CFD PREDICTION OF THE EFFECTS OF VISCOSITY ON THE INTERNAL FLOW OF A SCALE PRESSURE-SWIRL ATOMISER.

S.Shaikh*, U.Banaszak**, E.Von Lavante**, D.Cooper* and A.J.Yule*
****Dept.Mechanical Engineering, UMIST, PO Box 88, Manchester, M60 1QD, UK.***
*****University of Essen, D-45127, Essen, Germany.***

Email:samiashaikh2001@yahoo.co.uk

ABSTRACT

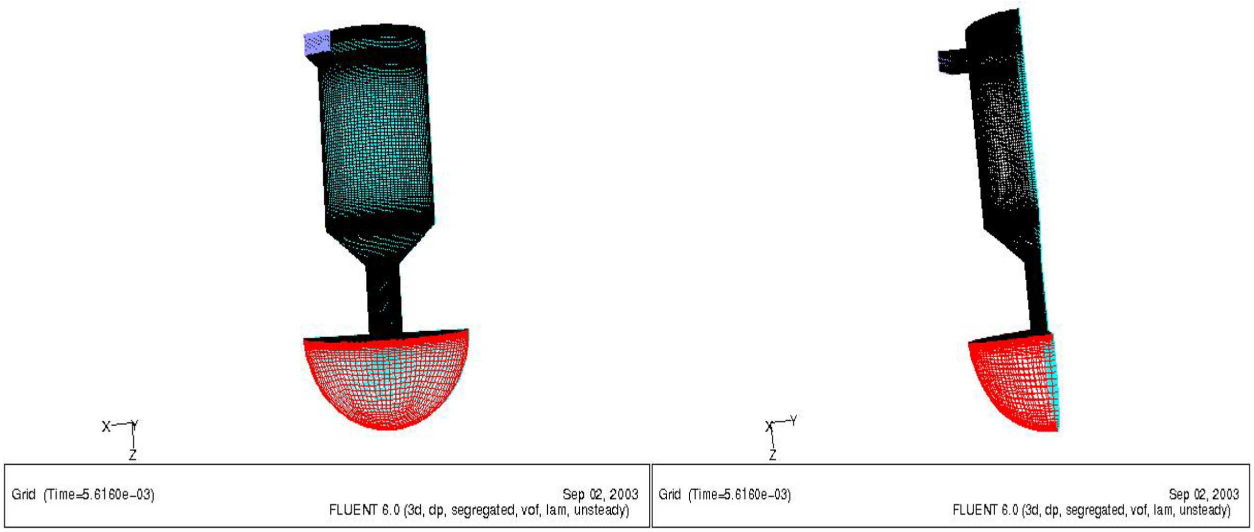
This paper predicts the effect of viscosity on the internal flow of a pressure swirl atomiser (PSA) using computational fluid dynamics (CFD). This work was in collaboration to the experimental work carried out at UMIST by Cooper et al, [1, 2, 3] and presented at earlier ILASS conferences, and Shaikh [4]. Both 2D and 3D simulations were made, where the boundary conditions for both were the velocity at the inlet and the pressure at the outlet. The CFD simulation has been carried out on Fluent version 6.1.18, using a segregated implicit solver with first order accuracy in time with a volume of fluid (VOF) model was used. The CFD data obtained qualitatively agrees with the experimental data and supports the experimental hypothesis that the fluid may be behaving like a semi-solid body.

INTRODUCTION

CFD simulations were made to predict the internal flow of a pressure swirl atomiser, using the commercial Fluent code. Previous work by Cooper [1] demonstrated the presence of Gortler vortices in the near wall region also Cooper [1] had found large toroidal secondary vortices within the swirl chamber. This was reinforced by the computational study carried out by Chinn et al [2] but the computational study also showed that the size and the position of the toroidal vortices alter slowly with time. In the previous case water was used as the liquid, in this paper a more detailed examination of the internal flow of an atomiser with a high viscosity liquid will be given. It had been hypothesised that the flow within the PSA may be behaving like a semi-solid body. The liquid used was a glycerol-water mixture at room temperature, where the ratio of glycerol to water was 3:2 by volume, this gave a viscosity at 20˚C of 11.77 +/- 0.05mPas.

