Numerical Modelling and Experimental Characterization of a Pressure-Assisted Multi-Stream Injector for SCR Exhaust Gas After-Treatment

Achinta Varna and Konstantinos Boulouchos
Swiss Federal Institute of Tech.

Alexander Spiteri and Panayotis Dimopoulos Eggenschwiler
EMPA

Yuri M. Wright
ETH Zurich/Combustion+FlowSolutions GmbH

ABSTRACT

Simulations for a pressure-assisted multi-stream injector designed for urea-dosing in a selective catalytic reduction (SCR) exhaust gas system have been carried out and compared to measurements taken in an optically accessible high-fidelity flow test rig. The experimental data comprises four different combinations of mass flow rate and temperature for the gas stream with unchanged injection parameters for the spray. First, a parametric study is carried out to determine the importance of various spray sub-models, including atomization, spray-wall interaction, buoyancy as well as droplet coalescence. Optimal parameters are determined using experimental data for one reference operating condition. The model is subsequently applied to all operating conditions with unaltered parameters and validated by means of the measured droplet velocity fields, droplet diameter distributions and spray-tip propagations which have been characterized by means of Particle Image Velocimetry (PIV), Phase Doppler Anemometry (PDA) and shadow imaging. Finally, the model is used to investigate in detail phenomena characteristic of inclined sprays in cross-flows. Effects such as formation of vortices behind the spray core for low cross flow velocities, greater entrainment of droplets at high flow rates as well as wall film dynamics and the interaction of droplets also with the side walls are discussed in detail, highlighting the importance of the respective phenomena at the different operating conditions.

CITATION: Varna, A., Boulouchos, K., Spiteri, A., Dimopoulos Eggenschwiler, P. et al., "Numerical Modelling and Experimental Characterization of a Pressure-Assisted Multi-Stream Injector for SCR Exhaust Gas After-Treatment," *SAE Int. J. Engines* 7(4):2014, doi:10.4271/2014-01-2822.

INTRODUCTION

Sprays propagating in cross-flow conditions are encountered in many practical applications since cross-flow improves the exchange of mass, momentum and energy between liquid and gas phase. This is favorable in processes such as urea injection in SCR systems [1, 2, 3, 4, 5, 6, 7], fuel injection in turbojet/ ramjet/scramjet [8, 9, 10], spark-ignition combustion employing both Direct-Injection Stratified Charge as well as Port-Fuel injection systems [11, 12, 13], chemical sprays in agriculture sector [14, 15, 16, 17] and sprays in gas conditioning industry [18]. Consequently a wide variety of experimental studies have been conducted to understand liquid sprays in cross-flows.

An extensive experimental study specifically concerning the nature of spray under SCR operating conditions has recently been presented in [6], while experiments addressing the effect

of angle of injection are reported in [8]. Investigations have also been carried out with and without the consideration of spray-wall interaction. As shown in [9], at low weber number (We), bag-breakup of liquid jet is witnessed at low momentum ratio (q) and Rayleigh breakup occurs at high q . At intermediate We , multimode breakup is observed at low q and Rayleigh break-up happens at high q . Spray dynamics is hardly altered by q at high We . In [14] it was found that the spray region could be divided into different zones on the basis of free stream velocity, induced air speed and liquid jet velocity. Formation of vortices downstream of spray and the extent of entrainment of droplets depend on the ratio of free stream and induced air velocities [14].

The gas will entrain small droplets thus preventing them from interacting with the channel floor. Larger droplets are deflected from their original path and the impingement region is relocated downwind. On the whole, droplets reaching the floor have a

much higher average diameter and velocity than they would have in the absence of cross-flow [19]. This unique feature of sprays in cross-flow enables the control of droplet size distribution actively at different locations [20]. Wall-jet vortices are formed when spray impacts the floor, which remove small droplets from the floor and introduce them into the flow [19]. However, cross-flow reduces the influence of these roll-up vortices. Increasing injection pressure reduces the effect of cross-flow. Increasing the injection timing increases film thickness and vapor layer thickness [11]. 'Splash' becomes relatively important phenomenon in the presence of cross-flows [12] and it also improves interaction between 'crowns' at the floor and droplets before impact such that the volume flux of small droplets moving from the floor increases.

The presence of a wall was found to decrease the penetration rate of the spray due to which the available impact energy is smaller than would be expected with free spray [12]. Cross-flow also helps in secondary atomization by film-stripping [13, 21]. The gas momentum promotes dispersion, partial vaporization and disruption of droplets. Cross-flow decreases normal velocity of impinging droplets. It carries away droplets more prone to deposit which leads to formation of thinner films. It also transports secondary droplets and evades their re-impact with the floor [13]. Diameters of secondary atomized droplets increase with increasing film thickness which could be achieved by reduction of cross-flow velocity or increase of fuel flow rate [22]. Film thickness increases steadily along flow direction [21].

