FLOW IN NOZZLES AND ITS INFLUENCE ON SPRAY BEHAVIOUR

Raúl Lima Ochoterena and Sven Andersson

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

Telephone: +46(0)31 772 52 99, Fax: +46(0)31 18 09 76,
E-mail: ochoterena@tfd.chalmers.se

ABSTRACT

The spray behaviour is strongly affected by the flow in the injection nozzle. It has been settled that cavitation and turbulence in a nozzle play an important role on the atomisation process. Therefore the influence on the atomisation process by the flow inside a nozzle, and its variations, was studied to highlight the influence of turbulence together with cavitation on the spray behaviour. Particle Image Velocimetry (PIV) measurements were done to obtain the velocity fields and turbulence intensities in the interior of three different up scale and transparent nozzles and also of the spray near the nozzle outlets. The flow seeding technique, developed for this work, was based on microbubbles generated by electrolysis using water as working fluid. Results of this work show that the used seeding method is effective and convenient. Also it has been observed that cavitation is present all the times that atomisation was achieved while turbulence could be present even though atomisation process was not successful.

1 INTRODUCTION

In order to understand spray behaviour it is important to know the flow conditions downstream and upstream of the nozzle outlet. The break-up of the jet downstream the nozzle is governed by different mechanisms depending on the conditions. The break-up in stagnant media is when a fluid overlays a less dense fluid, such as liquid injected into gas, perturbations at the interface tend to grow promoting the break-up into smaller droplets mainly due to Rayleigh-Taylor instability [1]. Regardless all the efforts made to understand atomisation, the basic phenomena that governs sprays conduct is not perfectly empathised.

The flow upstream the nozzle outlet has a strong influence on the atomisation process; therefore it is necessary to know the characteristics of the flow inside the nozzle. It has been thought that turbulence [2] and cavitation [3] inside a nozzle influence the spray behaviour, mainly primary and secondary break-up, but more studies must be performed to understand the complete phenomena. In fact, it is hard to isolate cavitation from turbulence, and turbulence from cavitation, due to the fact that cavitation occurs at high velocity flow when turbulence is frequently present. However the influence of each, individually or both of them together, are crucial in the liquid transformation from liquid bulk to droplets.

Recently a primary break-up model was introduced by von Berg *et al.* [4] and compared with results from experiments of asymmetric sprays, made at Chalmers by Ganippa *et al.* [5]. The study presented here is a continuation of these experimental studies. A contribution to understand the basic atomisation phenomena and the influence of cavitation and turbulence on sprays is made in the present work, pointing out velocity and turbulence measurements.

2 EXPERIMENTAL SETUP

Optical studies were done in transparent nozzles using Particle Image Velocimetry (PIV) technique. PIV was chosen to obtain the flow velocity profile inside the nozzles and spray zones. Microbubbles generated by electrolysis were used as a seeding technique during experiments. The experimental setup consisted of different devices that worked together supplying pressurised working fluid to the nozzles under study. Figure 1 shows the experimental setup and lists its different components without including the seeding neither the PIV systems.

The bubble generator (seeding system) was fitted upstream the settling chamber, due to the fact that inside the settling chamber there is a honeycomb device that minimise the turbulence carried from the pump. If the seeding system were fitted downstream the settling chamber, it would minimises the mixing between seeders and the working fluid. Three different transparent nozzles made of Plexiglas were used [5]. Two with the orifice aligned with the settling chamber axis and another with a right angle between the nozzle orifice axis and the

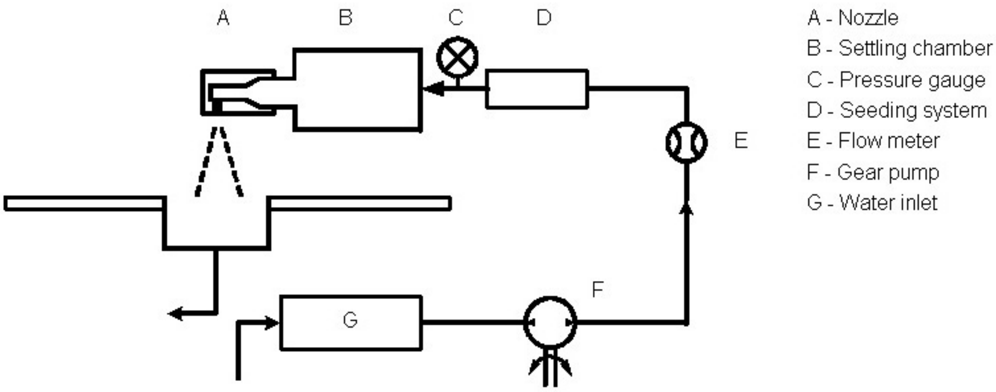


Figure 1:
Experimental setup (Chalmers Water Rig).

settling chamber axis. The flow in the middle section of each nozzle was studied. Therefore a laser light sheet was used to illuminate the particles travelling only in the middle section of the studied nozzle.