COMPUTATIONAL MODEL

The fluid was treated as a laminar unsteady fluid. The discretization scheme used for pressure was a body force weighted scheme, and also the PISO (pressure implicit splitting of operators) scheme for the pressure-velocity coupling and 2nd order upwind schemes were used for the momentum and swirl. In the actual simulation the effect due to gravity was also taken into consideration. Both 3-D and 2-D simulations were made where the boundary conditions for both were at the velocity at the inlet and the pressure at the outlet. The solver used in the 2D simulation was an axisymmetric solver with swirl. The 2-phase model used in the simulation was the volume of fluid model, with geo-reconstruct. As seen in Figure 1, an imaginary outlet plenum chamber was modelled at the outlet of the atomiser. This had a porous constant pressure boundary and was used in order to avoid enforcing unrealistic conditions at the outlet of the atomiser. The Fluent data were saved as a compressed binary file, which when decompressed allowed the mapping of U (axial velocity), V (radial velocity) and W (tangential velocity), static pressure and cell Reynolds number.



(a) **(b)**

Figure 1-Atomiser Mesh (a) Front View (b) Side View

COMPUTATIONAL RESULTS

The liquid/air volume fraction is shown in Figure 2, and it can be seen that in the top region of the air core there is a slight expansion moving downwards. This expansion corresponds to an increase in the axial velocity in that part of the air core, as shown in Figure 3. In the region near the atomiser inlet it can be seen that there is a zone of high tangential velocity, this corresponds well with the dynamic pressure contours shown in Figure 4.