While a broad range of experimental studies are reported in the literature as summarized above, computational fluid dynamic (CFD) studies of sprays in cross-flows are scarce. A numerical study regarding the effect of injection angle on droplet diameters and velocities is available [18]. It has also been documented that spray-wall interaction, drag force and buoyancy models are critical for sprays in urea-SCR systems [4]. [1, 2, 3, 4, 5] further reported simulations of urea-SCR systems but none of them have made detailed validation of the urea spray characteristics using extensive experiment data.

This study constitutes an attempt towards improved understanding concerning the specifics of liquid spray under SCR conditions by combining CFD and experiments. The effects of primary atomization, spray-wall interaction, buoyancy, droplet coalescence and film sub-models on the predictions have been carefully assessed. All the necessary models are optimized for one SCR condition using the available PIV, PDA and shadow imaging data. The validated set-up is subsequently validated by means of additional operating conditions. The findings are in good agreement with the fundamental characteristics of sprays in cross-flows which have been previously reported in the literature.

EXPERIMENTAL SETUP

Experimental data is gathered on a high temperature SCR flow rig [6] at the laboratory for internal combustion engines at Empa. Air of selectable mass flow is preconditioned to an adjustable temperature upstream of the rectangular injection duct. The mass flow of air is measured using hot wire anemometer. A commercially available 6-hole (orifice diameter $d_{noz}=210\ \mu\text{m}$) pressure-driven disc-type injector operated at 9 bar pressure injected liquid discontinuously. Optical access to the injection site was available on all four sides. A longitudinal section of the injection site is shown in Figure 1. Shadow imaging, PIV scattering and PDA measurements were conducted. Phase-locked shadow images are made with 20 ns pulsed laser backlighting at different times during the injection cycle. 60 images are captured at each point in time, resulting in approximately 500 images for the assessment of the complete injection process at a specific exhaust flow condition. For PIV, two 4 ns pulses from Nd:YAG laser are generated with 25 μs interval. The laser sheet is formed horizontally along the channel length. Droplet PIV measurements are carried out at various vertical positions; here 786 images pairs are captured in each plane and processed to produce the mean spray velocity field by cross-correlation. Two-dimensional PDA measurements are conducted on the axis centerline $y=0\ \text{mm}$ at $z=-14\ \text{mm}$. PDA data on 10'000 droplets is gathered at every location for each exhaust flow condition and signal validation rates are in the vicinity of 95%. For further details, the reader is referred to [6].

Figure 1. Cross-section through the channel axis indicating the location and orientation of the injection nozzle (injection direction along dash-dotted line), windows and coordinate system. The blue dashed rectangle indicates the window extent in the channel transversal direction normal to the viewing direction. Adapted from [6].

The air conditions which are used in this study are given in Table 1. Simulation is done without spray for 0.5 s to allow the flow field to reach steady state. Water is injected for next 60 ms with a velocity of 27.4 m/s. The injector has 6 orifices each of diameter (d_{noz}) 210 μm and temperature of injection is 293 K [6].

Table 1. Gas flow conditions used in the present work

Case name	mfr (kg/hr)	T (°C)	ρ_o (kg/m ³)	U_o (m/s)
m100t200	100	200	0.746	5.8
m100t300	100	300	0.616	7.1
m100t400	100	400	0.524	8.3
m400t300	400	300	0.616	28.2

NUMERICAL SETUP

Simulations are conducted using the commercial CFD finite volume solver STAR-CD v4.20 [23]. The computational domain is explained in Figure 2 where red line indicates the axial location of the injector. All the dimensions are in millimeters (mm). The cross section of the rectangular channel is 80*80 mm² and the length is 500 mm. According to the mesh coordinate system the geometric range extends in the three dimensions $x=[0,500]$, $y=[-40,+40]$ and $z=[-40,+40]$; where the values are in millimeters. The injector is fixed on the ceiling of the channel at an axial distance of 120 mm from the channel inlet. The spray is inclined at 50° to the flow direction (positive x-axis). The three dimensional hexahedral mesh contains around 1.1 million cells. The mesh is refined near the walls and early trajectory of spray. The volume of refined cell is 2 mm³ and that of coarse cell is 16 mm³. A grid sensitivity study has been carried out, showing that the given mesh is sufficiently well resolved. For high pressure fuel sprays, considerations put forward in [24] recommend grids with spray transversal resolutions of the order of the nozzle diameter. However, the mesh resolution used in this study was found to be sufficient, in part due to the much lower injection pressures and velocities of AdBlue dosing systems.

