Comparison of Spray Simulation and Measurement for a Multi-hole Injector with and without Wall Impingement

Dominik Kuhnke

Corporate Research and Development, Dept. of Applied Physics, Robert Bosch GmbH
Postfach 106050, 70049 Stuttgart, Germany
Email: Dominik.Kuhnke@de.bosch.com

Abstract

Spray-wall impingement is an important effect especially in engines with direct injection. However, the simulation of spray-wall impingement using existing models still shows significant deviations with respect to real sprays. Improved modeling is therefore required for a correct simulation of mixture formation and combustion. Prior to simulating spray-wall impingement, it is necessary to provide a spray with well defined properties. This work shows in a first step the simulation of a free stream spray and its validation by comparison to measurements. In a second step, we simulate spray-wall impingement using existing models. We evaluate and compare our results to Phase Doppler Anemometry measurements. As a main result we find that average droplet velocities before impingement are simulated correctly, while the modelling of post impingement velocities needs further improvement.

1 Introduction

Spray-wall impingement that occurs during the injection of fuel in direct injection engines has a major influence on a variety of parameters that are currently under consideration in the development of engines. These are especially mixture formation and energy efficiency as well as soot production. Commercial CFD-Codes (Computational Fluid Dynamics) that are applied for the engine design process usually use models from literature as [1][2], but even new codes still apply simple old models [3]. As these show little agreement to measurements especially at elevated temperatures, the intention of this study is to compare various spray-wall interaction models to actual measurements under comparable conditions. One crucial point of this comparison is the correct representation of the incoming spray with regard to the velocity distribution, which constitutes the first part of the present study. The second part deals with the comparison of several spray-wall interaction models with regard to the velocity of the secondary droplets.

2 Free Stream Spray

2.1 Boundary conditions and set-up

At the Chalmers University of Technology various free stream spray Phase Doppler Anemometry (PDA) measurements with a BOSCH 6-hole injector were carried out [4]. The injection pressure was 100bar, the ambient pressure 5bar at a temperature of 500K.

Droplet sizes and velocity components were detected in one of the six spray plumes at several radial positions in planes 10, 20 and 30mm downstream the nozzle with plane normal vectors oriented parallel to the injector axis, as shown in figure 1a. This is the reference case for our simulations.

The goal of the free stream simulation is to yield similar spray properties at 30mm below the nozzle in order to have the correct starting condition for the spray-wall impingement simulation. Thus, in the simulation, droplet properties are recorded in the same locations as in the experiment. According to the experimental setup with only one measured plume, most simulation domains are designed to contain only one of the spray plumes, as shown in figure 1b. A discussion of the effect of this simplifiction follows later.

2.2 Simulation

The code used for the simulation is the FIRE 7.3 CFD code from AVL Corp. [5]. Therein the motion of droplets is determined by the Lagrangian particle tracking approach, which solves the equation of motion for parcels of droplets with identical properties (*Discrete Droplet Method* (DDM) of Dukowicz [6]). Turbulence dispersion is defined by the Eddy-Lifetime model [7] that determines the fluctuating velocity normally distributed from the turbulence kinetic energy. Full interaction between drops and gas phase is considered for momentum, mass and heat but not for turbulence kinetic energy and dissipation, i.e. drops only receive but do not induce turbulence or dissipation to the gas phase. Two-way coupling of turbulence and dissipation in dense sprays is not available. As discussed later, this fact is probably the reason, why the turbulence level of the drops is lower in the simulation.

The initial droplet velocity is defined by the hole diameter and the mass flux through the nozzle hole; initial drop size distribution is taken from the measurements, drop sizes above 40 micron were not observed. Except for coarse grids, droplet size distributions are essentially conserved during spray propagation; this is also observed in the measurements and thus no further investigation about droplet sizes is performed here.

