Optimization of abrasive waterjet nozzle design for precision and reduced wear using compressible multiphase CFD modelling

C. Narayanan, D. Caviezel, D. Lakehal
ASCOMP AG
Technoparkstrasse 1, 8005 Zurich, Switzerland
narayanan@ascomp.ch

Abstract

The flow inside an Abrasive waterjet (AWJ) nozzle is extremely complex due to the interaction of the high speed water jet with the entrained air and the abrasive particles. The resulting three-phase flow inside the nozzle is not understood well primarily due to the difficulty in making measurements (or even viewing) inside the small nozzle. The nozzle geometry is typically optimized by trial and error by looking at the characteristics of the abraded workpiece. Three dimensional computations could be the ideal candidate to make better designs for the nozzle. Compressible multiphase models can be used to simulate many of the flow features in the nozzle such as the hydraulic flip caused by cavitation just aft of the orifice, the mixing and spreading of the water jet, the acceleration of abrasive particles on contacting the water jet and the collisions of the abrasive particles with the walls. In addition the collision statistics can be used to estimate erosion rates of the nozzle material. The computational multi-fluid dynamics software TransAT is used in this study. Results from CFD simulations of a complete nozzle/cutting-head is presented and compared to experimental data in terms of predicted particle exit velocity for different sized particles for different operating pressures and abrasive flow rates. The collision statistics, accumulated during the simulation, is used to estimate the location of significant wear. The potential of using CFD to optimize nozzle design is discussed.

1 INTRODUCTION

Erosion of material by solid particles accelerated by a high-speed water jet is the basic process of abrasive water jet (AWJ) machining. A typical commercial AWJ system consists of a pump, a mixing and acceleration section, a positioning system, and a catcher. Depending on the method of dosage of abrasive particles into the water jet, AWJs can be classified as injection jets or suspension jets. For practical cutting applications, injection jets are more commonly used, wherein an AWJ is formed by accelerating small solid particles (typically Garnet) through contact with a high-speed water jet (see Figure 1).

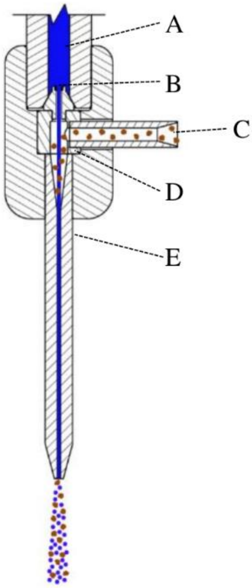

Figure 1 Sketch of a typical abrasive water jet cutting head.

The high-speed water jet, in turn, is formed in an orifice placed on top of the mixing and acceleration head. The solid particles are dragged into the mixing chamber through a separate inlet by the low pressure created by the water jet in the mixing chamber. Mixing between the solid particles, water jet and air takes place in the mixing chamber, and the acceleration process occurs in the focusing tube. After the mixing and acceleration process, a high speed three-phase mixture leaves the tube at velocities of several hundred meters per second.

Abrasive water jet machining is a rapidly growing technology capable of processing a variety of materials. Since AWJ machining relies on erosion by fine abrasive particles, mechanical loads exerted on the workpiece are localised, and the flow of water leads to very low thermal stress on the workpiece. Further expanding the area of application of the abrasive jet for example as a milling tool, or to address the issue of increasing the precision of the cut, to control the depth of cut, or to reduce wear of the cutting is still a challenge. Since there are several parameters that affect the material removal rate and profile, such as pump pressure, jet feed rate, abrasive mass flow rate, stand-off distance, and abrasive breakage inside the cutting head, it becomes difficult to obtain the desired local material removal. The process is also subject to fluctuations in pressure and abrasive mass flow rate. Therefore, one of the main factors of the success of this process is an accurate understanding of the jet properties at the exit of the cutting head. Several methods have been used to characterise the mixing and acceleration process experimentally. However, most of the experiments were unable to accurately measure the velocities of abrasive particles as they leave the focussing tube, due to the complex topology of the three-phase mixture and the small size of the mixing chamber. Measurement of particle exit velocities was carried out through the use of ultra-fast X-ray velocity (Balz, Mokso, Narayanan, Weiss, & Heiniger, 2013) – however, this is a very expensive undertaking and is not widely available. In spite of that, making measurements inside the cutting head to characterize the mixing and acceleration of the abrasives was not possible. However, a comprehensive set of experimental data was made available for the validation of phenomenological models as well as for validating advanced computational fluid dynamics (CFD) based models through this experiment.