Figure 2. Mesh used in this study. Dimensions are in mm.

Standard RNG $k-\epsilon$ turbulence model is used for the Reynolds-averaged Navier-Stokes (RANS) calculations carried out in this study. 'No slip' and 'adiabatic' boundary conditions with standard wall function formulation are used. Technical air consisting of Nitrogen and Oxygen is used instead of exhaust, for non-reactive simulations. Ideal gas law, Sutherland law and polynomial c_p are used to calculate density, molecular viscosity and specific heat respectively.

Two-Phase Treatment

No significant differences were observed between water and Urea-Water-Solution (UWS) spray characteristics in a previous study [6], hence water has also been adopted in the simulations. To treat the dispersed two-phase flow a fully two-way coupled Eulerian-Lagrangian technique is used with source terms in the respective conservation equations for mass, momentum, turbulence quantities, species and energy. Evaporation is modelled based on the correlation of Ranz-Marshall; further details can be found in [23].

For droplet atomization, several models are available in STAR-CD as given in [23], which have however been developed for high-pressure injection configurations. Two different approaches have been compared in this study: 1) the so-called 'blob' model, which injects mono-disperse droplets with diameters corresponding to the orifice diameter and 2) a prescribed droplet size distribution as discussed in detail below.

A variety of droplet secondary break-up models is available in the flow field solver, cf. [23]. In agreement with the experimental findings reached in [6], the inspection of droplet Weber numbers revealed, that secondary break-up does not occur. As a consequence, no significant differences in terms of droplet diameters and velocities at different locations could be observed between the Reitz-Diwakar, Pilch-Erdman, Hsiang-Faeth and the KHRT model. As a consequence, the choice of the model for droplet secondary break-up is not important as will be discussed later.

Inter-droplet collision is treated by the model proposed by O'Rourke (hence forth referred to as the 'standard' model) which distinguishes between coalescence, separation and bouncing but only droplets located in the same cell are considered for interaction. As outlined in [23], several extensions of the original model are implemented in STAR-CD in the so-called 'advanced' collision model. These include accounting for droplet time scale effects on coalescence and collision and further make use of a cell clustering approach which enables interaction of droplets in neighbouring cells.

RESULTS AND DISCUSSION

This Section is organized as follows: first, the effects of phenomena like buoyancy, spray-wall interaction etc. on the droplet distribution are studied. Later, assessment of primary atomization models is done using experimental data. The optimized numerical set-up is used to carry out simulations for different operating conditions and the results are validated to prove the model's applicability. This spray model is subsequently exercised to study various features of inclined sprays in cross flows.

A qualitative study is conducted to examine the effects of phenomena like buoyancy, droplet-droplet collision, spray-wall interaction and wall-film as captured by STAR-CD models.

Reitz-Diwakar primary atomization model injects droplets with diameters equal to the product of nozzle diameter and contraction coefficient (and is hence also termed 'blob' model). The choice of this model is motivated by the fact that all droplets are identical in size at the nozzle exit which enables detection of the impact of different phenomena on droplet diameters independently of the primary atomization process. The Basic model has 'Standard' droplet collision model and it doesn't feature buoyancy and film formation.

Figure 3 shows the scatter plot of droplets on the vertical plane passing through the centerline of channel. The droplet data is collected 10 ms after SOI. Collection of data at later times results in very large numbers of droplets and hence makes 'qualitative' comparison difficult. By comparing first and second images of Figure 3, it is perceived that 'buoyancy' shows only a moderate effect on droplet motion, as observed in previous literature too [4].

Figure 3. (cont.) Scatter plot of droplets present on the vertical plane passing through the channel center for m100t400 case. Reitz-Diwakar primary atomization model is used.