Grid effects

Simulating sprays brings up the problem of creating a suitable grid which is on the one hand capable to resolve the gas phase as fine as necessary, but on the other hand coarse

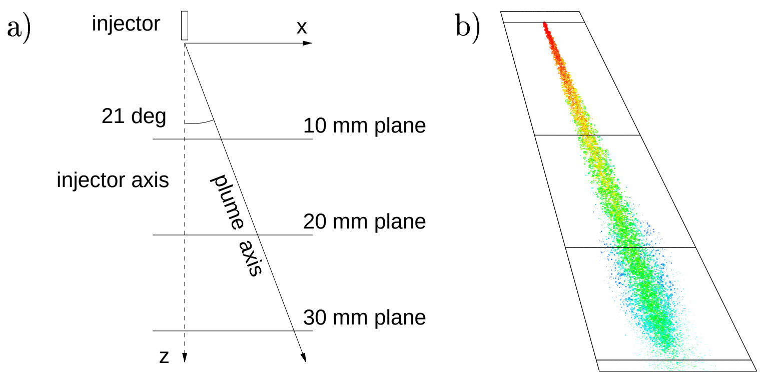
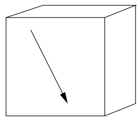
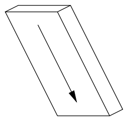
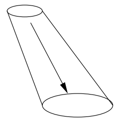
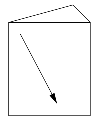


Figure 1: a) Free stream setup: Droplet detection planes in simulation and measurement located orthogonal to injector axis. b) Spray orientation and detection planes within grid

Table 1: Simulation Grids: arrow shows the spray plume direction

Grid	A	B	C	D
Topology	hexahedron	hexahedron	o-grid	cake
Shape				
Cell size* [mm^3]				
#1:	$1.6 \times 0.6 \times 0.3$	$0.13 \times 0.15 \times 0.11$	$0.083 \times 0.054 \times 0.23$	$0.1 \times 0.072 \times 0.35$
#2:	$0.17 \times 0.1 \times 0.23$		$0.065 \times 0.054 \times 0.114$	

* Minimum cell dimensions specified with respect to x, y, z coordinates according to figure 1.

enough to regard the droplet parcels as small compared to the cell volume. According to Abraham [8], the steep gradients in the gas phase velocity can only be resolved correctly, if the cell size is smaller than the orifice diameter. This corresponds to the usual assumption, that the latter yields the typical length scale of the turbulence, which must be resolved by the grid. In contrast to this, the Lagrangian particle tracking assumes a drop spacing much greater than the drop diameter and a large void fraction (volume fraction of gas in the computational cell). Otherwise numerical instabilities can occur.

First simulations of all six spray plumes in a simple grid reveal very low gas velocities between two adjacent spray plumes of the multi hole injector. Due to the little interaction of plumes caused thereby, most of the simulations are performed in a grid that only contains one of the six plumes. The big advantage of this is a much better resolution with a reasonable number of grid cells. To evaluate the effect of this simplification we can compare the *penetration* of the spray, which means the distance of the spray plume tip from the nozzle hole, where the tip location is defined by a 99% mass criterion. We see essentially a slight deviation of the spray towards the injector axis due to a decreasing pressure (as observed even stronger in the measurements [4]); moreover we see a reduced velocity of the smaller drops (deviation below 3m/s or 12%), while penetration is hardly affected. This result is obtained from a simulation in a 60 degree sector geometry (D, table 1), computed one time with symmetry boundary and one time with open boundary condition.

Simulations on a coarse rectangular cube grid (A1) show a lower penetration when compared to finer grids (A2 or B), while the grid face orientation (spray-parallel in B) has no effect on the penetration, see figure 2a. Figure 2b shows, that further refinement

