International Multidimensional Engine Modeling User's Group Meeting at the SAE Congress 2002

IC-ENGINE SPRAY MODELING – STATUS AND OUTLOOK

Reinhard TATSCHL, Christopher v. KÜNSBERG SARRE and Eberhard v. BERG

AVL List GmbH
Hans-List-Platz 1
A-8020 Graz
AUSTRIA

ABSTRACT

In the last decade 3D-CFD has been successfully established for the simulation of IC-engine fuel spray formation and propagation processes. The accuracy of the calculation results, however, strongly depends on the consistent modeling of the injector flow characteristics and the models adopted for simulation of the primary and secondary atomization processes. Hence, careful validation of the individual models and the availability of proper boundary conditions at the injector exit serve as major prerequisites for the successful analysis and optimization of high pressure sprays in diesel as well as in gasoline engines.

The present article provides an overview of the current status in modeling the governing mechanisms of spray formation under direct injection diesel engine conditions. Major emphasis is given to the modeling of primary and secondary break-up and the generation of proper boundary conditions at the injector exit.

1. INTRODUCTION

In the last decade 3D-CFD has been successfully established for three dimensional simulation of fluid flow, mixture formation, combustion and pollutant formation in internal combustion engines. In direct injected engines the accuracy of the simulation results and hence their contribution to design analysis and optimization strongly depends on the predictive capabilities of the models adopted for simulation of the injector flow, spray formation and propagation characteristics.

The present article provides an overview of the proper boundary conditions and models required for successful simulation of the spray formation/propagation characteristics in direct injected diesel engines. Individual model results are validated against selected experimental data. Finally, an outlook on future developments in IC-engine spray modeling is given. For all cases presented in this study the CFD code FIRE is used for simulation of the relevant injector flow and spray formation and propagation processes [1-7].

2. SPRAY MODEL

Currently the most common spray description is based on the Lagrangian discrete droplet method [8]. While the continuous gaseous phase is described by the standard Eulerian conservation equations, the transport of the dispersed phase is calculated by tracking the trajectories of a certain number of representative parcels (particles). A parcel consists of a number of droplets and it is assumed that all the droplets within one parcel have the same physical properties and behave equally when they move, break up, hit a wall or evaporate. The coupling between the liquid and the gaseous phases is achieved by source term exchange for mass, momentum, energy and turbulence. Various sub-models account for the effects of turbulent dispersion [9], coalescence [10], evaporation [11], wall interaction [12] and droplet break up [13].

3. BREAK-UP MODELING

The atomization of IC-engine fuel sprays can be divided into two main processes, primary and secondary break-up. The former takes place in the region close to the nozzle at high Weber numbers. It is not only determined by the interaction between the liquid and gaseous phases but also by internal nozzle phenomena like turbulence and cavitation. Atomization that occurs further downstream in the spray due to aerodynamic interaction processes and which is largely independent of the nozzle type is called secondary break-up.

The classic break-up models like TAB (Taylor Analogy Break-up), RD (Reitz and Diwakar) and WAVE do not distinguish between the two processes [14]. The parameters of these models are usually tuned to match experimental data further downstream in the region of the secondary break-up. Originally, these parameters are supposed to depend only on nozzle geometry, in reality they also account for numerical effects.

Other models like ETAB (Enhanced TAB), FIPA (Fractionnement Induit Par Acceleration) or KH-RT (Kelvin Helmholtz - Rayleigh Taylor) treat the primary break-up region separately [14]. Hence, they in principle offer the possibility to simulate both break-up processes independently. The correct values for the additional set of parameters, however, are not easy to determine due to the lack of experimental data for the primary break-up region.

Despite the sometimes tedious tuning of these model parameters the use of break-up models is generally advantageous compared to the initialisation of measured droplet distributions at the nozzle orifice. In the first approach the droplets are simply initialised with a diameter equal to the nozzle orifice (blob injection), the droplet spectrum automatically evolves from the subsequent break-up processes. The latter approach gives satisfying

results only as long as injection pressure and droplet Weber numbers are low.