The 'Standard' droplet collision model of STAR-CD under-predicts coalescence of droplets as we can observe bigger droplets when 'Advanced' model is in use. Spray-wall interaction model is crucial to predict features such as rebound, spread, splash etc. Rebound and spread have greater impact on the flow under urea-SCR conditions considered in this study. 'Bai' model is found to be satisfactory in previous literature [3]. Creation of wall-film results in formation of fine droplets near the wall due to film stripping. This phenomenon can be observed in the fourth image of Figure 3 and hence it is decided to utilize the available 'Film' model [23]. Film formation has been proved to be crucial in other works [11, 12, 13, 21]. Formation of wall-film also causes creation of deposits which could be important for future studies. It can be observed that Reitz-Diwakar model assumes parent droplets of same size. Secondary atomization doesn't take place since the weber number is too low (less than 10) [6]. As a result, most of the droplets have similar diameters until they hit the wall which is not the practical case. This is expected since blob and most of the standard models are developed for high pressure diesel sprays where secondary atomization occurs soon [23, 25, 26]. On the other hand, Rosin-Rammler (RR) model assumes a droplet distribution based on user inputs which is more realistic. No droplet data is available near the nozzle exit since the dense spray region cannot be characterized experimentally. Consequently it's a challenging task to arrive at the 'correct' RR parameters at downstream locations where experimental data exists due to the various physical processes taking place between the nozzle exit and the validation locations. Most of the literature regarding low pressure sprays use RR model but there are no guidelines to arrive at the optimum RR parameters [1, 2, 3, 4, 18].

The choice of primary atomization model is critical for low-pressure sprays. Blob model predicts large droplets having higher momentum which prevents entrainment by the cross flow [23]. The deviation of spray by cross flow is also reduced. Instead, these large droplets hit the floor with great momentum and low incidence angle due to which they rebound to greater heights. Bending of the spray and entrainment of small droplets into the flow are important phenomena observed in urea sprays at high exhaust flow velocities. RR model assumes an ideal diameter distribution and we can observe all the features of

Figure 3.

sprays in cross flow. Based on the findings reported for both the blob and RR models, it can be concluded that RR model is a viable model and hence has been adopted also in previous studies.

Validation of droplet diameters is vital to predict features like entrainment of small droplets into the flow, evaporation and spray-wall interaction. PDA measurements are conducted on the plane 26 mm above the floor ($z=-14$ mm) and at different axial and transverse locations. In [Figure 4](#), CFD droplet diameters are compared for m100t200 case along the centerline ($y=0$) at axial locations 0mm and 50mm from spray axis (coordinates [165,0,-14] and [215,0,-14] mm respectively). Simulation droplet data are extracted at nine different times after the flow has attained quasi-steady state (time interval=0.5 ms; time range: 38 ms to 42 ms after SOI). Numerous values are tried for Rosin-Rammler shape parameter, k , and scale parameter, d_0 . It's decided to use $k=3$ based on the comparison of shape of droplet diameter distribution curves of experiments and simulations (not shown). Many previous studies have employed k in the range of 1.21 to 3.27 [1, 2, 3, 4, 18].

[Figure 4](#). Droplet diameter distribution for m100t200 conditions with different RR parameters. Measurements are at coordinates [165,0,-14] mm (left) and [215,0,-14] mm (right). Cases with $k=3$ and $d_0=50$ μm , 75 μm and 100 μm .

[Figure 4](#) shows the effect of d_0 on droplet diameters. It's observed that $d_0=50$ μm has close resemblance to PDA curves on the spray axis (i.e. [165,0,-14] mm). But CFD over predicts entrainment of droplets due to which the droplet distribution for $d_0=100$ μm shows better agreement for axial locations greater than 25 mm from the spray core (i.e. [190,0,-14]mm). On the right side of [Figure 4](#), it can be seen that $d_0=100\mu\text{m}$ has better prediction at location [215,0,-14] mm. PIV and shadow imaging comparisons are also better for $d_0=100\mu\text{m}$. $d_0=50$ μm case has numerous small droplets which are carried away by the flow resulting in coarse spray core. The leftover bigger droplets in this core are exposed to greater momentum exchange with the flow. Hence, the droplets of same size would experience higher drag in $d_0=50$ μm case and are removed before they reach 26 mm plane. This is the reason for absence of smaller droplets at [215,0,-14] mm in $d_0=50$ μm case since they are entrained earlier. Higher drag in $d_0=50$ μm case is also supported by the fact that large number of droplets at [215,0,-14] mm have lower velocities as seen in [Figure 5](#).

[Figure 5](#). Droplet u-velocity distribution for m100t200 case with different RR parameters. Measurements are at [165,0,-14] mm (left) and [215,0,-14] mm (right)

Finally, it is decided to use $d_0=100$ μm for future simulations since it matches PDA, PIV and shadow imaging data better than 50 or 75 μm cases. The left image of [Figure 6](#) clearly confirms that the 'blob' model assumes droplets of same size at the nozzle exit and these droplets almost retain their diameters till they reach 26 mm plane ($z=-14$ mm). Right image of [Figure 6](#) shows that droplets attain better distribution after the reflection from channel floor. On the whole, droplet diameter and velocity distribution are not good in blob model.

[Figure 6](#). Droplet diameter at coordinates [165,0,-14]mm (left) and [215,0,-14]mm (right) from Reitz-Diwakar model for m100t200 case.