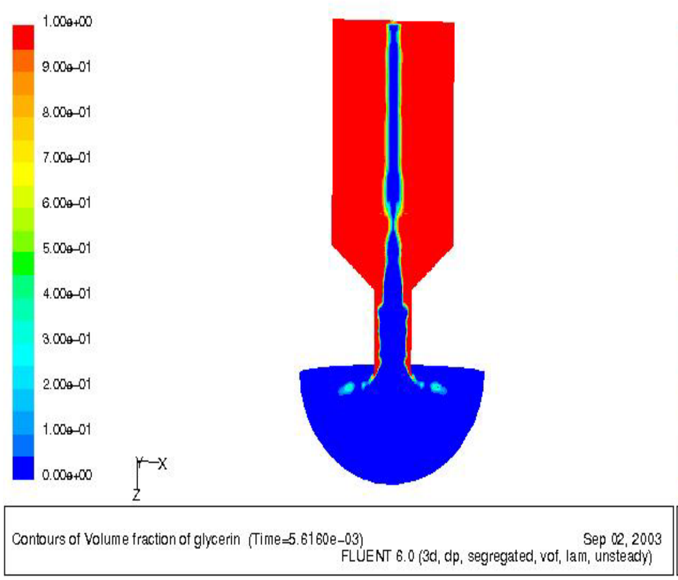


Figure 2-Volume Fraction of Glycerol-Water to Air

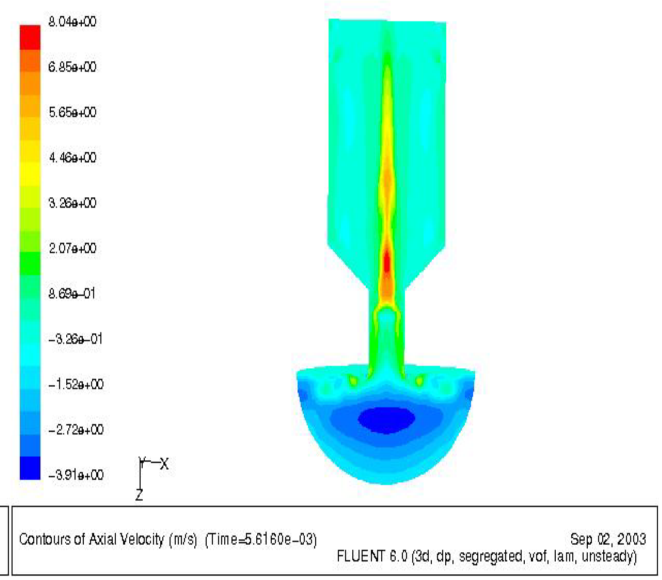


Figure 3 – Axial Velocity Contours

In reference to the earlier hypothesis, the high axial velocity at the edge of the air core may be due to 'Couette Flow' type of behaviour exhibited by the air core and the main flow. The air core and the main flow are behaving as two cylinders, one within the other, moving at different velocities relative to each other. The main shape of the air core is observable with all the different properties (i.e. volume fraction of glycerol-water to air, axial velocity, dynamic pressure and cell Reynolds number) as can be seen in the Figure1, 2 4 and 8, respectively.