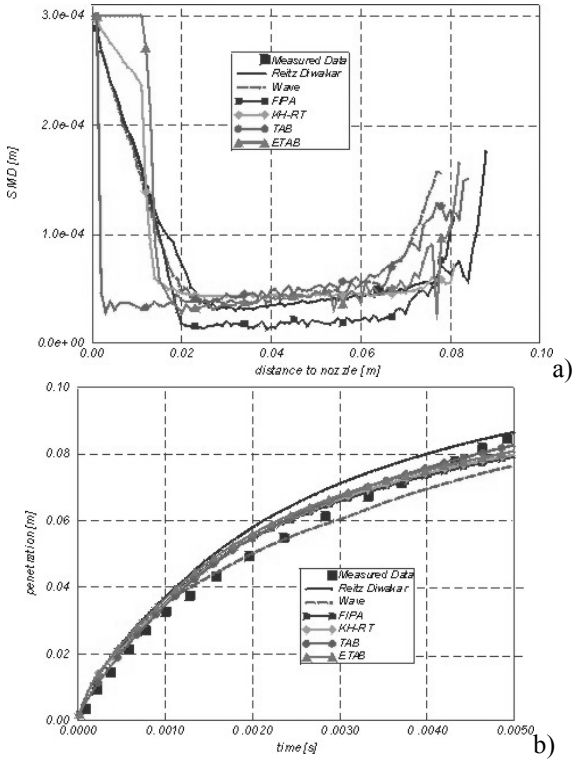


Figure 1 Comparison of different break-up models; a) Sauter Mean Diameter, b) spray penetration

The introduction of break-up models has considerably simplified the simulation of spray processes. In the past a number of different approaches has been presented and it is not easy to decide which one to choose for a specific simulation task. It turns out that practically all the break-up models are capable of reproducing measured data, as long as model constants are properly chosen (Figure 1, [14]).

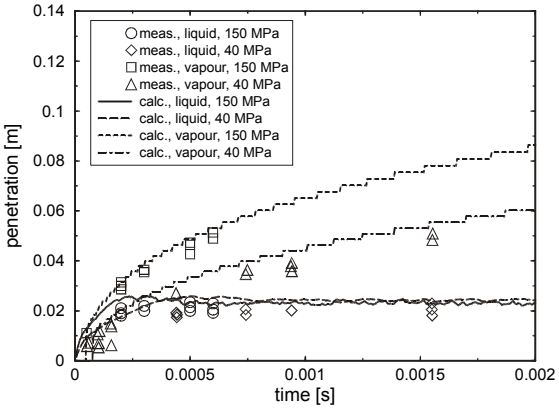


Figure 2 Calculated vs. measured liquid and vapor fuel penetration; injection pressure variation; chamber pressure: 60 bar, chamber temperature: 800K

For evaporating sprays Figure 2 shows the results of spray bomb simulations for a chamber density of 25 kg/m³ and a temperature of 800 K together with the experimental data taken from [15]. The WAVE model is used here to account for the spray / droplet break-up processes. It can be seen that the liquid penetration is nearly independent of the injection pressure level. The propagation of the fuel vapor phase, however, directly scales with the injection pressure level – higher injection pressure leads to a significant increase in the vapor penetration. For the injection pressure variation study all parameters of the spray and injector flow models have been kept constant. It can be seen that the agreement between experiment and simulation is very good for both the vapor and the liquid penetration characteristics.

4. INJECTOR FLOW COUPLING

The knowledge of the flow characteristics of the injected fuel at the nozzle exit is a key issue for a successful simulation of the spray primary break-up and hence the spray propagation characteristics in diesel engines. Recent modeling effort has lead to the successful coupling of the local flow conditions at the injector exit with advanced primary break-up models that account for injector flow induced turbulence as well as cavitation effects on the primary spray break-up processes [16]. The subsequent aerodynamic break-up processes are again modeled with the well established secondary break-up models.