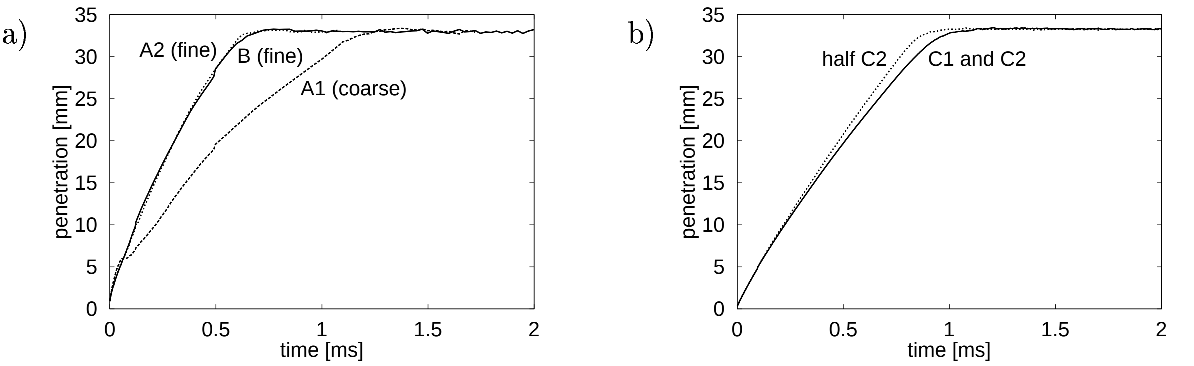


Figure 2: a) Coarse grid A1 results in slower penetration than fine grids A2 and B. b) Simulation of a half spray plume yields less penetration; grid refinement from C1 to C2 has no effect.

from grid C1 to C2 has no effect, and that the application of grids containing only half of the spray (assuming symmetry) is not advisable, as it yields different penetration and no convergence with grid refinement. Time behaviour in figures 2a and 2b is not comparable, as injection velocities are different. The maximum value of the penetration is limited by the size of the grid.

A better resolution with less cells is achieved by an o-grid as illustrated in table 1, case C, and figure 3. In contrast to Abraham [8] no instabilities occur with the applied fine resolutions, thus the grid C1 is preferred and used in the following.

Drag Effects

When compared to measurements, droplet velocities in the spray plume centre at 30 mm below the injector are too low when using the standard drag model of Schiller and Naumann [9], see figure 4, whereas the trend of the correlation between drop diameter and drop axial velocity is the same as the measured one. Assuming that the drag law of Schiller and Naumann yields a correct penetration at atmospheric pressure, we introduce a linear interpolation which reduces the drag coefficient linearly with increasing pressure to 50% at 5bar. Another simulation with this reduced drag now yields correct average velocities at the 30 mm, see figure 4, and an appropriate time-resolved behaviour, see

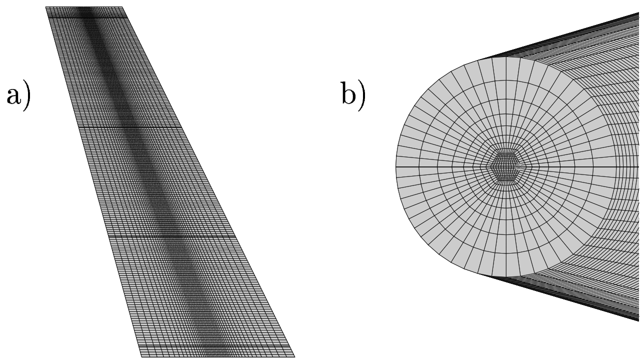


Figure 3: Grid C1 with refinement in core region yields good results with reasonable number of cells. a) axial cut b) top view

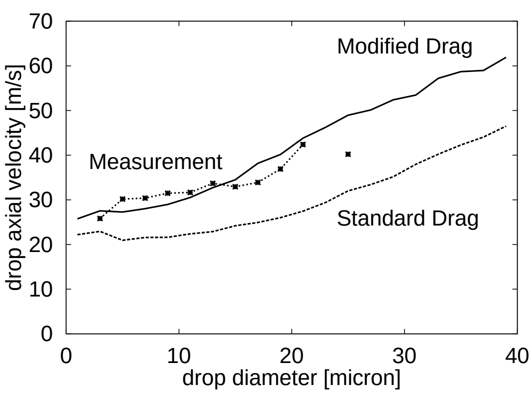


Figure 4: Application of pressure-dependent drag correction yields better agreement of size-velocity correlation with measurements at 30 mm from nozzle; simulation in grid C1, data detected at single point in spray plume centre; experimental data connected with lines only for diameters where a sufficient number of drops for averaging is detected.

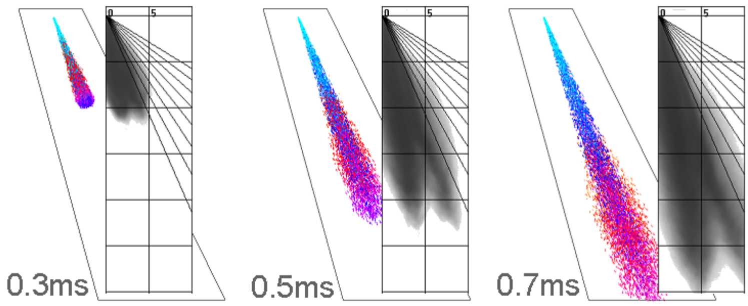


Figure 5: Visual comparison of time dependent penetration of simulation in grid C1 with pressure-dependent drag correction (resp. left) and measurement (resp. right). Backlight visualisations show half of a spray (3 plumes).

figure 5. Whether this correction is necessary to compensate for numerical or for physical effects, seems of minor importance for spray-wall interaction studies. The intention here is just to obtain correct pre-impingement velocities for the following simulation of spray-wall interaction. However, validation of this approach by simulation at different ambient pressures and comparison to measurements is clearly necessary.