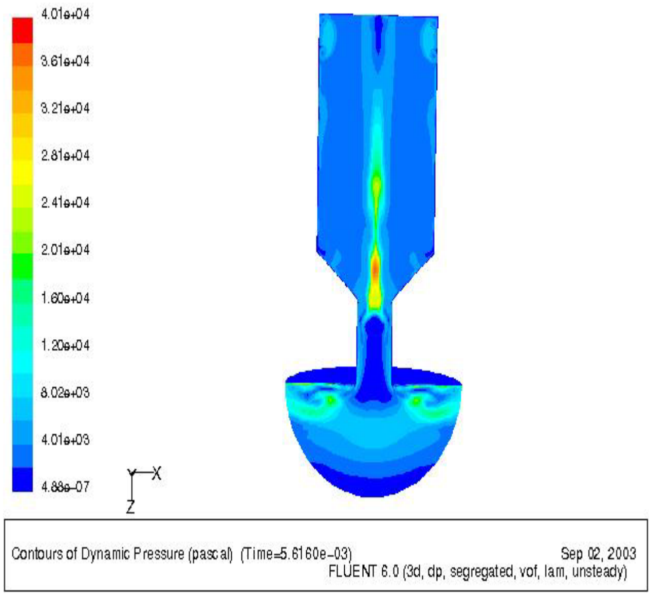


Figure 4 – Dynamic Pressure Contours

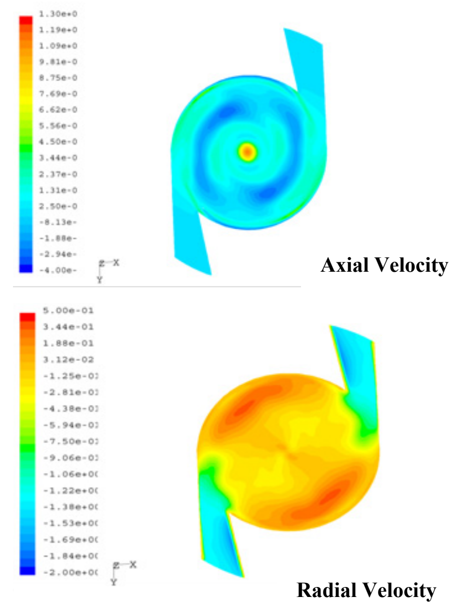


Figure 5 – Horizontal Cross Section of Atomiser at Mid-Plane of Inlet

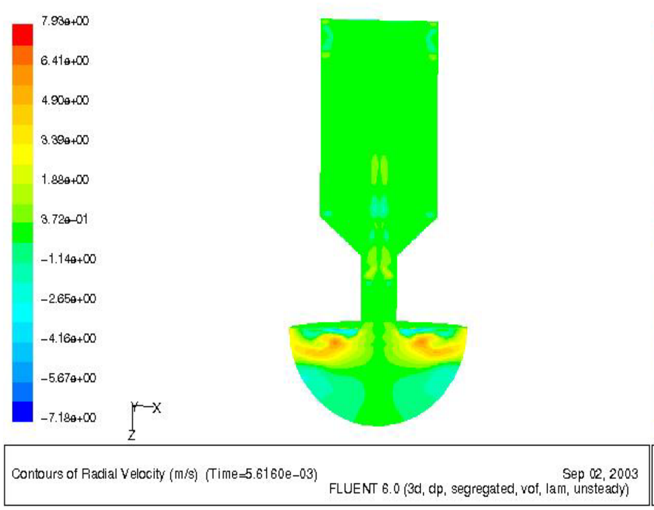


Figure 6 – Radial Velocity Contours

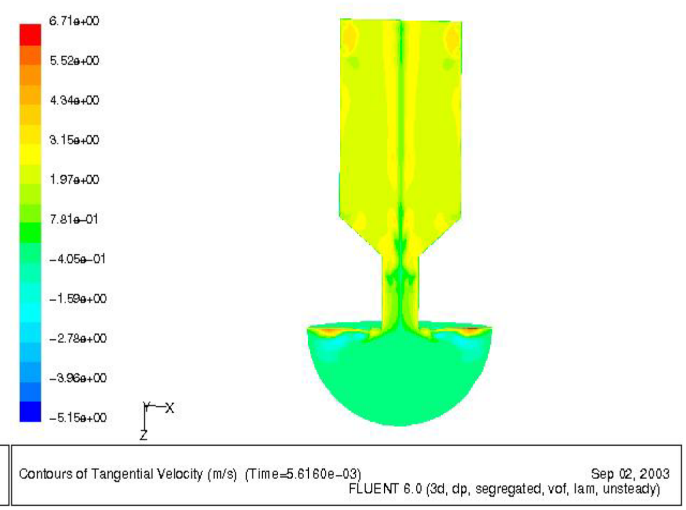


Figure 7 – Tangential Velocity Contours

Figure 8 indicates that the flow is turbulent, in order to get a true cell Reynolds number profile, each cell will have to be divided up into a further 1000 cells (i.e.10x10x10, in the x, y and z direction). This is a computational restriction, as this would require an unrealistically large amount of storage and computational time. However, if this was possible it is hypothesised that, one would be able to view the local turbulent activity in each of the cells that is the presence of small eddies. Therefore, at a microscopic level the internal flow is turbulent, but at the macroscopic level the flow is laminar, due to the rotational effect of the flow stabilizing the overall flow, i.e. surpressing the turbulence.

