Chapter 6

**CFD Simulation of Abrasive Water Jet Flow:**

*Numerical Formulation and Validation*

This chapter introduces the governing differential equations with assumptions for multiphase flow followed by numerical modelling, grid generation and grid independence test. A numerical scheme involving relevant turbulence model and relevant boundary conditions to solve the flow domain is also covered in this chapter. Validation study carried out with the results available in the literature, is included at the end of the chapter.

6.1 Introduction

In AWJ machining a mixture of water and abrasive called slurry is expelled through the converging nozzle at high velocities. In the nozzle, static pressure of the slurry is converted into a high frequency impact jet causing erosive action through the flux of abrasive particles acting on impacting materials. Slurry flow in AWJ nozzle is influenced by the abrasive parameters such as its size, concentration and type of particles, nozzle geometric parameters like inlet diameter, focus tube diameter and length apart from inlet operating pressure. The present research work investigates the influence of these parameters on jet kinetic energy and wall shear stress in the nozzle. The AWJ nozzle which is connected to the cutting head is shown in Figure 6.1.
6.2 Geometric model of AWJ nozzle and computational flow domain

Two geometric models of the AWJ nozzle i.e., single-step conical nozzle (which is commercially available) and multi-step conical nozzle (under research by the present work) are shown in Figures 6.2 and 6.3 respectively. The single-step conical nozzle consists of a circular inlet bounded by a converging cone and a relatively longer focus tube with circular exit. The function of the converging nozzle is to improve the kinetic energy of the jet and to align the streamlines into a densely packed bundle as it enters the focus tube. Further, the function of focus tube is to stabilize the flow such that the jet is coherent and kinetic energy on the work piece is with least loss. The multi-step conical nozzle consists of three step conical sections each of length 4 mm. The inlet diameter of the nozzle is kept at 4 mm and also the focus tube diameter is taken to be 1.3 mm with focus tube length taken 17 mm as that of the plain conical nozzle for performance comparison. Between each conical section a straight duct of length 4 mm is provided. The conical steps of the multi-step conical nozzle are designed with half cone angle of 5° for the first and second sections followed by 9.23° for the third section.

Fig. 6.2 Geometric model of AWJ (single-step conical) nozzle

Fig. 6.3 Geometric model of multi-step conical AWJ nozzle
The computational domain consists of flow field in the nozzle with converging inlet region which is connected with constant exit section called the focus tube. A mixture of abrasive and water is let into the nozzle at the inlet and is carried down through the conical sections to the focus tube. Geometric modelling in the present work is developed on the basis of research article by Guihua Hu et al. (2008).

6.3 Computational grid generation

Generation of the computational grid requires considerable attention in numerical analysis because grid resolution must capture the flow gradients accurately. Numerical solution involves longer computational time for computational grids of poor quality. Sometimes it may not generate accurate results too. For grid generation, speed of CPU and its memory imposes a limit on size and resolution of the grids in the meshed model. Therefore, numerical analysis requires good expertise in meshing.

In the present work, computational domain is modelled using commercially available preprocessor routine called GAMBIT. The fluid domain in the computational model is discretized with control volumes (cells) of appropriate sizes as shown in Figure 6.4. The enlarged portion of the meshed model shows the boundary attached mesh which helps to capture the wall gradients. To reduce computational time, the flow domain is taken to be axisymmetric and the control volumes are generated by quad type cells, which is the preferred choice for the kind of flow chosen in the present work. Since boundary attached flows are critical for the fluid flows close to the wall, it requires a finer mesh so as to capture steep velocity gradients in this region. Wall region in the flow domain is closely modelled using the boundary layer mesh technique for extracting high velocity gradients near the boundary walls. Computations are carried out on the axisymmetric flow domain.
A structured cell arrangement is adopted to mesh the computational domain in order to obtain accurate solutions. The model consists of two dimensional control volumes (elements) of quad type. The maximum size of the elements is limited to an edge length of 0.001 mm which makes the boundary mesh finer near the walls to capture the effects of wall gradients. Conservation equations are solved for each control volume to obtain the velocity and pressure fields. The computational flow model is solved using finite volume approach in which the governing equations are integrated around the mesh elements. Initially, coarse mesh is adopted and an appropriate solution is obtained. Then the mesh algorithm is changed to make the mesh finer so as to achieve more accurate results until two successive results agree to a desired level of accuracy for proper convergence of results. In the present case convergence is obtained when all the dependent variable residuals fall below $1 \times 10^{-5}$ at all grid points.