Knowledge of the particle velocity and the energy distribution across the abrasive jet is of great importance because the materials cut or eroded due to wear, which depends on the kinetic energy of each particle. Several studies present phenomenological or lower order models where the particle exit velocities have been predicted (Narayanan, Balz, Weiss, & Heiniger, 2013; Momber, 2001; Tazibt, Parsy, & Abriak, 1996). In particular the model of Narayanan et al. (2013) takes into account the influence of the mixing tube diameter and length, orifice diameter, abrasive feed tube diameter and length, mixing chamber diameter and length, hydraulic pressure, abrasive flow rate, density and viscosity of both air and water, density of abrasive, discharge coefficient of the nozzle, ambient pressure and temperature, and abrasive size distribution. Due to that fact that the influence of geometry and flow parameters are taken into account, this one-dimensional three-phase model of the flow in the cutting head has been used to optimize the abrasive kinetic energy for a given set of operating conditions through the use of genetic algorithms (Schwartzentruber, Narayanan, Liu, & Papini, 2016).

Lower-order modelling has limited applicability in design, as they cannot provide detailed insight into the complex flow in the mixing chamber. Computational fluid dynamics (CFD) therefore appears to be an attractive method to further improve the performance characteristics of a cutting head including the possibility to reduce wear. Advanced three-dimensional CFD-based modelling has also been attempted by Prisco and D’Onofrio (Prisco & D'Onofrio, 2008). However they simulated only the two-phase water and air flow inside the cutting head. Ahmed et al. (Ahmed, Siores, Naser, & Chen, 2001) presented a CFD model where all the three phases are modelled as Eulerian phases. However, very limited validation was carried out.

The nozzle geometry is typically optimized by trial and error by looking at the characteristics of the abraded workpiece. Three dimensional computations could be the ideal candidate to make better designs for the nozzle. Compressible multiphase models can be used to simulate many of the flow features in the nozzle such as the hydraulic flip caused by cavitation just aft of the orifice, the mixing and spreading of the water jet, the acceleration of abrasive particles on contacting the water jet and the collisions of the abrasive particles with the walls. In addition, the collision statistics can be used to estimate erosion patterns on the nozzle surface.

This study presents results from 3D CFD simulations of a complete nozzle/cutting-head and compares it to experimental data in terms of predicted particle exit velocity for different sized particles for different operating pressures and abrasive flow rates. The collision statistics, accumulated during the simulation, is used to estimate the location of significant wear. The potential of using CFD to optimize nozzle design is discussed.

2 MATHEMATICAL MODEL

The mathematical model of the three-phase flow inside the AWJ cutting head primarily consists of modelling the two-phase air-water flow by a compressible multiphase mixture model based on the phase-averaged approach. The multiphase mixture model has to be augmented by a cavitation model to capture the vapour production and hydraulic flip at the orifice resulting in the creation of a stable high-speed water jet. The abrasive particles motion and interaction with the two-phase fluid flow is achieved by employing the Lagrangian particle tracking approach.

CFD modelling of the abrasive particle motion is complicated by the fact that the abrasive particles are not small in comparison to the focussing tube diameter. A full moving object model coupled with multiphase flow modelling of the air, water mixture would be required, which is beyond the state-of-the-art of CFD simulations. Therefore, as a pragmatic approach the Lagrangian particle tracking model is used wherein the motion of each particle is modelled through the use of average phenomenological forces such as drag, buoyancy, lift, etc. acting on individual particles.

The modelling of turbulence is a complex issue for multiphase flows, and in this study the Reynolds Averaged Navier-Stokes (RANS) model is used such that it is directly applied to the fluid (air-water) mixture. CFD can also be used to predict wear in the nozzle and the Lagrangian particle tracking model is ideally suited for this purpose. The details of each of the above modelling approaches is given below.

