Turbulent solid-liquid flow through the nozzle of premixed abrasive water jet cutting systems

J Ye and R Kovacevic*

Department of Mechanical Engineering, Southern Methodist University, Dallas, Texas, USA

Abstract: A computational fluid dynamics analysis has been conducted for the steady state, turbulent, solid–liquid flow through nozzles used in premixed abrasive water jet cutting systems. The development of a theoretical approach to the evaluation of turbulent flow and particle dynamic properties in the nozzles is attractive because of the difficulties associated with direct measurements in nozzles of high flow speed and small dimension. Axisymmetric simulations have been performed with the commercial code FIDAP, using the standard $k-\varepsilon$ turbulence model. One-way coupling was considered in the simulations, which means that the effect of the presence of the dispersed solid phase on the dynamics of the liquid phase was neglected. The velocities and trajectories of solid particles were predicted. The effects of nozzle geometry on particle dynamic properties were studied. The predictions have been compared with available experimental and theoretical results published by other investigators. This modelling technique will assist in the nozzle design of premixed abrasive water jet systems and the prediction of jet cutting performance.

Keywords: abrasive water jet, computational fluid dynamics, solid-liquid turbulent flow

NOTATION

C_1, C_2	turbulence model constants
$C_{\rm D}$	drag coefficient
C_{μ}	turbulence model constant
d, D	focus tube diameter of nozzle
d_0	inlet diameter of nozzle
$d_{\rm p}$	particle diameter
f^{r}	drag force correction factor
k	turbulent kinetic energy
L	focus tube length of nozzle
L_0	nozzle length
Р	mean pressure
r	radial coordinate direction
Re	nozzle Reynolds number = $U_0 d_0 / v$
Re_{p}	particle Reynolds number = $ U - U_p d_p / v$
U^{\uparrow}	fluid velocity
(u, v)	z component and r component of particle
	velocity
u_r^2	radial stress component

 u_z^2 axial stress component

The MS was received on 20 October 1997 and was accepted after revision for publication on 29 September 1998.

 $u_z u_r$ shear stress component Е dissipation of turbulent kinetic energy θ tapered inlet angle of nozzle fluid dynamic viscosity μ fluid turbulent viscosity $\mu_{\rm t}$ fluid kinematic viscosity ν fluid density ρ $\sigma_k, \sigma_\varepsilon$ Prandtl numbers for k and ε particle response time

Subscripts

f fluid m maximum p particle phase r radial coordinate direction z axial coordinate direction

1 INTRODUCTION

The method of direct injection of abrasive slurry is relatively new in abrasive water jet cutting. In the direct injection system, also called the DIAjet system, a premixed abrasive slurry is pumped through a nozzle to form the cutting jet. Comparison between this method

^{*}Corresponding author: Department of Mechanical Engineering, Southern Methodist University, 3160 SMU Blvd, Dallas, TX 75205, USA.

and the conventional entrainment system shows that the premixed abrasive water jet system has better mixing efficiency, compact nozzle design and lower working pressure. Generally, it is more efficient for transferring fluid energy to particles and can produce greater power density for impacting particles and accordingly higher material removal rates [1]. The disadvantages of this system are severe wear in the nozzles and hardware complications in abrasive suspension handling and pumping.

In order to predict the jet cutting performance and wear mechanism in the nozzle, it is essential to know the particle velocities and trajectories in the DIAjet nozzle. Computational fluid dynamics (CFD) analysis is particularly useful because the velocity is so high and the dimensions are so small in the DIAjet nozzle that the direct measurement of particle velocities and the visualization of particle trajectories are very difficult. CFD analysis can provide not only information about the turbulent fluid flow but also information about abrasive particles in the nozzle.

This paper addresses the problem of two-phase (solid–liquid) turbulent flow in DIAjet nozzles and reports the results of a CFD analysis of the problem. The abrasive particle velocities and trajectories are calculated. Nozzles with different lengths and tapered inlet angles were evaluated for optimization of nozzle parameters.

