Eleventh International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
7-9 December 2015

A COMPARISON OF CFD SOFTWARE PACKAGES' ABILITY TO MODEL A SUBMERGED JET

Alasdair MACKENZIE[1*], A LOPEZ[1], K RITOS[1], M T STICKLAND[2], W M DEMPSTER[2]

[1] Weir Advanced Research Centre, University of Strathclyde, 99 George Street, Glasgow, SCOTLAND, G1 1RD

[2] Department of Mechanical and Aerospace Engineering, University of Strathclyde
*Corresponding author E-mail address: alasdair.mackenzie.100@strath.ac.uk

ABSTRACT

Particle erosion from slurry flow is a common problem in many industrial applications, including the mining and the oil and gas industry. Erosion modelling is a known complex problem, and consists of three equally important parts; fluid flow modelling, particulate flow modelling, and erosion modelling. It is the first of these, the fluid flow, which is analysed here. The paper compares three different computational fluid dynamics (CFD) software packages, ANSYS Fluent, Star-CCM+ and OpenFOAM, on their ability to model the fluid phase in the submerged jet impingement test. The computational results were verified by results from an experimental rig where the fluid flow was measured by particle image velocimetry (PIV). Despite the apparent simplicity of the jet impingement test, this paper highlights the difficulties of capturing the experimental results with computational methods.

NOMENCLATURE

CFD	computational fluid dynamics
fps	frames per second
GUI	graphical user interface
LDV	laser Doppler velocimetry
PIV	particle image velocimetry
r	radius
SG	specific gravity
SST	shear stress transport

INTRODUCTION

The study of wear due to particulate erosion is an established research field because of its complexity and the desire to increase the operational life of particle processing equipment, e.g. slurry pumps, crushers, cyclones etc. The current development of CFD techniques and their ability to accurately predict flow behaviour promises improved wear prediction since the particulate behaviour close to the wall can be predicted. A technique, recently explored by a number of authors, (Gnanavelu, Kapur, Neville, Flores, & Ghorbani, 2011; A. Mansouri et al., 2015) proposed that a wear model, i.e. a relationship between material removal and particulate behaviour (usually particle velocity and angle) can be developed using CFD. This methodology is usually explored using a fluid jet impact test and the authors show this as a promising approach to improving erosion prediction. However, only limited validation work on this aspect of the problem has been carried out. One frequently cited study is by Zhang *et al.* (Zhang, Reuterfors, McLaury, Shirazi, & Rybicki, 2007), where a submerged liquid jet impingement test, commonly used in erosion testing was studied using laser Doppler velocimetry (LDV), to measure velocity profiles and validate CFD simulations. Since then, the main focus of study has concentrated on the particulate behaviour and the erosion process. This paper revisits the fluid flow modelling, and investigates the effectiveness of a number of commercially available CFD codes to predict the submerged jets. ANSYS Fluent 15, Star-CCM+ 10.02 and OpenFOAM 2.3.x are compared to experimental data of a submerged jet impingement test constructed as part of this study.

Some surprising results emerge on the ability of available models to reproduce the experimental findings. The experimental set up is discussed and results compared.

SETUP

Experiment

The setup for the experiment can be seen in Figure 1 below. Particles are injected to the header tank with a mass concentration of <5%, and are well mixed. Due to the pressure caused by height difference, they follow the flow and impinge on the sample surface. Particles of diameter 20μm, with a specific gravity (SG) of 1 are used to track fluid velocities. The small particle diameter and equivalent density result in the particles following the flow (low Stokes number).