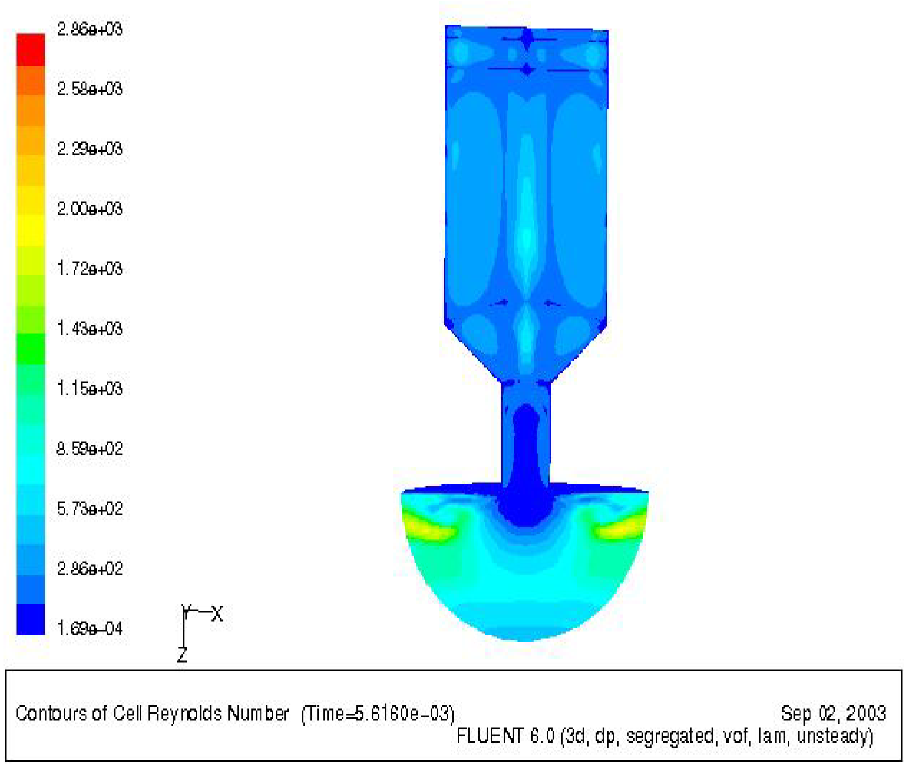


Figure 8-Cell Reynolds Number

Figure 5 shows the contours of the axial and the radial velocity components in the plane containing the two inlets. It can be seen that in the near wall region the axial velocity is relatively high but as the fluid inside the chamber starts to move under the incoming fluid, demonstrating a cork-screw type of behaviour, there is another thin annular region similar to the near wall region, where the axial velocity is high. It was hypothesised that this may be due to the liquid moving under the new incoming flow, which causes blockage similar to a solid stationary surface. In the central region, near the air core, a more pronounced area of increased axial velocity than in the near wall region can be seen. This increase in the axial velocity may be due to the increased viscosity of the liquid in comparison to the experimental results obtained with water [1]. As the liquid is rotating it is demonstrating a 'Couette Flow' type of behaviour, where it is recognizing other layers of the liquid as 'solid stationary layers'.

The glycerol-water solution is behaving like a semi-solid body, where a disturbance in the flow would damp out and not propogate through the liquid, however, with water a disturbance in the liquid would propogate through the water. From Figure 6, it can be seen that there is no major radial velocity component in the atomizer. This is also true for the tangential velocity plot (Figure 7), where the majority of the flow has a uniform tangential velocity.

However, it can be seen that in the axial velocity contours (Figure 3) there is a high velocity in the near wall region and the edge of the air core. This increase in the axial velocity is very noticeable in the nozzle exit region, which may be related to the film thickness of the liquid in the nozzle exit region. As mentioned earlier, the high axial velocity at the edge of the air core may be due to 'Couette Flow' type of behaviour exhibited by the air core and the main flow, where the air core and the main flow are behaving as two cylinders, one within the other, moving at different velocities and possibly directions relative to each other.