2 DIAjet SYSTEMS AND NOZZLE PERFORMANCE

The earliest work relating to DIAjet was presented by Fairhurst *et al.* [2] who proposed the application of direct injection of abrasive materials in the water jet and built the first low-pressure laboratory system. Since 1986, DIAjets have been used in applications such as metal and rock cutting [3-5], nuclear and underwater decommissioning [6, 7], etc.

The flow in a DIAjet nozzle is a turbulent, highspeed, solid-liquid two-phase flow. This complex problem has not been sufficiently investigated. The influence of different parameters is not clearly understood at the present time. Hashish [1] made a comparative evaluation of DIAjets and conventional entrainment systems. By a simplified analysis, he showed that DIAjets are more efficient than entrainment abrasive water jets (AWJs) by a factor of 2 at an abrasive loading ratio of 1. Also, DIAjets are potentially over 20 times more dense in terms of kinetic power delivery to the workpieces. Laurinat *et al.* [8] presented test results concerning the influence of the abrasive flowrate as well as the nozzle design on the cutting efficiency. They qualitatively discussed the effects of nozzle geometry on particle acceleration and particle velocity. Guo *et al.* [9] described a onedimensional analysis of the accelerating process of particles in a DIAjet system. They showed a distance function that connects the particle moving distance and the velocity ratio U_p/U_f . Their experimental results stressed the importance of the particle velocity in cutting performance, while the flowrate of water has little effect on the cutting results.

In one-dimensional analysis [9-11] the fluid velocity is assumed to be uniform in the nozzle cross-section and the turbulent properties of the flow are not considered in the analysis. However, the fluid velocity in a jet nozzle is far from a uniform distribution and the flow turbulence plays an important role in the transportation and acceleration of abrasive particles. Obviously, the one-dimensional model is inadequate for analysis of the complex turbulent flow in jet nozzles. In order to predict the flow and particle properties in jet nozzles, a more rigorous analysis is needed.

3 FINITE ELEMENT MODELLING

The task of this work is to use CFD analysis for predicting the fluid and particle dynamic properties and optimizing the geometrical parameters of DIAjet nozzles. The FIDAP code version 7.05 is used for mathematical modelling of the turbulent solid–liquid flow in the DIAjet. Figure 1 shows a schematic of a DIAjet nozzle. In order to avoid non-essential complexities the assumption of one-way coupling between phases is used. This means that the dynamics of the fluid phase drives the motion of the abrasive particles, while the presence of the abrasive particles has no effect on the dynamics of the fluid phase.

The fluid flow field is solved in the first step. The turbulent flow was modelled using the $k-\varepsilon$ (turbulent kinetic energy and energy dissipation) method, which has been widely used for high Reynolds number applications. The governing equations for steady, incompressible, turbulent, constant properties, axisymmetric flow are as follows:

$$\frac{\partial U_z}{\partial z} + \frac{\partial U_r}{\partial r} + \frac{U_r}{r} = 0 \tag{1}$$

$$\rho \left(U_z \frac{\partial U_z}{\partial z} + U_r \frac{\partial U_z}{\partial r} \right) \\
= -\frac{\partial P}{\partial z} + \mu \left(\frac{\partial^2 U_z}{\partial z^2} + \frac{\partial^2 U_z}{\partial r^2} + \frac{1}{r} \frac{\partial U_z}{\partial r} \right) \\
- \rho \left[\frac{\partial \overline{u_z^2}}{\partial z} + \frac{1}{r} \frac{\partial}{\partial r} (r \overline{u_z u_r}) \right]$$
(2)

B07597 © IMechE 1999



Fig. 1 Schematic of a DIAjet nozzle



Fig. 2 Finite element mesh for the nozzle

$$\rho\left(U_{z}\frac{\partial U_{r}}{\partial z}+U_{r}\frac{\partial U_{r}}{\partial r}\right)$$

$$=-\frac{\partial P}{\partial r}+\mu\left(\frac{\partial^{2} U_{r}}{\partial r^{2}}+\frac{\partial^{2} U_{r}}{\partial r^{2}}+\frac{1}{r}\frac{\partial U_{r}}{\partial r}\right)$$

$$-\rho\left[\frac{\partial \overline{u_{z}u_{r}}}{\partial z}+\frac{1}{r}\frac{\partial}{\partial r}(r\overline{u_{r}^{2}})\right]$$
(3)