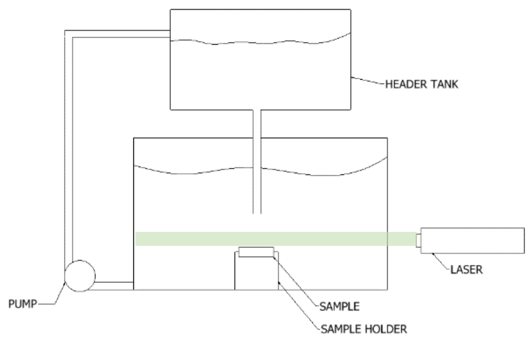

Figure 1: Experimental setup

The camera was set to 500 fps (frames per second), with the image focused on the light sheet and particles. The frame straddling technique was used to capture the particle movements, as standard cross correlation was not capable. The frame rate, combined with the laser pulses (set to 100μs), was fast enough to provide enough movement of the particles to carry out correlation. The particles need to move more than 6-7 pixels, but less than the size of the frame, to make post processing possible. FlowManager (Dantec Dynamics), a commercial program was used to analyse seed particle velocities and produce measured velocity fields. The pump recirculated the water back to the header tank, for further experiments. The flowrate was calculated by measuring the increase in weight of the tank over a known period of time. This was done multiple times to reduce measurement errors, and the average flowrate was found to be 3.2 kg/min, or 0.053kg/s.

Computational Fluid Dynamics

Two commercial CFD codes and an open source CFD code were examined: ANSYS Fluent 15, Star-CCM+ 10.02 and OpenFOAM 2.3.x. They were all set up with the same boundary conditions to replicate the experiment. Water had properties: density of 998.2kg/m^3, and viscosity of 0.001003 kg/ms.

The measured flowrate from the experimental test was used as the inlet boundary condition, and other boundary conditions can be seen in Figure 2. As shown in Figure 3, a refinement region around the area of interest was made, and a large enough volume was left around the impingement surface to ensure no recirculation took place. The nozzle was more than 10 diameters long, and the top surface was too far away so as not to interfere with the impingement zone. The geometry was drawn according to the experimental test as a half symmetric model, in order to reduce computational time.

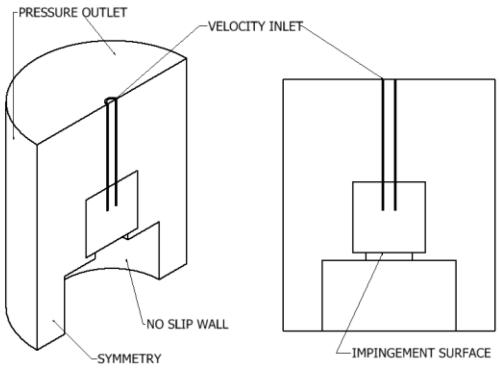

Figure 2: Geometry and boundary conditions used for CFD simulations

Initially the mesh for each code was generated by their own mesh generation application, and were shown to be mesh independent. However, the StarCCM mesh was chosen as the reference mesh, as it had the best quality and uses polyhedral cells that reduce the artificial diffusivity of the flow. The mesh had 3.2 million cells, and used 16 inflation layers near the walls with a growth ratio of 1.2, and total thickness of 0.6mm. There was also a refinement region around the impingement area, since this is where high gradients exist.

Each solver then ran using this mesh, and results are compared here. The y-plus value was checked on the nozzle wall and impingement surface, and it was kept below 1 for all simulations.

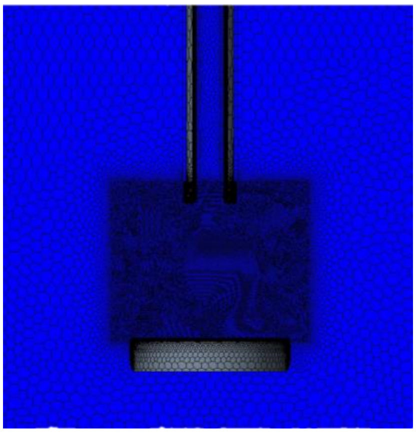

Figure 3: Mesh used for all simulations- showing refinement region

The standard k-epsilon turbulence model was used for all three software packages, with the scalable wall function used in fluent (since y*<11). To try and match experimental data better, an additional k-omega shear stress transport (SST) turbulence model was run in StarCCM, since various papers use this to model the jet impingement test (Mansouri, Shirazi, & Mclaury, 2015; Nguyen, Poh, & Zhang, 2014). Convergence was ensured, with each residual being less than 10^{-4} before results were taken.