Turbulence Effects

As figure 4 shows, the mean droplet velocities in simulation and measurement agree quite well while the dispersion of the velocity is much lower in the simulation. This is shown by the standard deviation of droplet velocities in figure 6a and by the velocity histogram figure 6b.

This discrepancy clearly shows, that the interaction between droplets and gas needs better modelling. The droplet dispersion model used in FIRE can either be the Eddy-Lifetime model or the model of O'Rourke, depending on the time step size. As these are small enough, the model of O'Rourke is never applied, since droplets do not cross more than one eddy per time step. In order to increase the turbulence of the drops, we test the influence of increased turbulent interaction time resulting in insignificant change in the droplet turbulence. Even an *artificial* increment of the turbulent velocities seen by the droplet mainly results in a general increment of the gas phase turbulence, but not

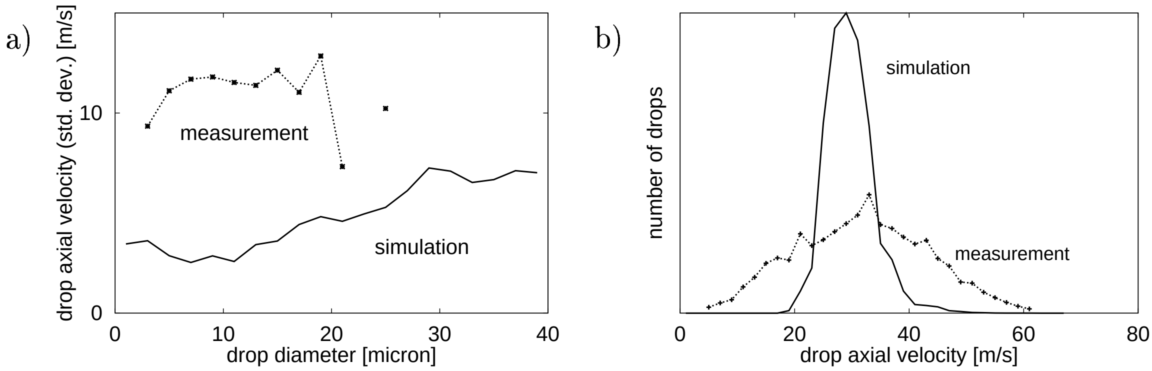


Figure 6: Simulation with modified drag in grid C1: a) Standard deviation of the axial velocity at 30 mm distance from the nozzle in dependence of the droplet size. Variation in measurement curve for drop sizes above 18 micron and omitted connection line due to low number of drops detected; b) Velocity distribution shows large deviation of simulation from experiment.

in a significant change of the droplet velocity dispersion. Thus the main reason for this effect is presumably the missing coupling of drop turbulence to gas turbulence.

3 Wall impingement

3.1 Boundary conditions and set-up

The set-up is quite similar to the free stream case, only the wall and the measurement planes are tilted by 9° to obtain an impingement angle of 30° to the wall normal, see figure 7a. The experiments from Chalmers University of Technology [4] were performed at 381K, 435K and 489K wall temperature, 100bar injection pressure, 5bar chamber pressure and 500K gas temperature. Measurement planes are located at 0.5, 1, 3 and 5 mm above the surface. Simulations apply several wall interaction models under the same conditions.

3.2 Comparison of wall interaction models to the experiments

FIRE provides five non-trivial wall interaction models: The three *walljet* models are based on the model of Naber & Reitz [3] but differ in the ejection angle and droplet size. *Splash1* is a model motivated by the law described by Bai & Gosman [1], but includes several modifications. The *wall film splash* model accounts additionally for the surface roughness, it is essentially the model of Mundo, Sommerfeld and Tropea [2]. A more detailed description can be found in the FIRE manual [5].