The Reynolds stresses are given by

$$-\rho \overline{u_r^2} = 2\mu_t \frac{\partial U_r}{\partial r} - \frac{2}{3}\rho k$$
$$-\rho \overline{u_z u_r} = \mu_t \left(\frac{\partial U_r}{\partial z} + \frac{\partial U_z}{\partial r}\right)$$
$$-\rho \overline{u_z^2} = 2\mu_t \frac{\partial U_z}{\partial z} - \frac{2}{3}\rho k$$
(4)

To describe the turbulent transport, the two-equation $k-\varepsilon$ turbulence model is used, whereby the turbulent viscosity is determined from the relation

$$\mu_{\rm t} = C_{\mu} \rho \, \frac{k^2}{\varepsilon} \tag{5}$$

The kinetic energy of turbulence, k, and the energy dissipation, ε , are obtained from the following equations:

$$U_{z}\frac{\partial k}{\partial z} + U_{r}\frac{\partial k}{\partial r} = \frac{\partial}{\partial z} \left(\frac{v_{t}}{\sigma_{k}}\frac{\partial k}{\partial z}\right) + \frac{1}{r}\frac{\partial}{\partial r} \left(\frac{v_{t}}{\sigma_{k}}r\frac{\partial k}{\partial r}\right) + G - \varepsilon$$
(6)

$$U_{z}\frac{\partial\varepsilon}{\partial z} + U_{r}\frac{\partial\varepsilon}{\partial r} = \frac{\partial}{\partial z}\left(\frac{v_{t}}{\sigma_{\varepsilon}\partial z}\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(\frac{v_{t}}{\sigma_{\varepsilon}}r\frac{\partial\varepsilon}{\partial r}\right) + C_{1}\frac{\varepsilon}{k}G - C_{2}\frac{\varepsilon^{2}}{k}$$
(7)

where the generation of turbulent kinetic energy, G, is given by

$$G = v_{t} \left\{ 2 \left[\left(\frac{\partial U_{z}}{\partial z} \right)^{2} + \left(\frac{\partial U_{r}}{\partial r} \right)^{2} + \frac{U_{r}^{2}}{r^{2}} + \frac{\partial U_{r}}{\partial z} \frac{\partial U_{z}}{\partial r} \right] + \left(\frac{\partial U_{z}}{\partial r} \right)^{2} + \left(\frac{\partial U_{r}}{\partial z} \right)^{2} \right\}$$

The constants appearing here are given by Pope [12] for a round jet as $C_{\mu} = 0.09$, $\sigma_k = 1.00$, $\sigma_{\varepsilon} = 1.30$, $C_1 = 1.44$ and $C_2 = 1.92$.

The governing equations for turbulent flow were discretized using the finite element method and solved with the FIDAP code. Figure 2 illustrates the finite element mesh for the computational domain. Because the flow is assumed to be axisymmetric, only the upper half of the flow domain needs to be solved.

After solution for the fluid flow field, the particle motion and trajectories are determined by solving Lagrangian equations of particle motion. It is assumed that particles are spherical and there is no

Table 1 Standard simulation conditions

Nozzle dimensions:	
diameter d	1 mm
length L	20 mm
straight cylindrical tube diameter	3 mm
d_0	
straight cylindrical tube length	6 mm
tapered inlet angle θ	60°
Density ρ	1000 kg/m^3
Molecular viscosity μ	0.001 kg/(m s)
Particle diameter $d_{\rm p}$	0.01, 0.05, 0.1 and 0.25 mm
Particle density $\rho_{\rm p}^{\rm P}$	4000 kg/m ³
Inlet conditions:	
z velocity u	100 m/s
r velocity v	0 m/s
Turbulent kinetic energy k	$0.001u^2$
Dissipation ε	$k^{3/2}/0.2d$
Inlet Reynolds number Re	3.0×10^{5}

Proc Instn Mech Engrs Vol 213 Part B



Fig. 3 Velocity vector plot of the water flow field in the nozzle



Fig. 4 Water velocity distribution in the nozzle centre-line

particle-particle interaction. The force balance between inertial and drag forces are considered in this study. Further study is required to quantify the importance of other forces such as virtual mass effects, lift, Basset forces, Magnus forces, etc.