RESULTS AND DISCUSSION

The PIV data were analysed on FlowManager using 100 images; thus giving 50 frames. A filter was applied to each frame to mask the area of interest, then a vector range of +3 to -3m/s was added to remove any noise. The 50 frames were then averaged, with the resulting vectors superimposed in Figure 4 below. The vectors can be seen, along with the plate, nozzle and light sheet. The laser sheet can be seen lighting up the surface of the sample, and the seeding particles throughout the liquid.

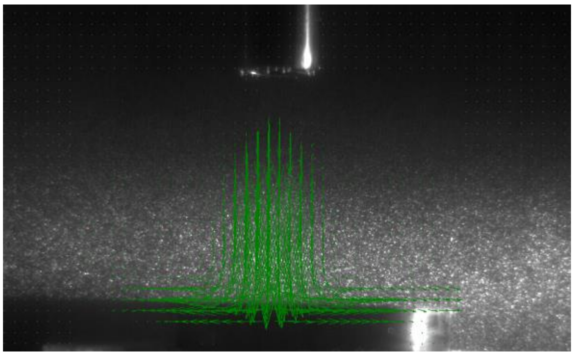

Figure 4: Average vector plot from FlowManager

A sample line 1mm above the plate was then drawn, and velocity vectors extracted. The same sample line was drawn in the CFD packages for comparison purposes. This height is suitable for particle data collection that can be used for erosion equations. Further away, the data wouldn't be relevant to surface impacts, and closer would yield lower velocities due to the boundary layer. Furthermore there is also the practical issue of sampling the flow with PIV at less than 1mm away from the surface.

Figures (5-7) show a comparison of the measured and computed velocities. The x-axis shows distance from the centre of the nozzle, going to the extents of the plate, 12.5mm in each direction. As the below figures indicate, there is general agreement between all of the k-epsilon solvers. All three CFD packages overpredict the velocity in the centre of the jet (less than r=2.5mm) and near the edge of the plate (more than r=7mm), whereas they all (apart from the k-omega SST) underpredict the velocity at the two peaks around r=5mm.