2.1 Seeding system

Because of the high relation between drag and buoyancy, small bubbles appeared to be a very attractive seeding technique, also microbubbles generated by electrolysis have an accurate and dominant size value very near to the mean. The bubbles average size is mainly a function of the electrode material, diameter, electrical tension and electrolyte. Microbubbles are a very competitive seeding choice because of its stability which obey to the fact that a reduced size bubble gives a big relation between surface tension forces and shear, permitting them not to implode or coalesce even in adverse pressure changes.

A simplified sketch of the developed bubble generator is presented in Figure 2. It can be seen a tube with interchangeable electrodes inside with supports made by a dielectric material, also it can be seen the electrical connectors outside the tube on the dry zone.

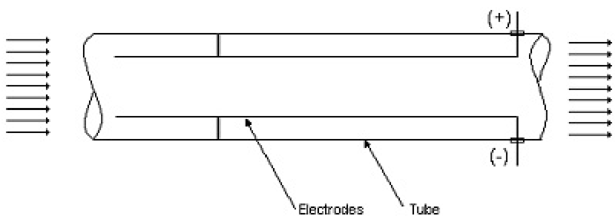


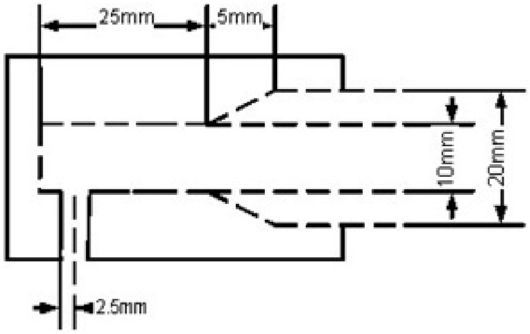
Figure 2:
Developed microbubbles generator.

The electrodes material was aluminium and had a diameter of one and a half millimeters. The amount of detached bubbles were controlled through varying the applied electric intensity. In order to make water an electrical conductor sodium chloride was added into the water with a concentration of 0.02 M. Water temperature was 291 K.

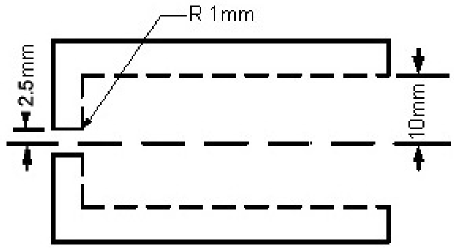
2.2 Nozzles

The nozzles used were made of Plexiglas. One of them has different geometrical characteristics from the rest, as the angle of the nozzle. The other characteristics, like internal hole diameter and length, are the same.

The nozzle embossed in Figure 3a is known in this work as 90 degrees nozzle or perpendicular nozzle. Figure 3b is the 0 degrees nozzle with a straight and smooth inlet (slightly roundly corners). The third nozzle (which is not presented on the drawing) has the same geometrical characteristics as the 0 degrees nozzle, except that this specific one, has no rounded corners at the inlet of the orifice. This is presented here as straight and sharp inlet nozzle.



(a) 90 degrees nozzle



(b) 0 degrees nozzle (smooth corners)

Figure 3:
Drawings at a middle cut of the studied nozzles.

3 RESULTS

PIV measurements were done for each nozzle at two flow regimes. Velocity profiles and turbulence intensity were obtained in a horizontal cut at the middle of each nozzle for the different cases, in such a way that the flow inside the nozzle and the spray were both illuminated simultaneously. In all the cases the main flow comes from the bottom of the paper to the top. The jets were injected with quite low velocities (order of 10 m/s) into a quiescent atmosphere at room conditions (1 Bar, 291 K). Thus atomisation because of aerodynamic forces is small and it is almost entirely caused by cavitation and turbulence in the orifice.