Figures 7b and 7c show exemplary two impingement models with extremely different spreading: b) is representative for the *wall film splash* and the *walljet* models, as these models all produce secondary droplets only close to the wall; c) only the *splash1* model shows droplets in larger distance from the wall. The illustration here is scaled by drop size; illustrations scaled by drop mass reveal that most of the mass remains close to the surface. The seeming contradiction between size and mass results from the parcel clustering of drops.

The analysis of droplet post-impingement velocity vectors at the same PDA-points in experiment and simulation allows a more detailed comparison, see figure 8. It turns

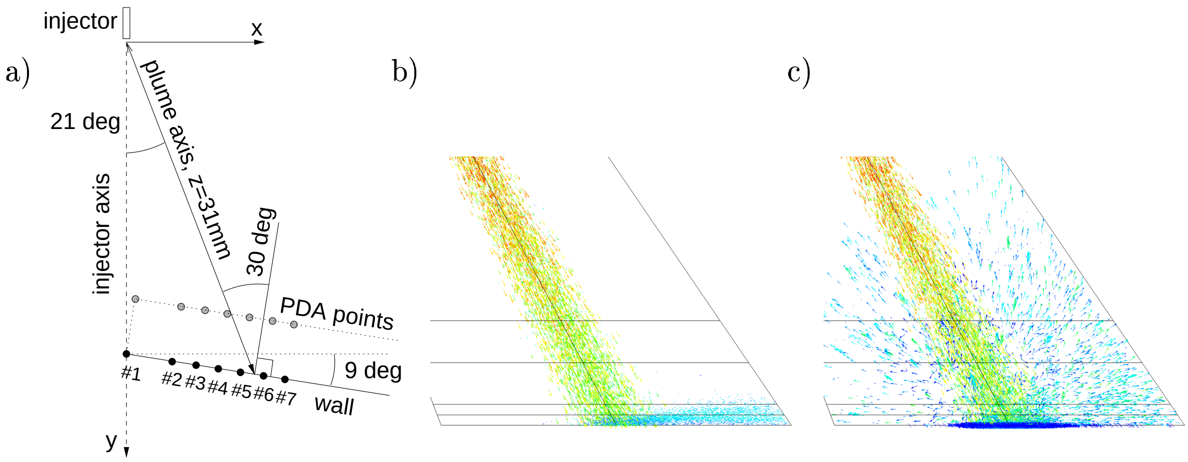


Figure 7: a) Sketch of experimental set-up with normal impingement angle 30°, showing the location of PDA measurement points. Results of calculations with FIRE spray-wall interaction models b) *wall film splash* and c) *splash1*, showing primary (pre-) and secondary (post-interaction) drop velocity vectors; vector size scaled by drop size.

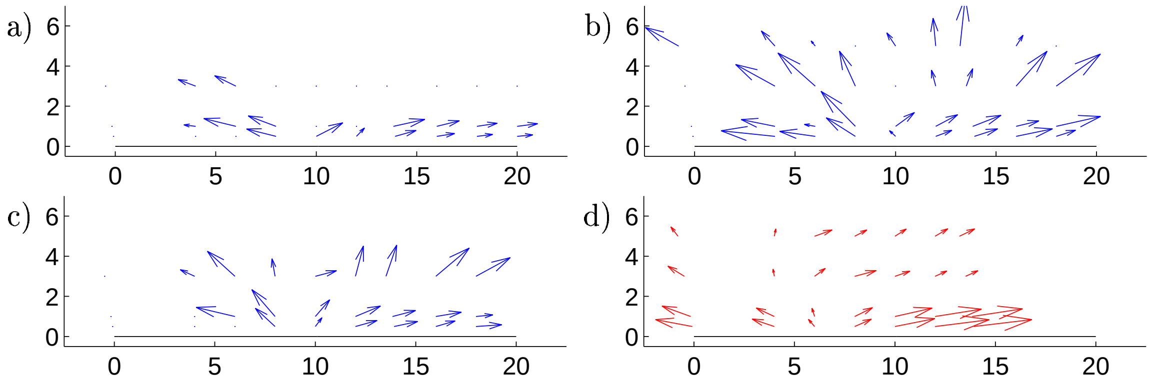


Figure 8: Spatially resolved secondary droplet velocity vectors; axes units are space coordinates in mm: abscissa denotes distance from injector axis along surface, ordinate the height from surface; a) *walljet2* b) *splash1* c) model developed in present work d) experiment: The impingement point is shifted towards injector axis (located at x=0mm) as discussed in subsection 2.2, *Grid effects*.