2.1 Compressible Multiphase Mixture Model

The compressible multiphase flow model is described in detail in the paper by Labois and Narayanan (Labois & Narayanan, 2016). The model includes solving for the volume fraction of each phase, the mixture momentum, and the mixture temperature equations. The mass transfer between the liquid and vapour phases is modelled using the cavitation model (applicable in the near orifice zone if the interest is to capture the hydraulic flip in detail). For all the cases reported here, the RANS turbulence model was used along with turbulent dispersion between the phases.

2.2 Cavitation model

The high pressure water flow through a sharp orifice leads to rapid vapour generation through cavitation and at high enough flow rates, results in a hydraulic flip. This phenomenon is an essential ingredient of abrasive water jet cutting since it increases the precision of the cut by focussing the high speed water jet. The vapour production along with the further mixing of entrained air, water and abrasives can be simulated using multiphase CFD through the use of cavitation models. The cavitation model proposed by Yuan et al. (Yuan, Sauer, & Schnerr, 2001), referred to as the Sauer model, is used in this study.

2.2.1 Sauer cavitation model

In the Sauer cavitation model, the evaporation and condensation rates are given as below,

$$\dot{m} = \frac{\rho_v \rho_l}{\rho_m} \alpha(1-\alpha) \frac{3}{R_b} \left[\frac{2}{3} \frac{(p_{sat} - p)}{\rho_l}\right]^{\frac{1}{2}} \text{ for } p < p_{sat},$$

and,

$$\dot{m} = -\frac{\rho_v \rho_l}{\rho_m} \alpha(1-\alpha) \frac{3}{R_b} \left[\frac{2}{3} \frac{(p - p_{sat})}{\rho_l}\right]^{\frac{1}{2}} \text{ for } p > p_{sat},$$

where ρ_v, ρ_l, are vapour and liquid densities, ρ_m is the mixture density, and α is the vapour-phase volume fraction.

$$R_b = \left(\frac{\alpha}{1-\alpha} \frac{3}{4\pi n_b}\right)^{\frac{1}{2}}$$

where R_b is the radius of the vapour bubble as a function of the volume fraction of vapour and the nucleate density (n_b). If the local pressure is below the saturation pressure (p_{sat}) then the liquid evaporates and vice versa. The only tuneable parameter in this model is the nucleate density (n_b). This constant has to be typically tuned for each class of problems.

2.3 Lagrangian Particle Tracking

In the Lagrangian particle tracking approach, each particle is treated as a point particle; although the volume of fluid displaced by the particles can be taken into account. The position and velocity of the particle is tracked using forces such as Drag, Lift, etc. The drag interaction between the particle and fluid is accounted for through the so called two-way coupling approach (Narayanan, Lakehal, & Yadigaroglu, 2002).

2.4 Wear modelling

The CFD simulation records all collisions of the abrasives with the cutting head walls which includes the incident velocity vector and the location of the collision. After the simulation, the erosion rates are estimated by associating each collision recorded to a particular region (triangle) of the triangulated geometry of the desired object. Several erosion models could be used; in this study the DNV model (DNVGL, 2015) is employed. The DNV model (in fact most impact erosion models) rely on a similar model template as shown below. The erosion rate is given as,

$$\dot{E}_{DNV} = \dot{m}_p K U_p^n F(\alpha)$$

Where the erosion rate is proportional to the mass flow rate of colliding abrasives, the incident velocity and the angle of incidence (α). K and n are model constants that depend on the materials being cut. The constants have to be obtained through calibration with experiments as they do not directly represent a particular material property. The velocity exponent (n) also needs calibration through experiments. The exponent has a strong impact on the erosion rate estimates. Due to the above reasons, erosion estimates are generally qualitative unless specifically validated with experimental data. However, the erosion pattern is expected to be more representative of reality. For this demonstration study the model constants are given in Table 1. The constants have been chosen from (DNVGL, 2015) for stainless steel.

Table 1: Model constants used for the DNV erosion model.

Constant	Value
K	2.0e-9
n	2.6

2.5 TransAT

The CFD code TransAT© (ASCOMP, 2016) used for this study, is a finite-volume code solving the Navier-Stokes fluid-flow equations. Pressure-velocity coupling is solved using a pressure-based scheme for both incompressible and compressible flows. Grid generation and geometry representation is achieved on Cartesian grids using the Immersed Surfaces method. Both steady state and transient time marching schemes could be used along with 2nd-order schemes for convection and diffusion terms. TransAT is parallelized using MPI and domain decomposition on distributed memory clusters. Turbulence effect can be modelled using Reynolds Averaged Navier-Stokes (RANS) as well as scale resolving methods, such as Very Large Eddy Simulation (VLES) and LES for modelling turbulence effects.