By assuming that the main forces acting on the particle are drag and inertia, the motion equation for a single particle can be written as

$$\frac{\mathrm{d}\boldsymbol{u}_{\mathrm{p}}}{\mathrm{d}t} = \frac{f}{\tau} \left(\boldsymbol{U} - \boldsymbol{u}_{\mathrm{p}} \right) \tag{8}$$

where

$$\tau = \frac{d_{\rm p}^2 \rho_{\rm p}}{18\mu} \tag{9}$$

$$f = \begin{cases} 1 + 0.15 R e_{\rm p}^{0.687} & 0 < R e_{\rm p} \le 1000\\ 0.0167 R e_{\rm p} & R e_{\rm p} > 1000 \end{cases}$$
(10)

 τ is the particle response time for momentum transfer between fluid and particle, and f is a drag force correction factor which enables the Stokes drag formula to be used when the particle Reynolds number, Re_p , is of the order of unity or larger.

The velocity, turbulent kinetic energy and turbulent dissipation are specified at the inlet to the nozzle. A uniform velocity profile in the z direction was employed. Turbulent kinetic energy and dissipation were calculated from the formula given in reference [13]. It is assumed that the abrasive particles are moving at the same velocity as the carrier fluid when they enter the nozzle.

4 DISCUSSION OF CFD RESULTS

The main emphasis now is on the application of the CFD analysis to DIAjet nozzles in premixed abrasive water jet systems to assist in nozzle design and development. In order to demonstrate the capability of the CFD analysis in simulating turbulent flow and particle dynamic properties, a set of tests with different particle sizes and nozzle geometries was carried out. The 'standard' simulation conditions are shown in Table 1, to which certain changes will be made and assessed individually.



Fig. 5 Particle velocities in the nozzle centre-line: (a) L = 20 mm, (b) L = 30 mm, (c) L = 40 mm

4.1 Water flow field

A water velocity of 100 m/s is prescribed at the nozzle inlet. Figure 3 shows a vector plot of the water velocity in the nozzle. The velocity distribution at the nozzle outlet is a typical power law curve with a maximum velocity at the nozzle centre-line. Figure 4 shows the variation in water velocity along the nozzle centre-line. The water velocity information provides a basis for particle trajectory calculation. The losses due to gradual contractions in pipes were established by analysis of crane test data. For an initial velocity of the water jet of 100 m/s and for the selected nozzle geometry the pressure difference will be around 120 MPa. For this pressure difference the outlet velocity at the centre-line of the nozzle will be around 500 m/s. The water compressibility can be assumed to be negligible on the basis of the assumption that the Mach number is less than 0.3.

4.2 Particle velocity and trajectories

The particle velocity and trajectory are solved on the basis of the Lagrangian equation of particle motion. It is assumed that particles that impact on the nozzle wall will rebound at an angle equal to the angle of incidence; the restitution coefficient is set to 0.8; i.e. the particle velocity after the impact will be 80 per cent of the impact velocity.

4.2.1 Particle velocity along the nozzle centre-line

The water and abrasive particles have their maximum velocities at the nozzle centre-line. The particle velocity along the nozzle centre-line is representative. The comparison between water velocity and particle velocities along the nozzle centre-line is illustrated in Fig. 5. Because of the higher density, larger size, etc., abrasive particles cannot be accelerated as fast as water and they lag behind the water during the acceleration process. As the particle diameter increases, the slip between particle and water increases. Only smaller particles ($d_p = 0.01$ mm) approach the water velocity at the nozzle outlet under the standard simulation conditions (L = 20 mm, $\theta = 60^{\circ}$). Larger particles need more acceleration time (longer nozzle length).

Figure 5 also illustrates the effect of nozzle length on particle velocity. As the nozzle length increases, the particle velocity at the nozzle outlet is closer to the water velocity. In a nozzle of greater length, particles have a longer time to be accelerated and eventually have higher velocities at the nozzle outlet. Increasing the velocity of the abrasive particle is beneficial, since it results in a higher cutting speed in the cutting process. However, a greater nozzle length results in more water friction loss and may lead to more particle–wall interactions, which will cause more energy loss and more wear of the nozzle wall. These aspects raise the question of optimization of nozzle length.