out, that the droplet velocity vectors in the *walljet2* simulation, see figure 8a, agree at least at some locations qualitatively well with the experiment, but there are also locations, especially in more distant planes (e.g. at 5mm from the surface), where the simulation does not show a single droplet. This effect is even more pronounced for the other *walljet* models or the *wall film splash* model, which we do not display therefore. In contrast to this the Bai & Gosman-motivated model *splash1* shows splashed droplets of comparatively high velocity and size even far from the surface, see figure 8b.

In figure 8c we see the simulation results based on a new impingement model that we compose from two literature models: it uses the regimes as originally defined by Bai and Gosman [1], the wall angle according to their sketch and an azimuthal angle as suggested by Naber and Reitz (based on the potential flow law). When comparing the vector-plots of the local velocities, our new model captures the trend of the measurement better than the standard models of FIRE, showing a sideward deviation of droplets that propagate against the incoming flow.

None of the FIRE wall interaction models describes the behaviour correctly, and even our improved model described above still shows some deviation, so there is still the need for improved modelling. It is not surprising that simple models like the walljet models derived for diesel spray application do not yield close agreement with the gasoline spray impingement data. One of the principal reasons for the failure of the walljet models is, that these have a predefined restriction on the ejection angle with respect to the surface and do not allow droplets to be ejected at large wall angles. All models have in common, that they do not take into account the effect of wall temperature, ambient pressure and droplet interaction during the impact. Only the *wall film* model considers surface roughness in a very basic approach and *walljet0* does not even respect for droplet disintegration.

Furthermore the analysis of the experimental post-impingement velocities shows, that the droplet velocity depends on the wall temperature. The highest velocities were not observed for the high temperature case 489K but for the 435K wall, which is close to Leidenfrost temperature. This shows that the consideration of temperature, that has been neglected in the former impingement models, is necessary.

4 Conclusion

This work shows that by the application of our drag correction the simulation of free stream sprays yields little deviation in droplet size and average velocity as compared to experiments. However, to obtain a correct velocity dispersion improved turbulence modelling is still necessary. Moreover, with the provided spray, our spray-wall impingement model yields secondary droplet velocities that compare qualitatively better to measurements than the existing impingement models implemented in FIRE. However, there is still some deviation, which indicates the need to include the experimentally observed influence of the temperature in the wall interaction model.

5 Acknowledgement

I am indebted to Ronny Lindgren at the Chalmers University of Technology for communicating his yet unpublished PDA measurement results and to Thomas Stach at Robert Bosch GmbH dept. GS-IN/ENG1 for backlight photographs. Financial support is provided by the European Commission within the scope of *Framework Program 5*, energy project DWDIE, Contract No. ENK6-CT2000-00051.

References

[1] Bai C Gosman A D 1995 *Development of Methodology for Spray Impingement Simulation* SAE paper No. 950283

[2] Mundo Chr Sommerfeld M Tropea C 1995 *Droplet-Wall Collisions: Experimental Studies of the Deformation and Breakup Process* Int. J. Multiphase Flow **21**,151-173

[3] Naber J D and Reitz R D 1988 *Modeling Engine Spray/Wall Impingement* SAE paper No. 880107

[4] Lindgren R 2002 University of Chalmers, Gothenburg, Sweden, unpublished results

[5] FIRE 7.3 Software and Manual 2001, AVL LIST GmbH, Hans-List-Platz 1, A-8020 Graz, Austria

[6] Dukowicz J K 1980 *A Particle-Fluid Numerical Model for Liquid Sprays* J. Comp. Physics **35** 229-253

[7] Gosman A D Ionnaides E 1981 *Aspects of Computer Simulation of Liquid-Fueled Combustors* AIAA 81-323

[8] Abraham J 1997 *What is Adequate Resolution in the Numerical Computations of Transient Jets?* Society of Automotive Engineers (SAE) Paper No. 970051

[9] Schiller L Naumann A Z 1933 VDI **77** 318-320