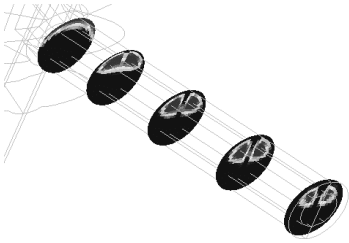


Figure 3 Liquid (dark) / vapour (bright) volume fraction distribution; VCO 2 x 0.14 mm, injection pressure: 1300 bar, chamber back-pressure: 53 bar

The mathematical model used for injector flow calculations is based on a multi-fluid formulation of the relevant conservation laws [17]. Figure 3 shows a representative result of a two-phase flow simulation in a diesel injector.

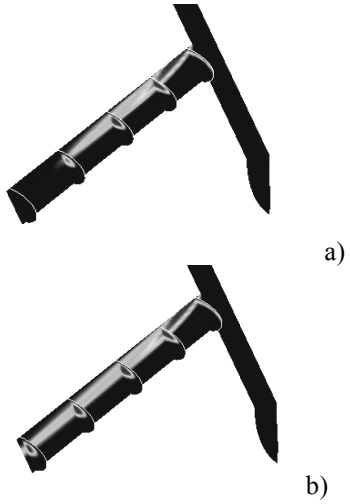


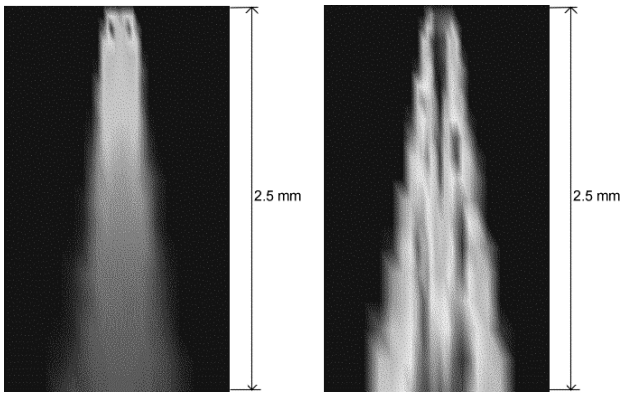
Figure 4 Liquid (dark) / vapour (bright) volume fraction distribution; VCO 2 x 0.14 mm, chamber back-pressure: 53 bar; a) injection pressure: 300 bar, b) injection pressure: 1300 bar

The fuel vapor distribution in the nozzle hole clearly reveals the three-dimensional nature of the injector flow, with the secondary flow motion strongly determining the location and shape of the cavitation induced fuel vapor distribution that finally leads to the formation of two distinct cavitation zones at the nozzle hole exit.

The extension of the cavitation induced fuel vapor containing region is strongly depending on the injection pressure and the chamber back-pressure conditions as well as geometrical details of the injector. Figure 4 demonstrates the increased cavitation tendency with elevated injection pressure.

Primary/secondary break-up modeling that accounts for the competing effects of turbulence, cavitation and aerodynamic induced break-up processes is based upon the spatially and temporally resolved injector flow data at the nozzle exit. In [16] the turbulence induced break-up is accounted for by solving a transport equation for the turbulent kinetic energy and its dissipation rate within the liquid fuel core.

The impact of the collapsing cavitation bubbles on the primary break-up is modeled via additional source terms in the turbulence model. The turbulence and cavitation induced break-up competes with the aerodynamic one until at a certain distance downstream of the nozzle exit the aerodynamic break-up processes become dominant.



a) b)

Figure 5: Primary break-up; nozzle 1 x 0.17; injection pressure 800 bar; a) turbulent / cavitation induced break-up rate, b) aerodynamic break-up rate [16]