[Figure 7](#) displays the droplet diameter distribution at different axial locations for m400t300 case with $k=3$ and $d_0=100$ μm . It's witnessed that the agreements are better for m400t300 case at both axial locations.

[Figure 7](#). Droplet diameter comparison of PDA and Rosin-Rammler model data for m400t300 case. Measurements are at coordinates [165,0,-14] mm (left) and [215,0,-14] mm (right). $k=3$ and $d_0=100$ μm .

The optimized CFD set-up is subsequently used to conduct simulations for all four conditions shown in [Table 1](#). The results are validated for all the cases using PIV, shadow imaging and PDA data.

PIV experiments have been conducted to estimate the droplet velocity field at different planes. [Figure 8](#) and [Figure 9](#) compare the droplet velocity magnitude and v-velocity predicted by CFD

and experiments at planes 16 mm ($z=-24$ mm) and 36 mm ($z=-4$ mm) respectively above the channel floor for m100t200 case. It is observed that v-velocities are predicted well by CFD. CFD is able to capture main features of inclined sprays in cross flow. At low cross flow velocity ($U_o=5.8$ m/s) and high spray velocity ($U_{\text{spray}}=27.4$ m/s), the spray core is too strong to be disturbed by the cross flow. Instead, gas curves around the spray core removing some of the small droplets from the border of the spray and leads to the formation of vortices. Spray core is strong in upper planes of the channel resulting in strong vortices. Higher recirculation is witnessed at 36 mm plane than at 16 mm plane. In the downstream, droplets decelerate to gas velocities due to drag. On the other hand, there is no prominent spray core and hence no vortices in a case with high cross flow velocities (m400t300 case: $U_o=28.2$ m/s, $U_{\text{spray}}=27.4$ m/s). The gas passes through the spray core removing most of the small droplets.

Figure 8. Comparison of PIV measurements (1st row), RR model (2nd row) and blob model (3rd row) at plane 16 mm above channel floor ($z=-24$ mm) for m100t200 case. Left half: velocity magnitude. Right half: v-velocity.

Figure 9. Comparison of PIV measurements (1st row), RR model (2nd row) and blob model (3rd row) at plane 36 mm above channel floor ($z=-4$ mm) for m100t200 case. Left half: velocity magnitude. Right half: v-velocity.

From Figure 10 and Figure 11 it can be observed that the droplets are accelerated by cross flow after the spray. These observations are supported by previous literature [14]. It should be noted that blob model is able to predict the droplet velocity field in a decent way even though the droplet diameter distribution is completely wrong. The velocity magnitude predictions are fair at high mass flow rates and relatively better for low flow rates of the gas.

Figure 10. Comparison of PIV measurements (1st row), RR model (2nd row) and blob model (3rd row) at plane 16 mm above channel floor for m400t300 case. Left half: velocity magnitude. Right half: v-velocity.

Figure 11. Comparison of PIV measurements (1st row), RR model (2nd row) and blob model (3rd row) at plane 36 mm above channel floor for m400t300 case. Left half: velocity magnitude. Right half: v-velocity.

Figure 12. Propagation of spray: Spray contours predicted by CFD- Rosin-Rammler model (red) and shadow imaging (green) at different times for m100t200 case. The 't' label within the figure indicates the time after SOI.

Figure 13. Propagation of spray: Spray contours predicted by CFD- Reitz-Diwakar model (red) and shadow imaging (green) at different times for m100t200 case. The 't' label within the figure indicates the time after SOI.

In the final step, spray boundaries estimated by simulations and shadow imaging experiments are compared in Figure 12 to validate the propagation of spray-tip. The determination of the experimental spray boundary is given in [6] and not repeated here. Numerically, scatterplots of the droplets scaled by their respective diameters have been employed to detect the boundary of the liquid portion of the spray; a scaling factor of $77\text{ }\mu\text{m/pixel}$ has been chosen to ensure that the complete spray is captured, since the core region is dominated by the large droplets. While the resulting images show sensitivity to the choice of the scaling factor in terms of the contiguous area between the individual droplets, the spatial extent of the liquid portion of the spray in terms of axial penetration were found to be (Error is less than 10 pixels or 0.7mm) unaffected by the choice of the scaling factor. The contours displayed in Figure 12 and Figure 13 are subsequently determined by binarizing these scatterplots.