3.1 Straight and rounded inlet nozzle (0 degrees nozzle with smooth corners)

This nozzle is a non-atomising one. That means that a clear blast appears at the outlet of the nozzle. In speed distribution graphs (Figure 4c and Figure 4d) it can be seen that the velocity profile and surface section of the jet, are almost constant since the nozzle contraction entrance. In Figure 4e, at low Re, it appears that turbulence is low all over the path of the fluid except at the inlet of the contraction zone, starting to get dissipation downstream of the entrance of the orifice and achieving almost zero levels at the jet zone.

Increasing the flow rate some interesting phenomena can be seen. Even though turbulence inside the contraction zone is high (Figure 4f) neither cavitation nor atomisation appear (Figure 4b), maintaining a constant section and velocity (Figure 4d) since the inlet contraction zone. In this case turbulence is not fully dissipated along the jet.

3.2 Straight and sharp inlet nozzle (0 degrees nozzle with acute corners)

This nozzle is an atomising one. That means that a spray will appear downstream the nozzle exit. For this nozzle cavitation is existent inside the contraction zone of the nozzle for almost all operational conditions. The cavitation length increases according to the increase of the flow, as well as the diameter of the spray. In cases when cavitation is covering all the length of the contraction zone, hydraulic flip will appear if the injection pressure is increased enough, being in this very case a non-atomising nozzle.

3.3 Perpendicular nozzle (90 degrees nozzle)

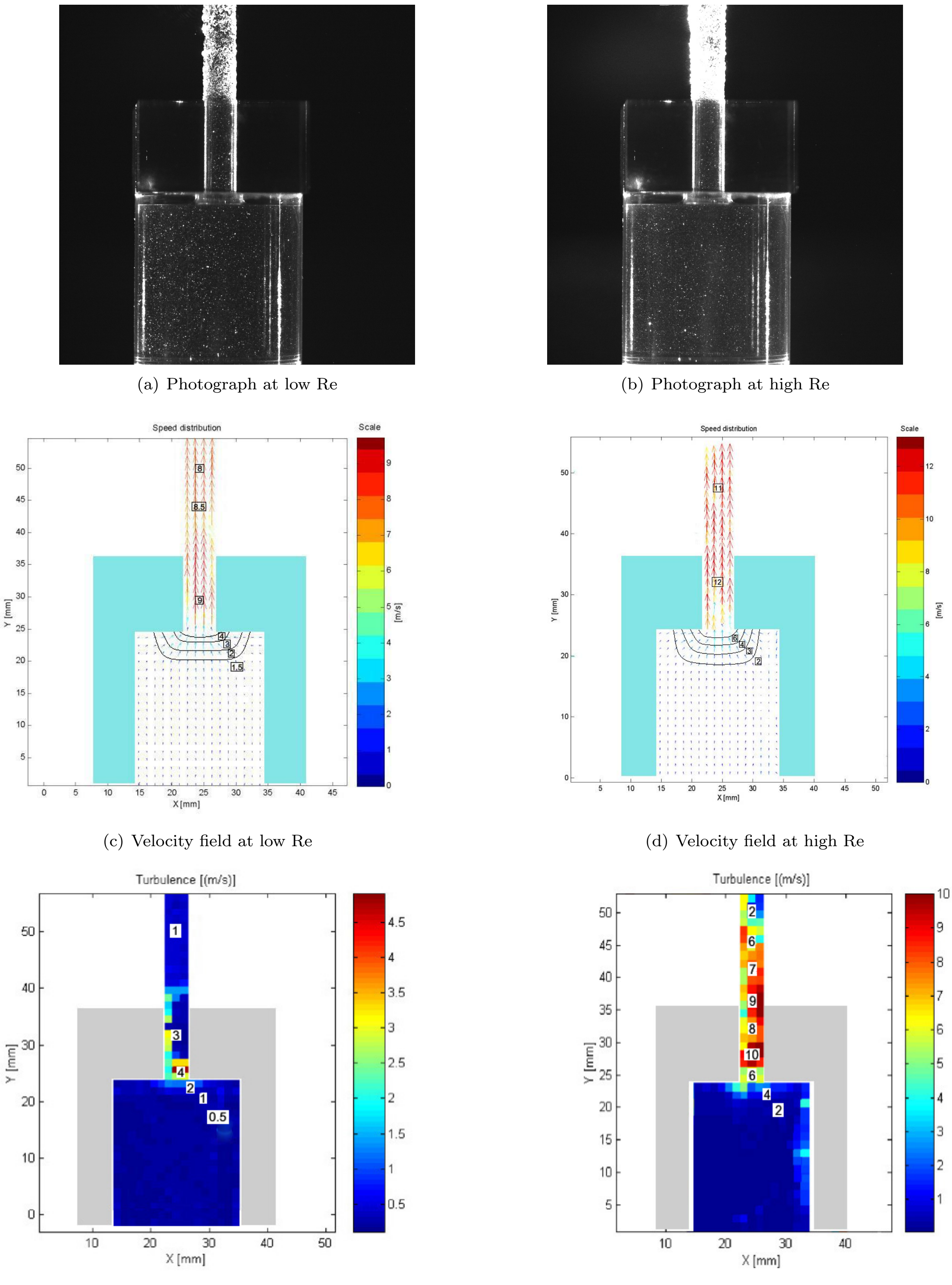
For data presented in Figure 5 it is important that for the interpretation of the PIV measurements on the spray zone, multiphase flow must be considered, due to the diminution of the spray density as the distance between the nozzle outlet increases.