The relative importance of turbulence / cavitation induced and aerodynamic induced break-up is demonstrated in Figure 5 which shows the local distributions of the turbulent and aerodynamic break-up rates in a section across the spray axis. It is evident that the maximum turbulence / cavitation induced break-up intensity is observed very close to the nozzle exit and can be attributed to the nozzle flow induced and (to a minor extent) to the cavitation collapse induced turbulent velocity fluctuations. Due to dissipation of the turbulent fluctuations, however, the turbulence induced break-up rate is significantly reduced with increasing distance from the nozzle exit until it becomes negligible at about 2.5 mm downstream of the nozzle tip. The aerodynamic break-up rates show the opposite behaviour, i.e. they are very low immediately at the nozzle exit but increase significantly with increasing distance from the nozzle, where the compact liquid core has already been significantly disintegrated due to primary break-up mechanisms. Finally, even at the spray axis high aerodynamic break-up rates can be identified, indicating complete fragmentation of the compact spray core.

Elevated injection pressure levels lead to higher injection velocities and hence increased turbulence levels which directly lead to higher turbulence induced break-up rates. Increased chamber back-pressure levels, however, affect mainly the aerodynamic break-up mechanisms via the impact of higher gas densities and hence elevated interaction forces between ligaments / droplets and gaseous phase [16].

5. EULERIAN DENSE SPRAY MODELING

As shown in the previous chapters fuel sprays in today's IC-engine applications are usually modeled adopting the Lagrangian treatment of representative parcels of droplets tracked in the surrounding gas flow field. This method is especially suitable for dilute sprays, but has shortcomings with respect to modeling of dense sprays where particle interactions are strongly influenced by collisions and parcels have to be rearranged and redistributed very often. Further problems are reported connected with bad statistical convergence [18] and also with dependence of the propagation of the spray on grid size [19].

An alternative approach is based upon adopting an Eulerian/Eulerian method treating different size classes of the spray droplets as separate, interpenetrating phases and solving conservation equations for each of them. The model under development is based on an Eulerian multiphase approach that has been derived from ensemble averaging of the conservation equations [20]. For each phase mass, momentum and energy conservation equations are solved as well as corresponding equations for turbulent kinetic energy and turbulent energy dissipation. Within each computational cell the droplet phases are characterized by a certain volume fraction. Thus all exchange processes related to droplet size or specific surface of the droplet phases depend on the flow regime and have to be modeled additionally. For the flow configuration considered here this concerns momentum transfer via drag and lift forces as well as mass transfer from secondary breakup, evaporation and collisions.

At the present state of implementation of the momentum transfer models the drag force takes into account effects of Reynolds number and volume fraction as well as deformation of the droplets. A first approach for turbulent dispersion force to treat interactions between gas phase turbulence and the droplet phases is also included. The models for lift forces cover Saffman and Magnus force. Further effects as, e.g., from virtual mass or Basset force have been neglected for the present application of droplet flow in a gaseous environment as is supported by the analysis of Sommerfeld [21].

Regarding mass transfer between the phases basic models for evaporation, secondary breakup and collisions have been implemented. Evaporation mass transfer is described according to differences of vapor pressure within the droplet phase and in the gas flow. For secondary breakup rate approaches from the standard WAVE and FIPA breakup models have been implemented. The collision

model takes into account coalescence as well as secondary breakup after collision according to a collision Weber number criterion. In general the exchange terms are formulated in a modular way to allow an easy coupling of additional models for the different interphase exchange processes. Matching of turbulence model and adaptation of turbulent dispersion force for the spray application is performed presently.

The model has already been applied to Diesel injection test cases using simplified but typical conditions. Effects of inlet conditions, various drag formulations and basic functionality of the secondary breakup, evaporation and collision models have been tested successfully [22].