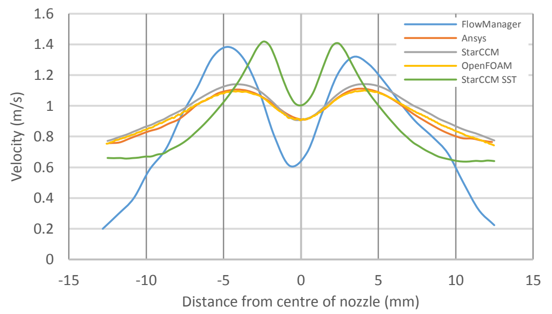

Figure 5: Velocity magnitude 1mm above plate

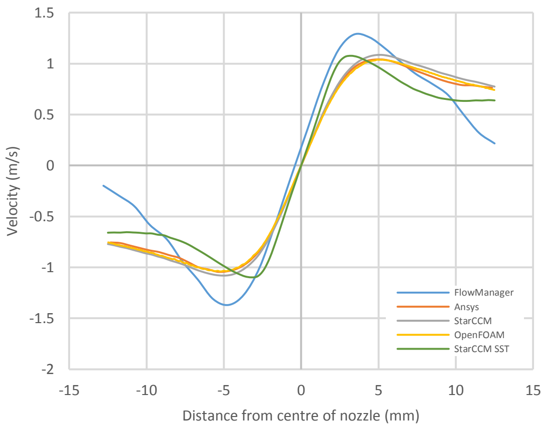

Figure 6: Radial velocity 1mm above plate

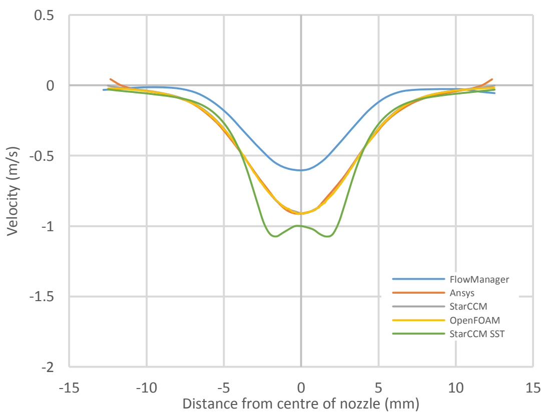

Figure 7: Axial velocity 1mm above plate

Fluent and OpenFOAM give very similar velocity profiles, whereas StarCCM is slightly different, mainly in the axial plane. This could be due to StarCCM's different wall modelling approach. Fluent and OpenFOAM both capture the position of the peak velocity magnitude, which is important for erosion modelling.

All software setups capture the radial components well, with the k-omega SST giving slightly better results near the outer edge of the plate.

The k-omega SST model captures the radial component of velocity very well, and also manages to capture the peak values of velocity magnitude. However it overpredicts the axial velocity more than any other model, leading to it not matching the PIV. All solvers struggle the most with the stagnation region in the axial direction.

Figure 8 shows a contour comparison of the PIV and Fluent velocity data. The jets have a similar shape, with the PIV having a stronger axial flow component in the centre than the CFD. This could suggest that the CFD over diffuses the flow, having a larger radial component further out from the centre of the jet (as seen in Figure 5, after 10mm from the centre of the jet).

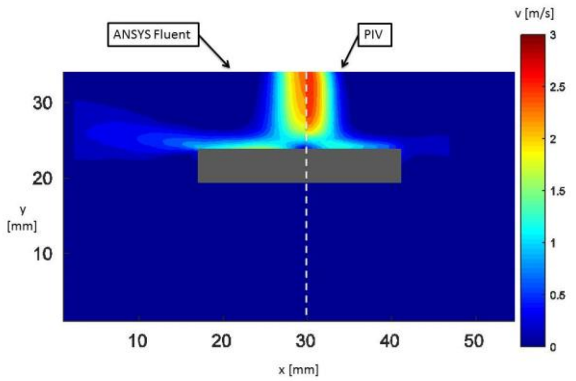

Figure 8: Velocity magnitude contour comparison

Discussion

The above results show that a 'simple case' is not as easily modelled as one might think. Current erosion prediction approaches require near wall accuracy of CFD models. The general trends are captured, however with all codes there is a discrepancy compared to the experimental velocity magnitudes by up to 40%. This is a significant difference, bearing in mind that the mass removal from particle impact is dependent on the velocity magnitude to the power of between 2 and 3, and impact angle. The CFD velocity graphs do follow the same shape as the experimental though, implying that erosion location should be accurate.

Figure 9 and Figure 10 taken from Zhang *et al*. show the results for radial and axial fluid velocity distribution of water released from the nozzle with average exit velocity of 12m/s. Their work was carried out using Fluent 6, and used the second order Reynolds stress turbulence model. The measured values of velocity were determined from point measurements using LDV at multiple positions on a grid. Although the turbulence model and boundary conditions are different, the results are similar in that the closer to the plate, the less able the CFD becomes to predict velocities accurately. The comparisons indicate that the experimentally measured values of axial velocity are 20-30% higher nearer the wall than predicted. While the radial velocities show even greater error, particularly at radial positions just beyond the nozzle radius. If a robust geometry independent erosion model is to be made, the near wall fluid modelling has to be improved.