3 CAVITATION AT THE ORIFICE

The production of vapour resulting in hydraulic flip at the orifice results in the formation of a stable high-speed jet. The amount of vapour and air entrained into the cutting head is an important aspect related to the performance of the nozzle, although the influence of the gaseous phase is known to be significantly lower than water. The AWJ cutting head under typical operating conditions has a very high volume fraction of air as compared to water and abrasive particles. The influence of the gas phase has not been very carefully characterized until now – including the possible formation of an exit shock and its influence on the abrasive particle velocity and velocity fluctuations. In the next section the application of the cavitation model to the flow of high speed liquid through a sharp orifice is presented.

3.1.1 Sharp orifice of Nurick (1976)

An example of the application of the multiphase cavitation model is presented in this section. The test case is based on the experiments by Nurick (Nurick, 1976) on the cavitation in sharp and rounded orifices. This case is presented in detail by Singhal (Singhal, Athavale, Li, & Jiang, 2002). The cavitation model of Sauer (Yuan, Sauer, & Schnerr, 2001) is used in this study. A sketch of the geometry for the simulation is shown in Figure 1. High pressure water enters the inlet and undergoes cavitation due to the sudden change in flow area at the orifice. The following fluid properties were used: Water temperature 300K, water density 1000 kg/m^3, vapour density 0.02558 kg/m^3, saturation pressure 3540 Pa, and surface tension 0.0717 N/m. The flow was considered to be incompressible. The following conditions are simulated:

- P_{exit} = 0.95 bar, $P_{0,inlet}$ = 1.9 bar - 2500 bar
- R/r = 2.88, L/r = 10
- Model parameter: nucleate density = 10^7

The simulation was run with and without cavitation model to see the effect of the cavitation model. Note that for the real flow inside a cutting head, a three-phase flow consisting of water, vapour and air can also be simulated.

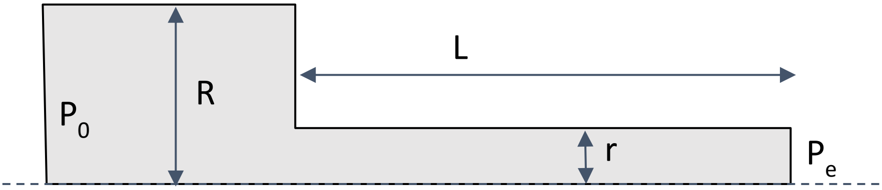

Figure 2 Sketch of the sharp orifice experiment of Nurick (1976).

The stream-wise velocity contours for the high pressure inlet cases (1000 and 2500 bar) are shown in Figure 3, where the black line shows the vapour-liquid interface (contour of volume fraction = 0.5). It can be seen that a stable jet of water is obtained in the simulations for high pressure ratios.

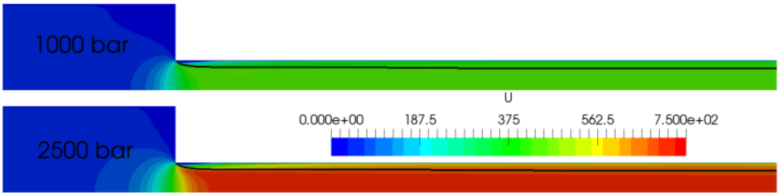

Figure 3 Hydraulic flip in sharp edged orifice simulated with the cavitation model

The predicted discharge coefficient is shown in Figure 4. The discharge coefficient is compared to the correlation developed by Nurick et al. (Nurick, 1976) and a very good match is obtained. In the absence of a cavitation model, the discharge coefficient remains constant. The tests were performed on two grids with the fine grids providing better match to the correlation.