6. REFERENCES

[1] Tatschl, R., Wieser, K., Reitbauer, R., Multidimensional Simulation of Flow Evolution, Mixture Preparation and Combustion in a 4-Valve Gasoline Engine, COMODIA 94, Yokohama, 1994

[2] Tatschl, R., Fuchs, H., Brandstätter, W. Experimentally Validated Multi-dimensional Simulation of Mixture Formation and Combustion in Gasoline Engines, IMechE C499/050/96, 1996

[3] Tatschl, R., Riediger, H., PDF Modelling of Stratified Charge SI Engine Combustion, SAE 981464, 1998

[4] Tatschl, R., Pachler, K., Winklhofer, E., A Comprehensive DI Diesel Combustion Model for Multidimensional Engine Simulation, COMODIA 98, Kyoto, 1998

[5] Tatschl, R., Riediger, H., Bogensperger, M., Multidimensional Simulation of Spray Combustion and Pollutant Formation in a Medium Speed Marine Diesel Engine, 1998 FISITA World Automotive Congress, Paris, 1998

[6] Tatschl, R., Wiesler, B., Alajbegovic, A., Kuensberg Sarre, Ch., Advanced 3D Fluid Dynamic Simulation for Diesel Engines, Thermofluidynamic Processes in Diesel Engines, Valencia, 2000

[7] Tatschl, R., Riediger, H., v. Künsberg Sarre, Ch., Putz, N., Fast Grid Generation and Advanced Physical Modelling – Keys to Successful Application of CFD to Gasoline DI Engine Analysis, ImechE C587/042/2000, 2000

[8] Dukowicz, J.K., A Particle-Fluid Numerical Model for Liquid Sprays, Journal of Computational Physics, Vol. 35, pp. 229-253, 1980

[9] Gosman A.D. and Ioannides, E., Aspects of Computer Simulation of Liquid-Fueled Combusters, J. Energy, 7, pp. 482-490, 1983

[10] O'Rourke, P.J., Modeling of Drop Interaction in Thick Sprays and a Comparison with Experiments, IMechE - Stratified Charge Automotive Engines Conference, 1980

[11] Dukowicz, J.K., Quasi-steady Droplet Phase Change in the Presence of Convection, Los Alamos Report LA-7997-MS, 1979

[12] Naber, J.D., Reitz, R.D., Modeling Engine Spray / Wall Impingement, SAE 880107, 1988

[13] Liu, A.B. and Reitz, R.D., Modeling the Effects of Drop Drag and Breakup on Fuel Sprays, SAE 930072

[14] v. Künsberg Sarre, Ch., Tatschl, R., Spray Modelling / Atomisation – Current Status of Break-Up Models, ImechE Seminar on Turbulent Combustion of Gaseous and Liquids, Lincoln, 1998

[15] Habchi, C., Baritaud, T. et al., Modeling Atomization and Breakup in High-Pressure Diesel Sprays, SAE 970881

[16] Tatschl, R., Kuensberg Sarre, Ch., Alajbegovic, A, Winklhofer, E., Diesel Spray Break-Up Modeling Including Multidimensional Cavitating Nozzle Flow Effects, ILASS Europe 2000, Darmstadt

[17] Drew, D.A., Passman, S.L., Theory of Multicomponent Fluids, Springer, New York, 1998

[18] Krüger, Ch., Validierung eines 1D-Spraymodells zur Simulation der Gemischbildung in direkteinspritzenden Dieselmotoren, Dissertation RWTH Aachen, März, 2001

[19] Abraham, J., What is Adequate Resolution in the Numerical Computation of Transient Jets?, SAE 970051

[20] Alajbegovic, A., Drew, D.A., Lahey, R.T., An Analysis of Phase Distribution and Turbulence in Dispersed Particle/Liquid Flows, Chem. Eng. Comm., Vol. 174, pp. 85-133, 1999

[21] Sommerfeld, M., Modellierung und numerische Berechnung von partikelbeladenen turbulenten Strömungen mit Hilfe des Euler/Lagrange Verfahrens, Shaker Verlag, Aachen, 1996

[22] v. Berg, E., Alajbegovic, A., Tatschl, R., Krüger, Ch., Michels, U., Multiphase Modeling of Diesel Sprays with the Eulerian/Eulerian Approach, 17th ILASS Europe Conference, Zürich, 2001