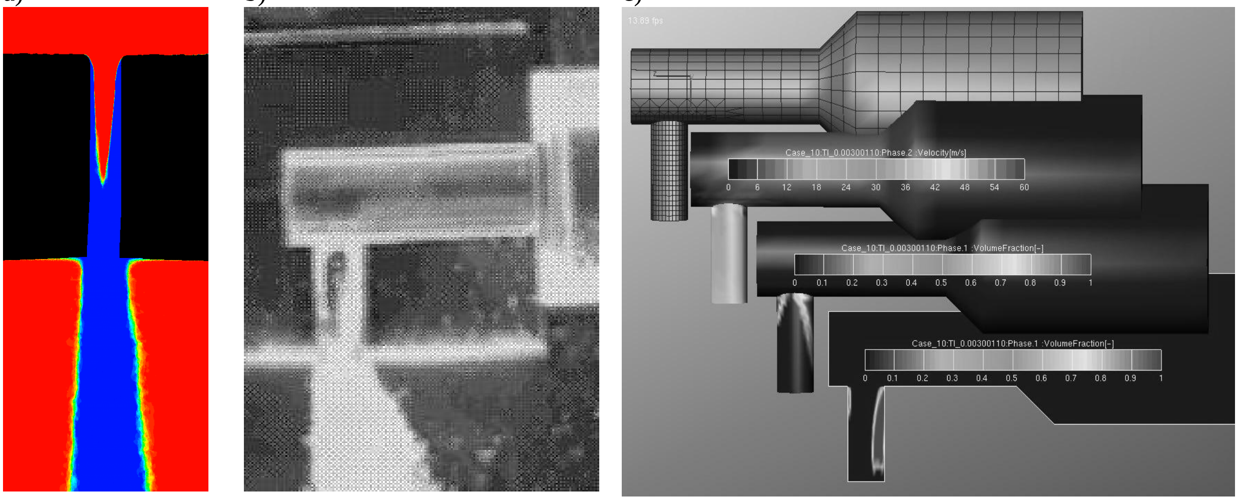

Figure 7. Comparison between experimental findings and calculation with FIRE8 two-fluid model
a) Result from AVL 2D-nozzle operated at 30 MPa pressure difference (slit width 100 microns)
b) Vapour distribution and spray shape in Chalmers nozzle (Re=78000, pressure difference: 0.2 Mpa)
c) Liquid phase velocity and vapour phase volume fraction (surface and cross sectional view) from FIRE 8

Summary

A new primary break-up model has been set up and implemented into the FIRE code. This model introduces detailed properties of cavitating nozzle flow into the break-up process. Droplets respectively droplet parcels are released from a liquid core of the fuel jet. The initial size and velocity distribution as well as the initial break-up angle is calculated from turbulent kinetic energy and turbulent energy dissipation in the nozzle orifice combined with an aerodynamic break-up mechanism. Modifications of turbulence along the core are also taken into account with a simple 1d k-ε model. As a characteristic feature the model predicts asymmetric spray shape and inhomogeneous distribution of droplet properties within the spray cone. Thus the model can explain spray behaviour near the nozzle in more detail than previous models and will support better understanding and optimisation of mixture formation. Validation of the model is presently done using experimental data from AVL and Chalmers University showing both, cavitation and spray formation simultaneously.

Nomenclature

C	Coefficient	S	TKE source term	σ	Surface tension
K	Turb. Kin. Energy	**T**	Reynolds stresses	τ_k	shear stress
M	Bubble mass	**v**	Velocity	<u>Subscript</u>	
$\dot{M}$	Breakup rate	α	volume fraction	1	Vapor
M	Momentum source term	ε	Turb. energy diss.	2	Liquid
N´´´	Number density	Γ	mass source term	A	Atomization
P	Pressure	ν	Viscosity	T	Turbulence
R	Droplet, bubble or core radius	ρ	Density	W	Aerodynamic diffusion

References

[1] K.Y. Huh and A.D. Gosman: Phenomenological Model of Diesel spray atomisation. Proceedings of the International Conference on Multiphase Flows, Sept. 24-27, Tsukuba, Japan, 1991
[2] G. Bianchi, P. Pelloni: Modelling the Diesel Fuel Spray Break-up by Using a Hybrid Model. SAE 1999-01-0226
[3] R. Tatschl, C. v. Künsberg Sarre, A. Alajbegovic and E. Winklhofer.: Diesel Spray Break-Up Modelling Including Multidimensional Cavitation Nozzle Flow Effects. 16th ILASS-Europe Conf., 11-13 Sept.2000, Darmstadt
[4] H. Grogger, A. Alajbegovic: Calculation of the Cavitating Flow in Venturi Geometries using Two-Fluid Model. 1998 ASME Fluids Engineering Division Summer Meeting. June 21-25, 1998, Washington, D.C., USA
[5] A. Alajbegovic, G. Meister, D. Greif, B. Basara: Three phase cavitating flows in high-pressure swirl injectors. Experimental Thermal and Fluid Science, 2002, article in press
[6] E. Winklhofer and A. Morozov, AVL List GmbH, Graz, private communications
[7] L.C. Ganippa et al.: The Structure of Cavitation and its Effect on the Spray Pattern in a Single-Hole Diesel Nozzle. SAE-Paper 2001- 01-2008