Flow analysis of abrasive micro-blasting with glycerol and acrylamide as carrier medium using computational fluid dynamics

S.Ramanathan¹, E.Naveen¹, R.M.Vignesh², S.K.Sudharson² and B.Charan²

¹Faculty-Department of Mechanical Engineering, Sri Sai Ram Engineering College, Chennai-44
²UG Scholar-Department of Mechanical Engineering, Sri Sai Ram Engineering College, Chennai-44

E-mail: sanjiv2020@gmail.com

Abstract. Continuous growth is observed in the field of Non-Traditional machining where the machining of newly developing materials is the need of the current segment. Nowadays Abrasive Fluid Jet Machining has used to machine a wide range of elastic and plastic materials including aerospace, automobile, ordinance and combat as well as day to day applications which require high strength to weight ratios. In Abrasive fluid jet machining, the abrasives are mixed with a liquid to form a slurry. The cutting performance is degraded by the rapid wear of the nozzle by the flow of the Abrasive Fluid mixture through the nozzles that lead to the divergence of the Abrasive Fluid Jet. The Angle of impingement affects the machining responses such that total cutting time has its influence. The Wear characteristics of the Nozzle Material are critical for such machining. The Nozzle inlet pressure of the abrasive fluid jet has a magnanimous effect on the erosion characteristics inside the nozzle. An analysis was carried out with constant Nozzle Taper angle and with Glycerol & Acrylamide solution as a carrier medium. The aim of this work is to analyze the effect of inlet operating pressure on wall shear and exit kinetic energy with respect to glycerol and Acrylamide solution. The two-phase flow analysis was carried by using a computational fluid dynamics tool CFX. The availability of optimized process parameters of abrasive fluid jet machining is limited to water practically. The other Carrier Medium for Abrasive Fluid Jet Machining can be explored widely. In this case, computational fluid dynamics analysis might provide better results than the real-time experimental work.

1. Introduction
The requirements for sustainable machining demand the Machining of Materials with minimum wastage. Machining contributes about 40% in the Manufacturing sector. One such machining process is Abrasive Water Jet Machining where the liquid pressure and abrasive are transformed thereby the Abrasive Water jet gains momentum such that the material removal is done. In the water jet cutting process, is limited to soft materials machining like Polymers, Foams, fleece, etc. by the wearing away of work material by means of Hydro-Kinetic energy. Cutting harder materials demand the use of abrasives that are mixed in the mixing chamber with pressurized water jet. The placement of nozzle as well as the orifice must be precise. This precise setup makes the abrasive waterjet to diverge minimally. The Wear of nozzle by the abrasives cause the divergence of fluid jet resulting in the geometrical inaccuracy of the machined components. The visualization of the fluid flow of AFJM must be done in advance of machining such that the flow properties are studied well before. The result of
The abrasive fluid flow in the nozzle was affected by the abrasive particles using CFD. Liu et al. [1] performed CFD simulation to determine the flow features of abrasive fluid. Some of the understandings from the various studies are shown:

The abrasive particles in water jet produce severe erosion of the jet nozzle material was observed by Deepak et al. [2]. The nozzle diameter was altered that directly affected the abrasive fluid jet kinetic energy. They concluded that the increased wall shear stress-induced is directly proportional to inlet pressure.

The Material Removal Rate (MRR) rises proportionally with arise in the inlet pressure of air and stops during the maximum rise of pressure was experimentally verified by Ray et al. [3].

The result of the input process constraints on the material removal rate (MRR) and external area & Entrance of the diameter of the obtained hole were measured by Bhaskar Chandr et al. [4].

Andrej Lebar et al. [5] detected that abrasive water jet (AWJ) implemented surfaces revealed higher surface finish in the higher part and rougher finish in the lower part with striations.

Coray et al. [6] stressed for the current AWJ cutting improvement techniques where the market for high precision parts machined by abrasive water jet (AWJ) machining is high. Dewan et al. [7] treated Granular Abrasive particles and carrier medium Water as a continuous phase. Inter-Phase Slip Algorithm (IPSA) was used for finding a solution for finding the fluid characteristics.

A theoretical approach for evaluating the turbulent flow and particle dynamic properties in the nozzles were developed by Kovacevic et al. [8]. It was developed because direct measurements in nozzles of high flow speed and small dimension were practically not feasible.

Liu et al. [9] performed CFD simulation that provided sufficient information of particles for determining the particle angle of impact, speed impinging and impingement location.

Ramanathan et al. [10] analyzed abrasive flow by changing the dimensional constraints of the nozzle using CFD Analysis for obtaining an optimized set of process parameters for effective machining.