Fig. 6 Effect of particle size on particle trajectories: (a) $d_p = 0.01$ mm, (b) $d_p = 0.1$ mm, (c) $d_p = 0.25$ mm

4.2.2 Particle trajectory in the nozzle

A set of tests was carried out to show the influence of the particle size and initial location on the particle trajectory. The particle diameter and initial location are chosen to be $d_{\rm p} = 0.01$, 0.10 and 0.25 mm and r = 0.6, 0.8 and 1.0 mm respectively. The results are plotted in Fig. 6. It is noted that the trajectory of a single abrasive particle depends both on the particle diameter and the initial location of the particle at the nozzle inlet. Particles are released into the flow domain at the nozzle entrance with the same velocity of fluid there. Particles of larger diameter deviate from the streamline and impact on the tapered section wall. As they rebound at reduced velocity, they interact with the flow field, which forces them to cross the nozzle centre-line and strike the nozzle wall again. If the nozzle tube is long enough, this process may be repeated several times until particles finally get out of the nozzle outlet. Clearly the size of particles has a big influence on particle trajectories. Small particles follow the fluid motion closely and do

not strike the nozzle wall, while the larger particles with initial locations of larger radial distances obviously have a higher chance of collision with the nozzle wall.

With the particle trajectory information it is possible to determine the location of impingement, the incident angle and the velocity components of an abrasive particle. This enables a quantitative evaluation to be made of the abrasive erosion at the point of impact by applying an erosion model, and then the erosion distribution to be determined by summing up the contribution of all abrasive particles.

4.2.3 Effect of nozzle geometry

The nozzle geometry greatly affects the fluid flow pattern and abrasive particle trajectory. The influence of tapered inlet angles on the particle velocity in the nozzle centre-line can be seen in Fig. 4.

Figure 7 shows the result of particle trajectories in a nozzle of greater length (increasing L from 20 to 40 mm). It is clear that a longer nozzle leads to more



Fig. 7 Particle trajectories in a nozzle (L = 40 mm, $d_p = 0.25 \text{ mm}$): (a) particle trajectories, (b) particle axial velocity component, (c) particle radial velocity component

collisions of particles on the nozzle wall. The velocity variations along the particle path show the particle velocity reduction after each collision with the wall. The zig-zag trajectories of particles cause momentum losses and contribute to the pronounced wear of the nozzle wall.

Figure 8 demonstrates the influence of a tapered inlet angle on the particle trajectory. Comparison of the particle trajectories in two cases ($\theta = 70$ and 50°) shows the difference in impingement times and incident angles. The particle with an initial location at r = 0.8 mm collides with the nozzle wall only twice when $\theta = 70^{\circ}$, compared with three times when $\theta = 50^{\circ}$. With a larger tapered inlet angle, the acceleration of water and particles is smoother and particles have relatively less chance of collision on the nozzle wall and accordingly cause less wear of the nozzle wall. A too large tapered inlet angle leads to a great nozzle length (L_0). An optimization can be achieved by balancing the requirements for nozzle compactness and better flow field.

4.3 Comparison with available experimental data

Experimental data are rare for solid–liquid flows, especially for high-speed cases. Simultaneously obtaining velocity information for both fluid and solid phases is still a challenging problem.

The CFD analysis method used in this work has been verified by the experimental data published by Zoltani and Bicen [14]. A solid-air round jet of 25.4 mm diameter with an exit velocity of 20 m/s and containing 80 μ m beads with a mass density of loading of 1.5 per cent was examined in their tests. The air and solid velocities were measured by laser Doppler velocimetry. Figure 9 shows that the numerical simulations are in quantitative agreement with the experimental study. The experimental data for the solid are slightly larger than those predicted. The explanation for this is that because the experiment was conducted at a downstream location of one jet diameter (z/D = 1), particles had a longer time to be accelerated.