Fig. 6.4 Computational flow domain with close up view of the meshed portion
6.4 Numerical scheme

The following section details the numerical scheme adopted in the flow simulation. The flow through AWJ nozzle involves two phases (i.e., mixture of water and abrasive) and hence physics pertaining to multiphase flow is to be adopted. General multiphase models which are available in Fluent solver are volume of fluid model, the mixture model, and the Eulerian multiphase model. An appropriate model is chosen based on the flow regime. The multiphase model is generally chosen by determining the particulate loading and the Stokes number. Particulate loading ($\beta$) is the mass density ratio of the dispersed phase to that of the carrier phase and it has a major impact on phase interactions. The Particulate loading for garnet abrasive at 10% volume concentration is calculated by equation 6.1.

$$\beta = \frac{\alpha_s \rho_s}{\alpha_l \rho_l} = 0.230$$  \hspace{1cm} (6.1)

The computed value indicate that the interaction between the two phases is two-way i.e., the fluid carrier influences the particulate phase via drag and turbulence and the particles in turn influence the carrier fluid via reduction in mean momentum and turbulence. Another parameter which helps to identify appropriate multiphase model is Stokes number ($S_t$), which is the ratio of the particle response time ($\tau_p$) to the system response time ($t_s$). The computation of the $S_t$ is detailed below.

Density ratio ($\gamma$) = $\frac{\rho_s}{\rho_l} = 2.3$ \hspace{1cm} (6.2)

The average distance ($L$) between the individual garnet particles in the flow at 10% abrasive concentration ($k$) is estimated by equation developed by Crowe et. al. (1998).

$$\frac{L}{d_p} = \left(\frac{\pi}{6} \times \frac{1+k}{k}\right)^{\frac{1}{3}} = 1.7925$$ \hspace{1cm} (6.3)

$$\tau_p = \frac{\rho_p d_p^2}{18 \mu_l} = 5.05129 \times 10^{-4}$$ \hspace{1cm} (6.4)
\[ t_s = \frac{I}{v} = 1.42 \times 10^{-3} \]  
(6.5)

\[ S_t = \frac{\tau_p}{t_s} = 0.355 \]  
(6.6)

For the present work, the calculated Stokes number \((S_t)\) is less than unity, which means abrasive particles will closely follow the fluid flow. In the case of two phase flows through nozzle, either mixture model or Eulerian multiphase model is suitable for the flow simulation. The present numerical simulation is carried out using Eulerian multiphase model which, though is more expensive in computation, but gives better accuracy.

The Euler’s governing equation of motion which is in the form of partial differential equations is used to predict the flow variables. The solver uses a multi-fluid granular model to describe the flow behaviour of slurry. The stress induced by the solid phase is deduced through an analogy between the random particle motion arising from particle to particle collisions as well as thermal gradients of molecules in the fluid stream, taking into consideration the inelasticity of the granular phase. Intensity of the particle velocity fluctuations determines the stress, viscosity and pressure of the solid phase. The governing equations for conservation of mass, momentum and turbulence are solved for the steady incompressible flow using the governing equations as detailed below.

### 6.4.1 Continuity equation

The general form of continuity equation for two phase flow is given by the equation (6.7).

\[
\frac{\partial}{\partial t} \left( \alpha_q \rho_q \right) + \nabla \cdot \alpha_q \rho_q \mathbf{v}_q = \sum_{p=1}^{N} \left( \dot{m}_{pq} \mathbf{v}_q \right) + \dot{S}_q
\]  
(6.7)

In the equation 6.7, the first term represents the time dependent volume fraction and density and the second term represents the deviation of velocity along the direction of flow which is balanced by the equation of mass flow rate and the source term for the other phase of the
flow. The generalized equation is reduced to the equation 6.8 and is adopted for the present case of steady incompressible fluid flow. Since the source term on the right side has very little effect on the flow, it is neglected.

$$\nabla \cdot \alpha_q \rho_q v_q = \sum_{l=1}^{N} \left( \dot{m}_{pq} - \dot{m}_{qp} \right)$$

(6.8)

6.4.2 Momentum equations for fluid and solid phases

The general form of conservation equations for momentum of solid and liquid phases is given by the equation 6.9 and 6.10 respectively. These equations consist of time dependent variables for volume fraction, density and flow velocity which are balanced by the body forces.