The predictions over-estimate the spray-tip penetration in the beginning but, in general, the agreements are good. The spray spreading angle is overestimated by CFD at right-bottom corner of the spray. These discrepancies could be due to the selection of cut-off intensity for spray-boundary estimation in the experiments. In Figure 13, it can be observed that blob model is giving satisfactory results in terms of spray-tip propagation for m100t200 case. Based on the findings

presented above, the PIV, PDA and shadow imaging data validate the STAR-CD low-pressure spray model and it could be used for further analysis of inclined sprays in cross flows.

Figure 14 shows scatter plot of droplets on the plane $z=-14\text{ mm}$ for four cases considered in this study. These plots are obtained from CFD simulations and give a better picture of droplet velocity field. The vectors indicate u - and v -velocities, color-bar represents w -velocity and circle radius is proportional to the droplet size. From the first three graphs it is observed that the entrainment of droplets increases as temperature is increased while having a constant mass flow rate ($\text{mfr}=100\text{ kg/hr}$). The trajectories of droplets indicate the presence of vortices behind the spray core for low mass flow rates. The free stream velocity in cases m100t200, m100t300 and m100t400 are 5.8 , 7.1 and 8.3 m/s respectively. These values are significantly less than the spray velocity ($U_{\text{spray}}=27.4\text{ m/s}$). Hence the droplets' momentum is high due to which they are not easily entrained by the gas. As a result, the spray core is dense and acts like a solid cylinder. The gas flow across this dense spray core results in the formation of vortices. It can be observed that only small droplets from the edge of the spray are entrained by the flow. These small droplets are captured by the vortices behind the spray core and hamper their tendency to move downstream. For high mass flow rates ($\text{mfr}=400\text{ kg/hr}$), the free stream velocity ($U_o=28.2\text{ m/s}$ for m400t300 case)

is greater than the spray velocity. The gas has enough momentum to disrupt the spray core and entrain big droplets too. From the fourth graph of Figure 14, it can be observed that there is no clear spray core in m400t300 case. The spray core is coarse due to entrainment of significant number of droplets due to which there are no strong vortices behind the spray core. There are other theories to explain the nature of the droplet velocity field observed in this study. Bifurcation and blooming caused by Kelvin-Helmholtz instabilities [27] and counter-rotating vortex pair (kidney vortex) [28, 29] are widely accepted explanations. In-depth examination is required to gain additional knowledge regarding the same.

The droplets which have positive w-velocity (red and orange color) are moving up after rebounding from the floor. For mfr=100 kg/hr cases, we can observe 'red' droplets near the side-walls ($y=-40$ mm and $y=+40$ mm). It indicates that these

droplets had wide spray angles which caused them to deflect to the wall after 'rebound' from the floor. For mfr=400 kg/hr case, there are few droplets near side-walls. This phenomenon is due to the fact that most of the droplets near the edge of the spray are entrained by high velocity gas flows. According to the structure of the spray, droplets on the edge of the spray have wide spray angles. Since these droplets are entrained by the flow, the droplets reaching the floor ($z=-40$ mm) have small spray angle and they are deflected towards the centerline of the channel. Since the droplets are not reaching the sidewalls, the probability of film formation on side-walls is less. This prevents the formation of deposits on the side-walls which might improve the life-time of urea-SCR system. However, the droplets are restricted around the centerline of the channel which is not good for the distribution of ammonia near the catalyst entrance.

Figure 14. Location of droplets on the plane $z = -14$ mm. Circle size symbolizes droplet diameter, color-bar specifies w-velocity, and vectors show u and v velocities.

Figure 15. Film thickness on the channel floor for m100t200. The thickness values are in microns.

Preliminary investigations have been made regarding the formation of film on the floor ($z=-40$ mm). Figure 15 shows the variation of film thickness on the channel floor. It is observed that the film thickness increase with increase in x-coordinates. This phenomenon is observed in [21] and the CFD model is able to capture the trends. More validation by means of additional experimental data is needed before further conclusions can be made.

CONCLUSIONS

A combined experimental and numerical study is conducted towards improved understanding of inclined low pressure sprays in cross flows. The experimental data encompasses droplet PIV, two-dimensional PDA as well as shadow imaging data allowing for in depth validation of the CFD model. For the latter, a step by step approach has been followed to identify the most relevant spray processes. Model selection has been performed for one operating condition and subsequent validation is presented for three additional air flow conditions.

Concerning the spray sub-models it was seen that buoyancy only plays a minor role under SCR operating conditions, in agreement with previous findings reported in the literature. Conversely, spray-wall interaction plays a crucial role and the 'Bai' model predicts satisfactory results. The use of the 'advanced' droplet collision model was found to reproduce coalescence effects more accurately compared to the 'standard' model. The formation of films on the channel wall is an important phenomenon in urea-SCR systems; as a consequence the inclusion of a 'film model' was necessary for reliable predictions.