Because of cavitation at high Re it is difficult to obtain quantitative data regarding the flow properties inside the contraction zone. In these figures it can be seen that the maximum turbulence values, between the two operational regimes, are not far distant one from the another (Figure 5e and Figure 5f), but the cavitation in between both operational regimes does differ drastically, as it can be shown on Figure 5a and Figure 5b.

4 CONCLUSIONS

Microbubbles has proven to be an inexpensive and efficient seeding method for PIV measurements. Results have shown the velocity field and turbulence of the spray core and atomised droplets of the spray. Different constant speed zones on the spray were noticed.

Turbulence is dominant and continuously present in the axis of the contraction zone and spray core. The turbulence diminishes in the region out of the core of the atomization zone as long as the most exterior zone of



(a) Photograph at low Re

(b) Photograph at high Re

(c) Velocity field at low Re

(d) Velocity field at high Re

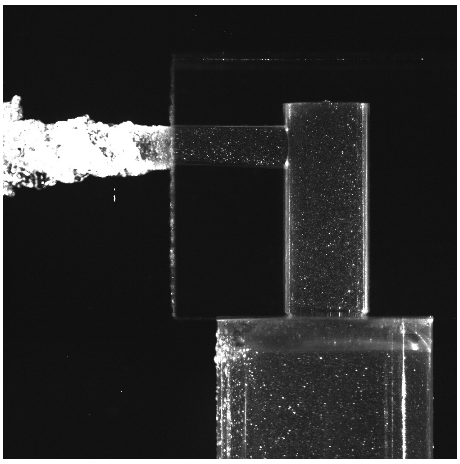
(e) Turbulence field at low Re

(f) Turbulence field at high Re

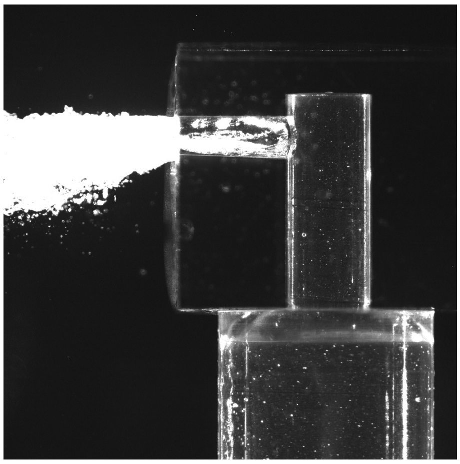
Figure 4:
Photographs, and velocity and turbulence fields for the 0 degrees nozzle with smooth inlet corners.

the spray is not reached. The turbulence on the most exterior layer of the spray is higher than in the middle layer probably because of aerodynamic forces.

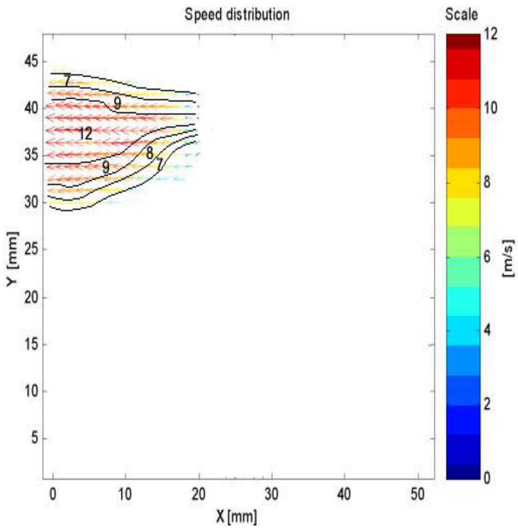
Cavitation was present all the times that atomisation was achieved. Turbulence intensity was high even though atomisation was not accomplished when the straight and rounded nozzle was under study. Therefore cavitation appears to be a more relevant factor in atomisation than turbulence, but it must be stressed that both of them are almost always present.