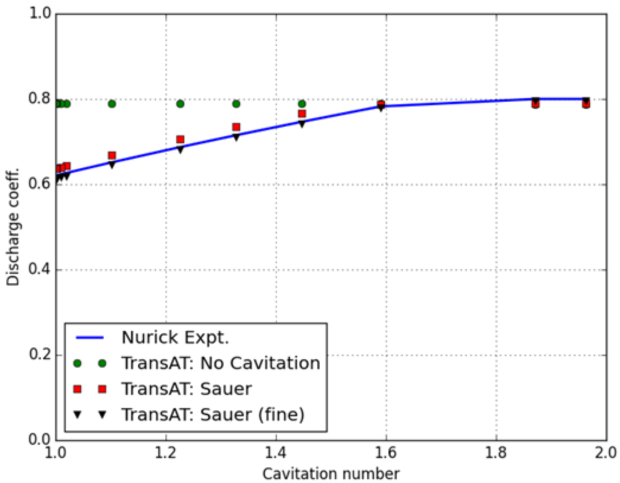

Figure 4 Predicted discharge coefficient for the sharp edged orifice case of Nurick (Nurick, 1976) and comparison to experimental correlation.

4 PERFORMANCE OF CUTTING HEAD AND WEAR

Application of multiphase CFD modelling to the flow inside a complete cutting head is presented in this section.

4.1 Validation data

Two experimental conditions are chosen in this study to be simulated using multiphase CFD, from the set of experimental data made available by Balz et al. (Balz, Mokso, Narayanan, Weiss, & Heiniger, 2013). Such an attempt has been missing due to the lack of quality measurements of particle exit velocities. The particle exit velocity of a large number of particles was measured in this study. Solid abrasive particles were identified on the acquired images with particle tracking velocimetry, which was processed to evaluate the abrasive particles spatial position and velocity. Water pressure and abrasive mass flow of the AWJ were varied to gather versatile information about the abrasive particles velocity and particle distributions.

The abrasive particles size measurement showed a Sauter mean diameter of 266μm, a minimum diameter of 184 μm and a maximum diameter of 421μm. The geometric parameters of the cutting head are given in Table 2.

Table 2 Dimensions of the cutting head for which measurements were made.

Cutting head geometry	Dimension
Orifice diameter	0.28 mm
Focussing tube diameter	0.8 mm
Focussing tube length	76 mm
Mixing chamber diameter	4.5 mm
Mixing chamber length	15 mm

4.2 Simulation setup and results

Simulations of the two experimental conditions presented below were performed. The computational grid consisted of mesh sizes of 300 μm along the jet direction and 40 μm in the cross directions resulting in a total of approximately 500,000 control volumes. The simulation was run for a period of 3 ms with a time step of approximately 0.1-0.3 μs. The cavitation model was not employed in these since the interest was in capturing the particle exit velocities to compare to the experimental data and to estimate the locations of highest wear in the cutting head. Particles in the diameter range 70-200 μm were simulated and were introduced through a tilted inlet on one side of the cutting head. Somewhat smaller particles were simulated as compared to the experiments, because it is known that some amount of breakage of the particles occurs inside the mixing tube.

The particles leaving the focusing tube were analysed for the exit velocities. The particle velocities leaving the focusing tube for the larger particle sizes simulated are summarized in Table 3. The range of velocities predicted by the simulations is within the range of experimentally measured values.

Table 3 Comparison of particle exit velocity with experimental results.

	Experiment 3	**Experiment 4**
Pressure	1430 bar	910 bar
Abrasive flow rate	250 g/min	211 g/min
Jet velocity	527 m/s	422 m/s
Particle exit velocity (Expt.)	296.82 ± 34.2 m/s	251.0 ± 50.2 m/s
Particle exit velocity (Sim.)	200-370 m/s	200-300 m/s

Figure 5 shows a snapshot of the flow inside the cutting head and Figure 6 shows a view close to the exit of the focussing tube. The blue surface represents the volume fraction of 0.5 and therefore, the liquid near the workpiece looks isolated. In reality, there is strong mixing and the local maximum volume fraction is lower in the surrounding areas.

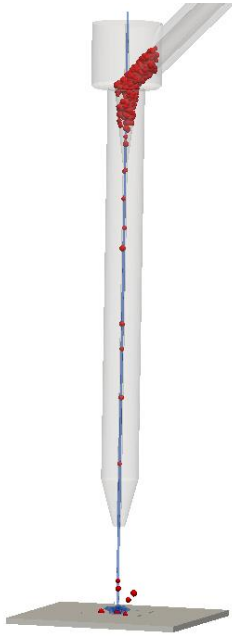

Figure 5 Snapshots of Experiment 4 simulation in 3D. The simulation was run for several flow through times. Full view of the computational domain. The particles are shown larger than their actual size.