For the atomization process two options have been compared, namely the Reitz-Diwakar ('blob') model as well as the use of a prescribed Rosin-Rammler droplet size distribution. The former primary break-up model was originally developed for high pressure fuel sprays where rapid secondary break-up occurs. It was seen that despite capturing the macroscopic spray morphology evolution correctly, the Reitz-Diwakar atomization model is unable to predict good droplet diameter distributions for the low pressure assisted SCR atomizer configuration investigated. Adopting a prescribed Rosin-Rammler model was found to reproduce considerably improved droplet diameter distributions following careful calibration of the model parameters by means of the available experimental PDA data at several locations. Further validation of the model is performed by comparison with the PIV, PDA and shadow

imaging data. It is seen that the velocity magnitude of droplets is predicted satisfactorily at high gas flows while the estimations improve at low flow rates. The channel transversal velocity predictions are good for all four cases considered. The shadow imaging comparisons confirm that the spray morphology evolution is in good agreement also with the employed Rosin-Rammler distribution.

The results from the validated model further reveal several common features of sprays in cross-flows: at low cross flow velocities, the strong spray core results in the formation of vortices which prevent small droplets from travelling downstream. The strength of the spray core and vortices decrease with increase in gas velocities and gas manages to pass through the core removing some of the big droplets. The number of droplets found near the side-walls decreases for increasing gas velocities. In high cross-flow velocities, absence of droplets near the side-walls hinders the formation of film and it might enhance the life-time of urea-SCR system. On the other hand, the distribution of ammonia which is related to the distribution of droplets would be inefficient near the entrance of catalyst. On the channel floor, the thickness of film increases along gas flow direction.

ACKNOWLEDGEMENTS

Financial support from the Swiss Competence Centre for Energy and Mobility (project "NaDiP"), the Swiss Federal office of the Environment, the Swiss Oil Association as well as Liebherr Machines Bulle (project "DeMiCo") are gratefully acknowledged.

The authors further thank Dr. Graham Wigley for support concerning the acquisition of PDA data.

CONTACT INFORMATION

Achinta Varna
ETH Zurich
Sonneggstrasse 3, ML L16
CH-8092 Zurich
Switzerland
achinta.varna@lav.mavt.ethz.ch
www.lav.ethz.ch