Ramanathan et al. [11] carried a simulation of abrasive particles using water as a carrier medium using CFD.

As of the literature the subsequent points shall be considered:

- Current nozzle angle of 30° would be equated with 60°, 15° and 45°.
- Glycerol & Acrylamide mixture would be employed as carrier medium.

The purpose of analysis are listed down:

- To analyze the flow features of the abrasive fluid jet on the inner surface of the nozzle
- To analyze the effect of wear on the surface of the nozzle with:
  a) A mixture of water and glycerol
  b) A mixture of water and acrylamide
- To optimize nozzle features and analysis constraints to minimize the nozzle wear out:
  a) The taper angle of the nozzle
  b) Carrier medium inlet pressure.

2. CFD Analysis

2.1. Modeling

Modelling was done and exported in IGES format using Pro E WildFire 2.0. The various nozzle taper angle modelling have been shown below through figures 1-4. Figure 1 shows the modeled nozzle taper angle at 15°. Figure 2 shows the modeled nozzle taper angle at 30°. Figure 3 shows the modeled nozzle taper angle at 45°. Figure 4 shows the modeled nozzle taper angle at 60°.
The dimensions of the Nozzle head are listed down:
The diameter of the Focus tube (Mixing tube): 0.76 mm
Focus tube distance: 76 mm
Nozzle Taper angle: 45 deg
The diameter of the Mixing chamber: 6 mm
The distance of Mixing chamber: 12 mm
Orifice dia: 0.2 mm
Water inlet dia: 2.5 mm
Abrasive inlet dia: 3 mm

2.2. Meshing
ANSYS ICEMCFD’s mesh creation properties provides the ability to parametrically generate meshes from the modeling file in various types:

i. Hexa and tetrahedral un structural meshing
ii. Cartesian with H-grid refinement Hybrid Meshes including tetrahedral, hexahedral, pyramidal prismatic elements
iii. Triangular and Quadrilateral surface meshes.

Figure 5 and 6 presents the mesh synchronized model of the Nozzle head with different angles.
3. Properties of the carrier medium
Properties of various Carrier medium at 25°C is given in Table 1.

| Property of the Fluid                      | Water          | Glycerol | Acrylamide | 90% water + 10% Glycerol | 90% water + 10% Acrylamide |
|-------------------------------------------|----------------|----------|------------|--------------------------|-----------------------------|
| Molecular weightg/mole                    | 18             | 92       | 72         | 1023.23                  | 1001.66                     |
| Dynamic Viscosity Ns/m²                   | 0.000893       | 0.90568  | 0.0011     | 0.09137                  | 9.134*10^-4                 |
| Density Kg/m³                             | 996.85         | 1260.7   | 1045       | 0.5659                   | 0.5543                      |
| Specific heat capacity J/kg K             | 4178           | 2416     | 1.93 * 10³ | 4001.8                   | 3953.2                      |
| Thermal conductivity W/m K                | 0.596          | 0.2816   | 0.17       | 25.4                     | 23.4                        |

4. Results and discussions
4.1. Outcome of Glycerol and Acrylamide of Mixture
4.1.1 Variation in Velocity
Figure 7 & Figure 8 presents the change in velocity alongwith the mixing chamber and focusing tube distance. In the case of mixing chamber, there is a steady decrease in the velocity and rises when at the mixing region and then it decreases gradually in both cases near the tube end. In the case of focusing tube distance, when the flow is across the nozzle the rise in velocity is detected. When the stream of fluid is along the focus tube.
Further kinetic energy drop is detected in both cases. The velocity variation almost same for both glycerol and acrylamide mixture.

4.1.2. Wall shear stress

Figure 7. Mixing Chamber.

Figure 8. Focus tube.

Figure 9. Mixing Chamber.
Figure 9 & Figure 10 presents the wall shear stress distribution throughout with the mixing chamber and focusing tube distance. Figure 9 shows that marginal increase wall shear at the mixing chamber entry for glycerol mixture and it continues along with the flow. The magnitude increases at the mixing region after that it declines steadily when the flow reaches the end of the tube. Figure 10 shows the increase in wall shear due to the flow over the nozzle in both cases. The relatively improved measure of wall shear was detected for glycerol mixture.

4.1.3. Energy Dissipation

Figure 11. Mixing Chamber.
Figure 12. Focus tube.