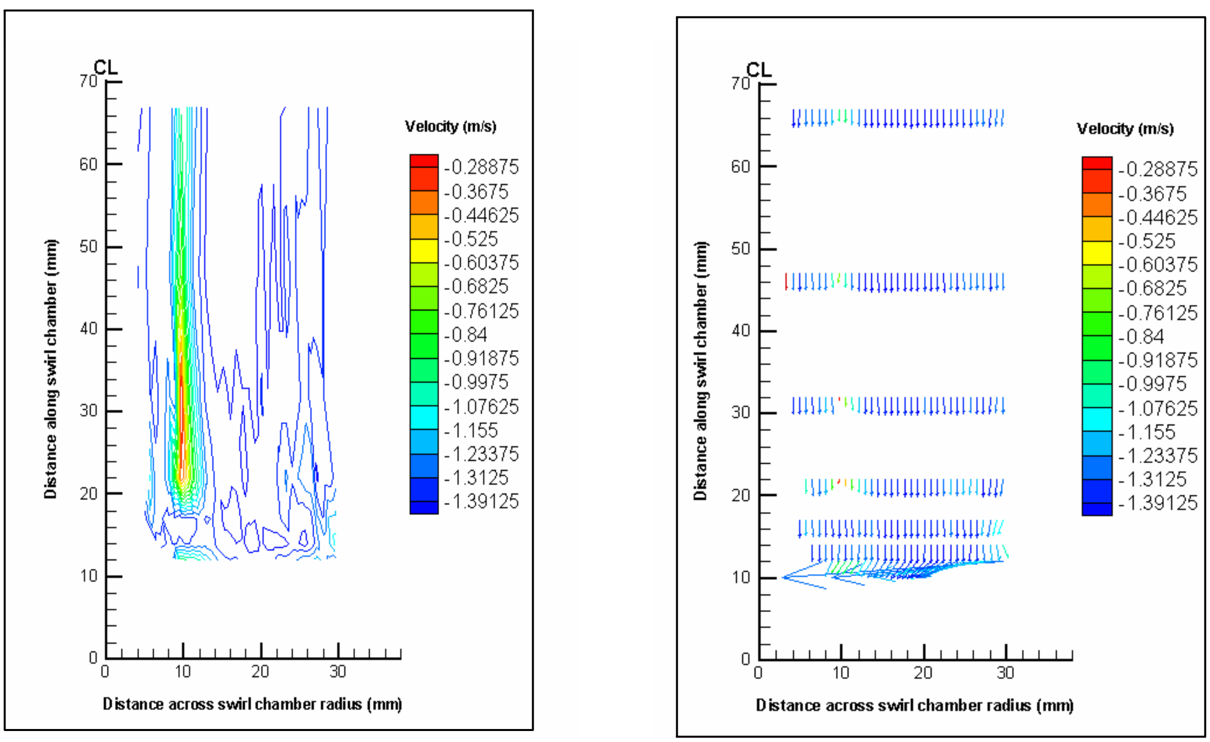
VALIDATION OF COMPUTATIONAL RESULTS WITH EXPERIMENTAL DATA

The experimental results were obtained using a Dantec Phase Doppler anemometry (PDA) system used in Laser Doppler Anemometry (LDA) mode. However, due to time constraints, only the axial velocity could be measured. This system is usually used for measuring both particle size and velocity, as the velocity was only being measured, then two of the three photo-detectors, normally used for signal phase measurement were disabled. A Dantec 3-D traverse unit was used with a resolution of 0.1mm on all axes. In order to make the traversing of the system easier, the LDA was configured in the backscatter mode. The receiver was 60˚ off-axis, so that both the receiving and transmitting optics could be easily traversed together.

The PDA system comprised of a 58N50 enhanced signal processor, a Fiber Flow 60X41 transmitter, a 57X40 optical probe and 57X10 receiving optics. The laser light was produced by an AEG 100mW air cooled Argon ion laser using the 514.5mm wavelength (green). The 57X40 optical probe has a 310mm focal length with a beam spacing of 74mm, hence giving a control volume of $\delta_x=\delta_y=0.0768$mm and a length of $\delta_z=0.6441$mm, where the control volume was the region which contains the fringes. The fringe spacing was 2.1706 µm and the velocity range was –2.71m/s to 6.51 m/s.

Comparing Figures 3 and 9, the computational results are in agreement with the experimental data. In the experimental data a peak can be seen in the near wall region, this is also visible in the CFD data. However, at the edge of the air core, the experimental data demonstrates a drop in the axial velocity, where the CFD data demonstrates an increase in the axial velocity.