NOMENCLATURE

CFD - Computational Fluid Dynamics

SCR - Selective Catalytic Reduction

PIV - Particle Image Velocimetry

PDA - Phase Doppler Anemometry

We - Weber number

q - Momentum ratio of liquid and gas

Re - Reynolds number of air

RNG - Re-Normalization Group turbulence model

RANS - Reynolds Averaged Navier Stokes

C_p - Specific heat of gas at constant pressure

d_{noz} - Nozzle orifice diameter

m_{fr} - mass flow rate of air

T - Free stream air temperature

ρ_o - Free stream air density

U_o - Free stream air velocity

U_{spray} - Spray velocity

d_o - Rosin Rammler scale parameter

k - Rosin Rammler shape parameter

RR - Rosin-Rammler

SOI - Start of Injection

u, v, w - Velocity components along x, y and z-axes

REFERENCES

- Birkhold, F., Meingast, U., Wassermann, P., and Deutschmann, O., "Analysis of the Injection of Urea-Water-Solution for Automotive SCR DeNOx-Systems: Modeling of Two-Phase Flow and Spray/Wall-Interaction," SAE Technical Paper [2006-01-0643](#), 2006, doi:[10.4271/2006-01-0643](#).
- Birkhold, F., et al., *Modeling and simulation of the injection of urea-water-solution for automotive SCR DeNOx-systems*. Applied Catalysis B: Environmental 70, 2007: p. 119-127.
- Jeong, S.J., Lee S.-J., and Kim W.-S., *Numerical Study on the Optimum Injection of Urea-Water Solution for SCR DeNOx System of a Heavy-Duty Diesel Engine to Improve DeNOx Performance and Reduce NH3 Slip*. Environmental Engineering Science, 2008. 25(7): p. 1017-1035.
- Ström, H., Lundström A., and Andersson B., *Choice of urea-spray models in CFD simulations of urea-SCR systems*. Chemical Engineering Journal 150, 2009: p. 69-82.
- van Helden, R., Verbeek, R., Willems, F., and van der Welle, R., "Optimization of Urea SCR deNOx Systems for HD Diesel Engines," SAE Technical Paper [2004-01-0154](#), 2004, doi:[10.4271/2004-01-0154](#).
- Spiteri, A.C. and Eggenschwiler P. Dimopoulos, *Experimental Fluid Dynamic Investigation of Urea-Water Sprays for Diesel SCR-DeNOx Applications*. Industrial & Engineering Chemistry Research, 2014. 53(8): p. 3047-3055.
- Montenegro, G. and Onorati A., *1D thermo-fluid dynamic modelling of de-NOx SCR systems for diesel engine exhaust gas after-treatment*. International Journal of Vehicle Design, 2006. 41: p. 285-306.
- Costa, M., Melo M.J., and Sousa J.M.M., *Spray Characteristics of Angled Liquid Injection into Subsonic Crossflows*. AIAA, 2006. 44(3): p. 646-653.
- Lee, S.S., Kim W.H., and Yoon W.S., *Spray formation by a swirl spray jet in low speed cross-flow*. Journal of Mechanical Science and Technology, 2010. 24(2): p. 559-568.
- Choi, S.M., et al., *Spray in Crossflow of a Rotary Atomizer*. Atomization and Sprays, 2012. 22(2): p. 143-161.
- Panao, M.R. and Moreira A.L.N., *Spray Impingement on a Flat Plate Under Cross Flow Conditions*, in ILASS-Europe 2002.
- Panao, M.R. and Moreira A.L.N., *Experimental Characterization of an Intermittent Gasoline Spray Impinging under Crossflow Conditions*. Atomization and Sprays, 2005. 15: p. 201-222.
- Panao, M.R., Moreira A.L.N., and Durao D.F.G., *Effect of a cross-flow on spray impingement with port fuel injection systems for HCCI engines*. Fuel 106, 2013: p. 249-257.
- Ghosh, S. and Hunt J.C.R., *Spray jets in a cross-flow*. Journal of Fluid Mechanics, 1998. 365: p. 109-136.
- Dekeyser, D., et al., *Assessment of orchard sprayers using laboratory experiments and computational fluid dynamics modelling*. Biosystems Engineering 114, 2013: p. 157-169.
- Endalew, A.M., et al., *A new integrated CFD modelling approach towards air-assisted orchard spraying. Part I. Model development and effect of wind speed and direction on sprayer airflow*. Computers and Electronics in Agriculture 71, 2010: p. 128-136.
- Endalew, A.M., et al., *A new integrated CFD modelling approach towards air-assisted orchard spraying-Part II: Validation for different sprayer types*. Computers and Electronics in Agriculture 71, 2010: p. 137-147.
- Bade, K.M., Kalata W., and Schick R.J., *Experimental and Computational Study of a Spray at Multiple Injection Angles*, in ILASS Americas 2010: Cincinnati, Ohio.
- Arcoumanis, C. and Cutter, P., "Flow and Heat Transfer Characteristics of Impinging Diesel Sprays Under Cross-Flow Conditions," SAE Technical Paper [950448](#), 1995, doi:[10.4271/950448](#).
- Kihm, K.D., Lyn G.M., and Son S.Y., *Atomization of Cross-Injecting Sprays into Convective Airstream Atomization and Sprays*, 1995. 5: p. 417-433.
- Arienti, M., et al., *Modeling Wall Film Formation and Breakup Using an Integrated Interface-Tracking/Discrete-Phase Approach*. Journal of Engineering for Gas Turbines and Power, 031501, 2011. 133(3).
- Özdemir, I.B. and Whitelaw J.H., *Impingement of an unsteady two-phase jet on unheated and heated flat plates*. Journal of Fluid Mechanics, 1993. 252: p. 499-523.
- CD-adapco, *Methodology Star-CD Version 4.16*. 2012.
- Pizza, G., et al., *Evaporating and non-evaporating diesel spray simulation: comparison between the ETAB and wave breakup model*. Int. J. Vehicle Design, 2007. 45: p. 80-99.
- Reitz, R. and Diwakar, R., "Effect of Drop Breakup on Fuel Sprays," SAE Technical Paper [860469](#), 1986, doi:[10.4271/860469](#).
- Reitz, R. and Diwakar, R., "Structure of High-Pressure Fuel Sprays," SAE Technical Paper [870598](#), 1987, doi:[10.4271/870598](#).
- Reynolds, W.C., et al., *Bifurcating and blooming jets*. Annual Review of Fluid Mechanics, 2003. 35: p. 295-315.
- Fric, T.F. and Roshko A., *Vortical structure in the wake of a transverse jet*. Journal of Fluid Mechanics, 1994. 279: p. 1-47.
- Kelso, R.M., Lim T.T., and Perry A.E., *An experimental study of round jets in cross-flow*. Journal of Fluid Mechanics, 1996. 306: p. 111-144.