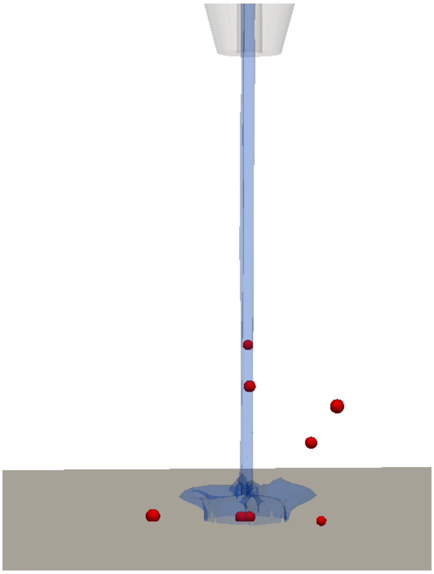

Figure 6 Snapshots of Expt. 4 simulation in 3D. Near the focusing tube exit.

The particle velocity depends strongly on the particle diameter as shown by the one-dimensional model (Narayanan, Balz, Weiss, & Heiniger, 2013). For Expt. 4, the small particles ($70\mu m$) have velocities between 125-350m/s, whereas the large particles ($150\mu m$) have velocities between 200-300m/s. The velocities of the larger particles seem to be closer to the experimentally measured values. For Expt. 3, small particles have velocities between 200-450m/s, and large particles (>= $150\mu m$) have velocities between 200-370m/s. The particles in each size class also exhibit a large variation in exit velocity, primarily due to different interaction times and locations with the water jet. The results also show that the scatter in the particle velocity measured in the experiments is not just due to different particle sizes but also due to the path followed by an individual particle in the focusing tube (shown in Figure 7).

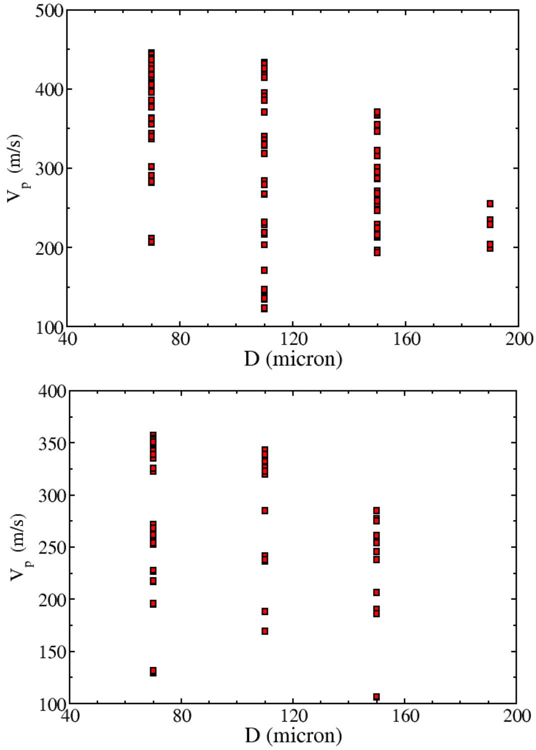

Figure 7 Particle exit velocities from simulation (Top) Expt. 3 (Bottom) Expt. 4.

4.3 Wear estimates

The estimate of wear has been obtained by analysing the collision data from the simulations, and calculating the erosion rates based on the DNV model. The erosion pattern is presented in Figure 8 below for Experiments 3 and 4. However, it shows that for the case where the inlet of particles is from the left, wear is higher on the right side of the mixing cone. Also, the exit tip of the focussing tube also undergoes strong wear. It appears that more particle statistics must be accumulated for reducing the statistical error, for better estimates inside the focussing tube. Even through the erosion rate estimates are not quantitatively correct (since we use stainless steel properties for the cutting head material), it is clear that CFD can provide very useful information to compare different designs so as to reduce wear.