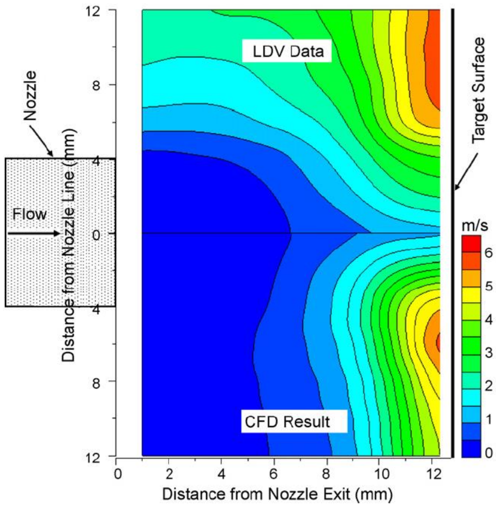

Figure 9: Radial fluid velocity, LDV data vs. CFD result (Zhang et al., 2007)

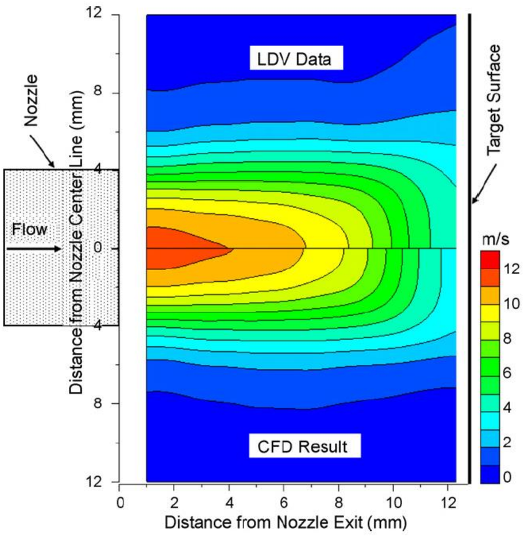

Figure 10: Axial fluid velocity, LDV data vs CFD result (Zhang et al., 2007)

Potential sources of error

The position of the laser relative to the nozzle was considered a source of error, however steps were taken to ensure this was minimised. The laser, and thus the light sheet, was first aligned with the centre of the jet, and then moved to 1mm either side and experiments repeated to determine the sensitivity to position. The results were analysed, and the position that gave the highest axial velocity was chosen: being an indication of the centre of the jet.

CONCLUDING REMARKS

A comparison between two commercial CFD codes (ANSYS Fluent, STARCCM) and an open source CFD code (OpenFOAM) with the velocity fields experimentally measured for a submerged jet, impacting on a plate indicate that:

1. Using two equation turbulence models, the CFD codes could not accurately predict the impacting jet flow field, 1mm above the plate.

2. Stagnation regions are still difficult to model.

3. Further work is required to see if particle trajectories predicted by CFD are affected by this inability to model near wall flows. If so, the jet impingement test on a flat plate may not be the best way to implement erosion modelling.

Noting that current approaches to erosion modelling have three requirements, fluid flow, particle flow, and erosion modelling, the first of these is often assumed. This paper demonstrates that errors in the fluid flow's magnitude predictions could be sizable, however the general shape is captured.

REFERENCES

Gnanavelu, A., Kapur, N., Neville, A., Flores, J. F., & Ghorbani, N. (2011). A numerical investigation of a geometry independent integrated method to predict erosion rates in slurry erosion. *Wear*, *271*(5-6), 712–719. doi:10.1016/j.wear.2010.12.040

Mansouri, A., Arabnejad, H., Shirazi, S. A., & McLaury, B. S. (2015). A combined CFD/experimental methodology for erosion prediction. *Wear*, *332-333*, 1090–1097. doi:10.1016/j.wear.2014.11.025

Mansouri, A., Shirazi, S. A., & Mclaury, B. S. (2015). Experimental and numerical investigation of the effect of viscosity and particle size on erosion damage caused by solid particles. *ASME*, 1–10 doi:10.1115/FEDSM2014-21613

Nguyen, V. B., Poh, H. J., & Zhang, Y.-W. (2014). Predicting shot peening coverage using multiphase computational fluid dynamics simulations. *Powder Technology*, *256*, 100–112. doi:10.1016/j.powtec.2014.01.097

Zhang, Y., Reuterfors, E. P., McLaury, B. S., Shirazi, S. A., & Rybicki, E. F. (2007). Comparison of computed and measured particle velocities and erosion in water and air flows. *Wear*, *263*(1-6), 330–338. doi:10.1016/j.wear.2006.12.048