Figure 11 & 12 presents the energy dissipation because of wall shear stress with the mixing chamber and focusing tube distance. Figure 11 presents that the level of energy dissipation is lower at the inlet of the mixing tube and it raises up to maximum value in the mixing chamber region and then sharply falls for both the cases. Figure 12 shows energy dissipation increases in the nozzle and it continuously varies along with the flow. The energy dissipation is comparatively high for glycerol mixture.

4.1.4. Pressure Gradient

Figure 13. Mixing Chamber.
Figure 13 & 14 illustrate the pressure gradient throughout with the mixing chamber and focusing tube distance. The velocity of the axis get triggered due to pressure gradient and hence drop and reason for eddies which will subsequently raise the energy dissipation. It has been detected from the graph that the pressure gradient is significantly higher at the inlet of the mixing tube for both the cases its measuremen decreases sharply just before the mixing region. The pressure gradient is maximum at the mixing region. There has been a marginal rise in pressure gradient has been observed for glycerol mixture near the outlet of mixing tube. For both cases, the pressure gradient is measurably low at the inlet of the nozzle.

5. Conclusion
Thus, the study of the flow of carrier fluid through a nozzle was carried out by CFD analysis and the down coming inference are drawn

5.1. Result of Nozzle Angle
- Kinetic energy loss throughout the focusing tube has been observed. This may be caused due to the collision of focusing tube wall and abrasive particles. At 45° taper angle the kinetic energy loss is appreciably less as well as there will be lower pressure gradient. Wall shear lowers at 30°

5.2. Result of Inlet Water Pressure
- A greater water pressure velocity accumulation is more. Additionally, when the flow is near the focusing tube there is a detection of kinetic energy loss for all cases. There is a raise in wall shear due to inlet water raise. At lesser pressures of water, there may be lesser pressure gradients.

5.3. Result of Glycerol and Acrylamide Mixture
- The velocity gain and shear rate are relatively uniform for glycerol and acrylamide mixture
- Comparatively reduced wall shear and energy loss in the flow have been observed for acrylamide mixture along with the nozzle and focus tube.
- Marginal decrease in pressure gradient has been observed for acrylamide mixture in the focus tube.
6. Reference

[1] H Liu, J Wang, R J Brown and N Kelson 2003 Computational Fluid Dynamic (CFD) Simulation of Ultrahigh Velocity Abrasive Water jet Key Engineering Materials 477-482

[2] Deepak D Anjaiah D and N Yagnesh Sharma 2011 Numerical Analysis of Flow through Abrasive Water Suspension Jet: The effect of Inlet Pressure on Wall Shear and Jet Exit Kinetic Energy Proceedings of the World Congress of Engg.

[3] PK Ray, Member and Dr A K Paul 1987 Studies on Abrasive Jet Machining Journal of the Institution of Engineers (India) 68

[4] Bhaskar Chandra and Jagtar Singh 2011 A Study of Effect of Process Parameters of Abrasive jet machining International Journal of Engineering Science and Technology 3

[5] Andrej Lebar and Michael Junkar 2004 Simulation of abrasive water jet cutting process: Part 1. Unit event approach Modeling and Simulation in Materials Science and Engineering 12 1159

[6] Coray PS, Jurisevic B, Junkar M and Heiniger KC 2012 Measurements on 5: 1 scale abrasive water jet cutting head models

[7] Dewan Ahmed and Elias Siores 2001 Numerical Simulation of Abrasive Water Jet for Taper Inlet Angles 14th Australasian Fluid Mechanics Conference Adelaide University Adelaide Australia

[8] Ye J and Kovacevic R 1999 Turbulent solid-liquid flow through the nozzle of premixed abrasive water jet cutting systems Proceedings of the Institution of Mechanical Engineers, Part B: Journal of Engineering Manufacture 213 59-67

[9] Liu H, Wang J, Brown RJ and Kelson N 2003 Computational fluid dynamics (CFD) simulation of ultrahigh velocity abrasive waterjet Key Engineering Materials 1 477-82

[10] S Ramanathan, J Jeykrishnan, J Sylvester Diaz, M Sarath, N Sriram, V G Ramanujam, C Pratheep, S Caleb Stevenson and B. Karthik 2018 Analysis of Taper Angle in Nozzle in Abrasive Fluid Jet Machining using CFD International Journal of Mechanical Engineering and Technology 9 271–279

[11] S Ramanathan, E Naveen, K Vijay, M J. CampbellTerrin and S AnishKisshore 2019 Flow Simulation in Abrasive Fluid Jet Machining with Water as Carrier Medium using CFD International Journal of Innovative Technology and Exploring Engineering (IJITEE) 8