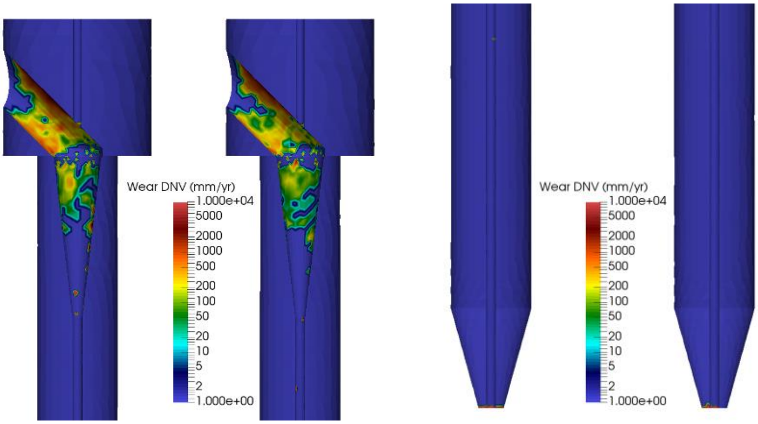

Figure 8 Wear pattern in mm/yr. estimated using the DNV erosion model (left) Expt. 3, (right) Expt. 4.

5 CONCLUSIONS

The modelling of three-phase flow with air, water and large particles at compressible conditions could be very useful to improve the design of the cutting head both in terms of improving efficiency and precision of cut, and for reducing wear. CFD can be used to address all aspects of the flow inside the AWJ cutting head starting from the hydraulic flip at the orifice to abrasive interaction and acceleration to wear.

REFERENCES

Ahmed, D. H., Siores, E., Naser, J., & Chen, F. L. (2001). Numerical simulation of abrasive water jet for different taper inlet angles.,. *14th Australasian Fluid Mechanics Conference*, (pp. 645-648). Australia.

ASCOMP. (2016). *Products > TransAT CFD Suite > TransAT Multiphase*. Retrieved from http://ascomp.ch/products/transat-suite/transat-multiphase/

Balz, R., Mokso, R., Narayanan, C., Weiss, D. A., & Heiniger, K. C. (2013). Ultra-fast X-ray particle velocimetry measurements within an abrasive water jet. *Experiments in fluids, 54*(3), 1-13.

DNVGL. (2015). *Recommended Practice, Managing sand production and erosion.* DNV GL.

Labois, M., & Narayanan, C. (2016). Non-conservative pressure-based compressible formulation for multiphase flows with heat and mass transfer. *ICMF-2016 – 9th International Conference on Multiphase Flow.* Firenze, Italy.

Momber, A. W. (2001). Energy transfer during the mixing of air and solid particles into a high-speed waterjet: an impact-force study. *Experimental Thermal and Fluid Science, 25*(1), 31-41.

Narayanan, C., Balz, R., Weiss, D. A., & Heiniger, K. C. (2013). Modelling of abrasive particle energy in water jet machining. Journal of Materials Processing Technology. *Journal of Materials Processing Technology, 213*(12), 2201-2210.

Narayanan, C., Lakehal, D., & Yadigaroglu, G. (2002). Linear stability analysis of particle-laden mixing layers using Lagrangian particle tracking. *Powder Technology, 125*(2), 122-130.

Nurick, W. H. (1976). Orifice cavitation and its effect on spray mixing. *Journal of Fluids Engineering*, 681-687.

Prisco, U., & D'Onofrio, M. C. (2008). Three-dimensional CFD simulation of two-phase flow inside the abrasive water jet cutting head. *International Journal for Computational Methods in Engineering Science and Mechanics, 9*(5), 300-319.

Schwartzentruber, J., Narayanan, C., Liu, H. T., & Papini, M. (2016). Optimized abrasive waterjet nozzle design using genetic algorithms. *23rd International Conference on Water Jetting.* Seattle: BHR Group.

Singhal, A. K., Athavale, M. M., Li, H., & Jiang, Y. (2002). Singhal, A. K., Athavale, M. M., Li, H., & Jiang, Y. (2002). Mathematical basis and validation of the full cavitation model. *Journal of fluids engineering, 124*(3), 617-624.

Tazibt, A., Parsy, F., & Abriak, N. (1996). Theoretical analysis of the particle acceleration process in abrasive water jet cutting. *Computational Materials Science, 5*(1), 243-254.

Yuan, W., Sauer, J., & Schnerr, G. H. (2001). Modeling and computation of unsteady cavitation flows in injection nozzles. *Mécanique & industries, 2*(5), 383-394.