Fig. 8 Effect of tapered inlet angle on particle trajectories: (a) $d_p = 0.1$ mm, $\theta = 70^{\circ}$; (b) $d_p = 0.1$ mm, $\theta = 50^{\circ}$



Fig. 9 Comparison between numerical simulation and experimental data

5 CONCLUSION

CFD analysis is a valuable supplementary technique for the design and development of improved abrasive water jet cutting systems. Available experimental data provide confirmatory comparisons. With the help of CFD analysis, researchers and designers can have a better understanding of the solid–liquid flow field associated with the DIAjet cutting technology before the equipment is actually developed and manufactured. The information gained in CFD analysis is of help in the optimization of design parameters.

The CFD analysis shows that the internal shape of the nozzle is critical to the transport and acceleration of abrasive particles and to the wear of the nozzle wall. The optimized tapered inlet angle, nozzle diameter and nozzle length will lead to better energy utilization and less abrasive erosion in the nozzle.

REFERENCES

- 1 Hashish, M. Comparative evaluation of abrasive-fluid jet machining systems. In Proceedings of ASME 1989 Winter Annual Meeting, PED, 1989, Vol. 41, pp. 13–21.
- 2 Fairhurst, R. M., Heron, R. A. and Saunders, D. H. DIAJET, a new abrasive waterjet cutting technique. In Proceedings of 8th International Symposium on *Jet Cutting Technology*, BHRA, Durham, England, September 1986.
- **3 Bloomfield, E. J.** and **Yeomans, M. J.** DIAJET—a review of progress. In Proceedings of 1st Asian Conference on *Recent Advances in Jetting Technology*, CI-Premier PTE Limited, Singapore, 1991, pp. 21–30.
- **4** Yazici, S. and Summers, D. A. The investigation of DIA-JET (direct injection of abrasive jet) cutting of granite. In Proceedings of 5th American Water Jet Conference, Toronto, Canada, 29–31 August 1989, pp. 343–356.
- 5 Summers, D. A., Yao, J. and Wu, W. Z. A further investigation of DIAjet cutting. In Proceedings of 10th International Symposium on *Jet Cutting Technology*, 1991, pp. 181–193 (Elsevier Science, London and New York).
- 6 Walters, C. L. and Saunders, D. H. DIAJET cutting for nuclear decommissioning. In Proceedings of 10th International Symposium on *Jet Cutting Technology*, 1991, pp. 427–440 (Elsevier Science, London and New York).

- 7 Surle, R. Abrasive jet cutting system for decommissioning, SESAME database. Thermie /06048/88/FR/, EEC project, 1991.
- 8 Laurinat, A., Louis, H. and Tebbing, G. Premixed abrasive water jets—the influence of important parameters. In *Jet Cutting Technology* (Ed. A. Lichtarowicz), 1992, pp. 577– 591 (Kluwer Academic Publishers, Dordrecht).
- 9 Guo, C., Cheng, D. and Liu, L. Mathematical modeling of the accelerating process of particles in DIA-JET. In Proceedings of 7th American Water Jet Conference, Seattle, Washington, 28–31 August 1993, pp. 287–293.
- 10 Abudaka, M. and Crofton, P. S. J. Theoretical analysis and preliminary experimental results for an abrasive water jet cutting head. In Proceedings of 5th American Water Jet Conference, Toronto, Canada, 29–31 August 1989, pp. 79–87.
- 11 Nadeau, E., Stubley, G. D. and Burns, D. J. Prediction and role of abrasive velocity in abrasive water jet cutting. *Int. J. Water Jet Technol.*, 1991, 1(3).
- 12 Pope, S. B. An explanation of the turbulent round-jet/ plane-jet anomaly. Am. Inst. Aeronaut. Astronaut. J., 1978, 16(3), 279–280.
- 13 Dosanjh, S. and Humphrey, A. C. The influence of turbulence on erosion by a particle-laden fluid jet. Wear, 1985, 102, 309–330.
- 14 Zoltani, C. K. and Bicen, A. F. Velocity measurement in a turbulent, dilute, two-phase jet. *Experiments in Fluids*, 1990, 9, 295–300.

Copyright of Proceedings of the Institution of Mechanical Engineers -- Part B --Engineering Manufacture is the property of Professional Engineering Publishing and its content may not be copied or emailed to multiple sites or posted to a listserv without the copyright holder's express written permission. However, users may print, download, or email articles for individual use.