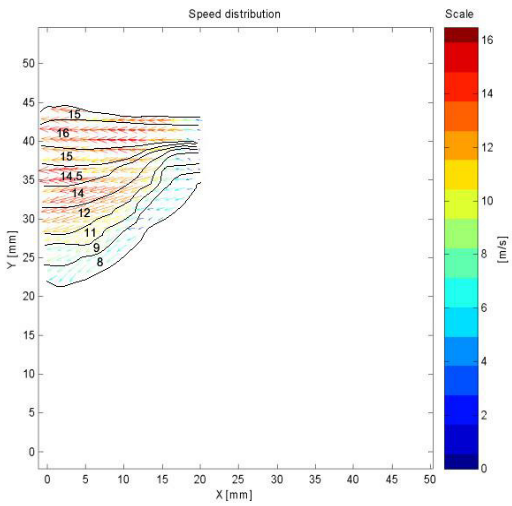
(a) Photograph at low Re



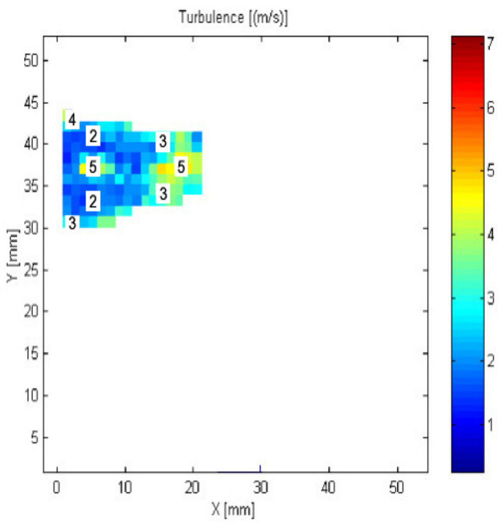
(b) Photograph at high Re



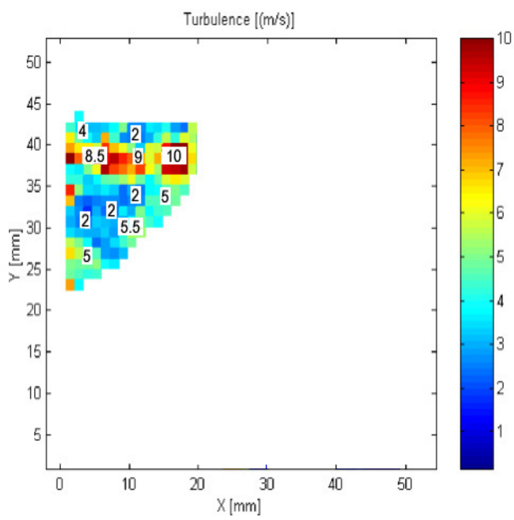
(c) Velocity field at low Re



(d) Velocity field at high Re



(e) Turbulence field at low Re



(f) Turbulence field at high Re

Figure 5:
Photographs, and velocity and turbulence fields for the perpendicular nozzle (90 degrees nozzle).

5 ACKNOWLEDGMENTS

We would like to thank Volvo Technology for their support during the measurement campaign and The Combustion Engine Research Center at Chalmers (CERC) for their financial support to this project.

References

[1] Lamb, H., "Hydrodynamics.", 6th Edition. Dover, New York., (1932).

[2] Reitz R.D. and Bracco F.V., "Mechanism of Break-up of Round Liquid jets.", The Encyclopaedia of Fluid Mechanics., ed. Cheremisnoff, **Vol. 3**, 233-243 (1986).

[3] Bergwerk W., "Flow pattern in diesel nozzle spray holes.", Proc. Inst. Mech. Eng., **173:25**, 655-660 (1959).

[4] von Berg E., Grief D., Tatschl R., Winklhofer E., Ganippa L.C. "Primary break-up model for diesel jets based on locally resolved flow properties in the injection hole." ILASS-Europe, 9-11 September, 2002, Zaragoza, Spain.

[5] Ganippa L.C., Bark G., Andersson S. and Chomiak J., "A Contributory Factor in the transition from Symmetric to Asymmetric Jets in Cross-Flow Nozzles.", Experiments in Fluids., **36:4**, 627-634 (2004.)