From Figure 9, the sudden decrease in the velocity at 10mm from the air core corresponds with the decrease in the cell Reynolds number as can be seen in Figure 8, this is occurring at the edge of the air core. This sudden decrease in the axial velocity becomes more pronounced as the liquid moves down the swirl chamber towards the exit, this is also in agreement with Figure 8, where the area of low Cell Reynolds Number increases as the liquid moves further down the swirl chamber. It can be seen that in the near wall region, the axial velocity (Figure 3) decreases slightly, however in the experimental data (Figure 9) it can be seen that there is an increase in the velocity in the near wall region. This can also be seen in the cell Reynolds number plot (Figure 8). This behaviour may be due to the viscous shear stresses in the near wall region.



(a) **(b)**

Figure 9 – (a) Iso-Contours of U (axial) Velocity Component for Glycerol-Water Case and (b) Velocity Vectors for Glycerol-Water Case

Figure 9(b) shows that there is an increase in the velocity as the liquid is moving towards the nozzle exit, this agrees very well with the CFD results where there is a sudden increase in the axial and tangential velocity components (Figures 3 and 7). This is also supported by an increase in the total pressure relative to the ambient pressure, the dynamic pressure and the absolute pressure, as the velocity increases through the nozzle exit.

It should be noted that the velocities obtained from the experimental data are of inverted sign due to the way the LDA equipment was configured, however the velocities obtained from the CFD data also have inverted signs, as the axis used was configured in the opposite direction. Figures 9(a) and (b) exhibit negative velocities that demonstrate that the flow is moving downwards towards the nozzle exit, this is in agreement with the CFD data as can be seen in Figure 3, where the main flow within the chamber is also moving downwards towards the nozzle exit. The flow in both the near wall region and the edge of the air core is moving upwards, in the CFD case suggesting that some recirculation is occurring although the experimental data does not show this effect, however earlier experimental results for water obtained by Cooper [1] does demonstrate recirculation in the flow.

CONCLUSIONS

- The CFD data and experimental data generally agree with each other.
- The axial velocity iso-contours (Figure 9a) demonstrate minima in axial velocity that correspond to the edge of the air core; this also corresponds to the sudden decrease in cell Reynolds number from the CFD data.
- Both the CFD and the experimental data, show the presence of the air core contraction region, although the CFD data shows an increase in the axial velocity, whilst the experimental data shows a decrease in the axial velocity.
- The axial velocity plot from the CFD data shows the presence of high velocities in the near wall region and the edge of the air core, this can also be seen experimentally in Figures 9(a) and 9(b).
- The glycerol-water solution, demonstrated a Couette Flow type of behaviour, as the flow is acting a semi-solid body.

FURTHER WORK

In the current study, particular phenomena have been noticed in particular flow regions, in both the experimental and CFD data. Unfortunately due to time constraints, the CFD data and experimental data could only be obtained at one flow rate. Therefore it is proposed that further study should be carried out at the same flow rate, but different viscosities, also different flow rates should be explored. However, in order to obtain a more detailed analysis of the internal flow of the atomizer, experimental readings should be taken at different planes. Also as the CFD data presented is from one particular point in time, it would be beneficial to simulate the internal flow over a longer period of time and compare with the experimental data.

REFERENCES

[1] **Cooper, D.**, *The Internal Flow Field of a Pressure Swirl Atomiser*, M.Phil. Dissertation, Department of Mechanical Engineering, UMIST, 2001.

[2] **Chinn, J.J., Cooper, D. and Yule, A.J.**, *Experimental Measurements and Computational Predictions of the Internal Flow in a Pressure Swirl Atomiser*, ILASS Europe-99, Toulouse, France, July 1999.

[3] **Cooper, D., and Yule A.J.**, *Waves on the Air Core/Liquid Interface of a Pressure Swirl Atomiser*, ILASS Europe-01, Zurich, Switzerland, September, 2001

[4] **Shaikh, S.**, *A Numerical and Experimental Study of a Swirl Flow Atomiser Using a High Viscosity Liquid*, MSc Dissertation, M.A.M.E., UMIST, 2003.