**Momentum equation for fluid phase**

$$\frac{\partial}{\partial t} \left( \alpha_s \rho_s v_s \right) + \nabla \cdot \left( \alpha_s \rho_s v_s^2 \right) = -\alpha_s \nabla p - \nabla p_s + \nabla \cdot \tau_s + \alpha_s \rho_s g + \sum_{l=1}^{N} \left[ k_{ls} (v_l' - v_s) + (\dot{m}_{ls} v_{ls} - \dot{m}_{sl} v_{sl}) \right]$$

$$+ \left( F_s + F_{lift.s} + F_{vm.s} \right)$$

(6.9)

**Momentum equation for solid phase**

$$\frac{\partial}{\partial t} \left( \alpha_q \rho_q v_q \right) + \nabla \cdot \left( \alpha_q \rho_q v_q^2 \right) = -\alpha_q \nabla p + \nabla \cdot \tau_q + \alpha_q \rho_q g + \sum_{l=1}^{N} \left[ k_{pq} (v_p' - v_q) + (\dot{m}_{pq} v_{qp} - \dot{m}_{qp} v_{qp}) \right]$$

$$+ \left( F_q + F_{lift.q} + F_{vm.q} \right)$$

(6.10)

6.4.3 The turbulence modelling

In the present work, standard k-ε turbulence model is used to predict the effects of turbulent fluctuations of the flow velocities and scalar quantities. The model is found to be consistent with the physics of turbulent flows involved in the present work (Liu et. al, 2004 and Launder et. al., 1972). In this model the Reynolds stresses are assumed to be proportional to the mean velocity gradients, with the constant of proportionality being the turbulent viscosity. The values of k and ε required to model the turbulence effects are obtained by solving the following conservation type model equations.
**Equation for turbulent kinetic energy**

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j} \left( \frac{\mu}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + G_k - \rho \varepsilon \tag{6.11}
\]

**Equation for dissipation rate**

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho u_j \varepsilon) = \frac{\partial}{\partial x_j} \left( \frac{\mu}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} G_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \tag{6.12}
\]

Where, \(C_{1\varepsilon}\) and \(C_{2\varepsilon}\) are empirical constants, \(\sigma_k\) and \(\sigma_\varepsilon\) are “Prandtl” numbers governing the turbulent diffusion of \(k\) and \(\varepsilon\), \(G_k\) is the rate of production of turbulent kinetic energy.

\[
G_k = \mu \left( \frac{\partial u_j}{\partial x_j} + \frac{\partial u_i}{\partial x_i} \right) \frac{\partial u_j}{\partial x_j} \tag{6.13}
\]

\[
G_k = g_i \frac{\mu}{\rho \sigma_n} \frac{\partial \rho}{\partial x_i} \tag{6.14}
\]

The following empirically derived standard coefficient values are used in the present work.

<table>
<thead>
<tr>
<th>(C_{1\varepsilon})</th>
<th>(C_{2\varepsilon})</th>
<th>(C_\mu)</th>
<th>(\sigma_k)</th>
<th>(\sigma_\varepsilon)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.44</td>
<td>1.92</td>
<td>0.09</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>

**6.5 The solver scheme**

Accuracy of the solution depends on the method used to solve the governing equations over the computational grid. Fluent solver provides many schemes such as first order upwind scheme, second order upwind scheme, power law and Quadratic Upstream Interpolation for Convective Kinematics (QUICK) discretisation schemes for flow calculations. Optimal choice of discretization method depends on flow characteristics and type of grids generated.

In the present investigation, a boundary layer mesh provides a high resolution grid density which is required to resolve the separated shear layer on the nozzle wall surface. As a result, the discretization scheme implemented by solver is capable of resolving the steep gradients and turbulent mixing of the two phase flow parameters. In the present work, the QUICK discretization scheme is used for obtaining the solution. The linkage between pressure and
velocity is handled through a more efficient version of the well-known Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm developed by Patankar and Spalding (1980).

6.6 General assumptions in numerical modelling

- Flow is taken to be two-phase and steady turbulent flow.
- The primary phase is a continuous liquid phase and is incompressible.
- There is no slip between the two phases of flow.
- The primary liquid phase mixes homogeneously with the particles of equal diameter, constituting the solid phase.

6.7 Boundary conditions and operating parameters

Figure 6.5 shows the boundary conditions that are imposed on the computational domain, as per the physics of the problem. Inlet boundary condition is specified by applying operating pressure at the nozzle entrance. It is assumed that velocity at inlet is uniform over the cross section. At the exit, static pressure of effluxing flow is taken to be zero (gauge), so that the computation would proceed by the relative pressure differences across the grid volumes for the entire domain of the flow.

![Wall boundary conditions](image)

**Fig. 6.5 Boundary conditions for the flow domain**

Wall boundary conditions are impressed to bound the fluid and solid regions. In viscous flow models, as in the present case, velocity components at the wall are set to zero in accordance with the no-slip and impermeability conditions that exist at the wall boundary. The axis of the
nozzle is used to solve the computational domain as axisymmetric problem and suitable boundary conditions are imposed for the same i.e., the gradient of fluid properties are set to zero across the axis line. In the present numerical simulation, water is treated as primary phase and garnet abrasive as secondary phase. The Fixed input parameters used in the analysis are as shown in the Table 6.1 below.

Table 6.1 Fixed input parameters for numerical simulation

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density of water (phase-I)</td>
<td>998.2 kg/m³</td>
</tr>
<tr>
<td>Density of garnet abrasive (phase-II)</td>
<td>2300 kg/m³</td>
</tr>
</tbody>
</table>

6.8 The grid independence test

The grid independence test is performed to check the quality of mesh for solution convergence. Three mesh geometries consisting of 21200, 25440, 29690 control volume cells for plain conical nozzle and 58320, 87480 and 116640 control volume cells for multistep conical nozzle were tested. Grid test is made against the velocity distribution along the axis of the nozzle for inlet velocity of 25.6 m/s for the flow of garnet abrasive and water. The test results are discussed in the following section.

6.8.1 Velocity distribution

The results are simulated for three different mesh densities and axial velocity of flow all along the axis of the nozzle is plotted for each mesh density. From the grid independence study it is seen from the graphs (Figure 6.6 and 6.7) that there is only a minor variation of velocity in the downstream region of the nozzle which is about 1% for the case (c). However this variation is within the allowable limits. Hence considering lower computational time, an optimum mesh geometry consisting of 25440 and 58320 control volumes has been adopted in this work for
plain and multistep conical nozzle respectively. The runs were performed on computer having Intel ® Xeon® central processing unit with 8 GB RAM and clock speed of 3.10 GHz.

Fig. 6.6 Grid independence test results for single-step conical nozzle

Fig. 6.7 Grid independence test results for multi-step conical nozzle
6.9 Validation of the numerical model

In any numerical analysis of fluid flow, validation of the numerical model needs to be done necessarily. Generally validations are made against known results in a similar flow scenario either from a numerical analysis or from an established experimental procedure. It is found from the literature review that Guihua Hu et. al. (2008) conducted a numerical analysis for optimization of focus tube length of AWJ nozzle, the results of which are available for velocity variable along the axis of the AWJ nozzle. This has been adopted in the present work for validation. The graphs of the velocity distribution of one of the phases (Liquid phase) obtained in the present work and cited in the literature are shown in Figure 6.8 (b) and 6.8 (a) respectively. It is observed from these figures that, the flow velocity of one of the phases raises sharply up-to a portion of the nozzle where it changes from conical section to straight length. Thereafter it almost remains constant till the exit of the nozzle. Hence, the numerical model adopted in the present work has a fair agreement as regards to the distribution of velocity in the domain and calibrates the model adopted in the present work with the model taken from the literature.

![Graph showing velocity distribution](image)

Inlet velocity = 25.6 m/s

Fig. 6.8 (a) Velocity distributions as obtained by Guihua Hu et. al. (2008)
Fig. 6.8 (b) Velocity distributions as obtained in the present work