Vol-08 Issue 07, July -2024

ISSN: 2456-9348
Impact Factor: 7.936

iJETRM
International Journal of Engineering Technology Research & Management
Published By:
https://www.ijetrm.com/

ABRASIVE FLUID JET MACHINING NOZZLE WEAR ANALYIS USING CFD

Hariharan D[1]
Saraan S[2]
Arun Kumar R[3]
Magesh S[4]
UG Student, Department of Mechanical Engineering,
Karpaga Vinayaga College of Engineering and Technology

Mr.J.Ekanthamoorthy[5]
Assistant Professor, Department of Mechanical Engineering,
Karpaga Vinayaga College of Engineering and Technology

ABSTRACT
Abrasive fluid jet cutting is one of the most recently developed nontraditional manufacturing technologies. In Abrasive fluid jet machining the abrasives are premixed with a suspended liquid to form slurry. The general nature of flow through the abrasive fluid jet machining, results in rapid wear of the nozzle which degrades the cutting performance. Nozzle replacement costs play a significant role in the economics of the machining process and improvements in its wear characteristics, are critical for the growth of such machining technology. It is well known that the inlet pressure of the abrasive fluid suspension has significant effect on the erosion characteristics of the inside surface in the nozzle. The objective of the project is to analyse the effect of inlet operating pressure on wall shear and exit kinetic energy. The analysis would be carried out by using different fluids with abrasive particles and by changing the physical dimensions of the nozzle so as to obtain optimised process parameters for effective machining. The two-phase flow analysis would be carried by using a computational fluid dynamics tool CFX. The availability of optimized process parameters of abrasive fluid jet machining is limited to water and experimental test can be cost prohibitive. In this case computational fluid dynamics analysis would provide better results.

Keywords:
Computational fluid dynamics tool (CFD)

1.INTRODUCTION
An abrasive water jet is one of the most recently developed non-traditional manufacturing processes. This technique uses jet of water which contains abrasive material. Usually, the water exits a nozzle at a very high speed and the abrasive material is injected into the jet stream. This process is sometimes known as entrainment in which the abrasive particles become part of the moving water like the passengers become part of a moving train. Hence, as with a train the water jet becomes the moving mechanism for the particles. The purpose of the abrasive water jet is to perform some machining or finishing operations. The use of the abrasive water jet for machining or finishing purposes is based on the principle of erosion of the material upon which the jet hits. Each of the two components of the jet, i.e. the water and the abrasive material has both a separate purpose and a supportive purpose. It is the primary purpose of the abrasive material within the jet stream to provide the erosive forces. It is the primary purpose of the jet to deliver the abrasive material to the work piece for the purpose of erosion. However the jet also accelerates the abrasive material to a speed such that the impact and change in momentum of the abrasive material can perform its function. In addition it is an additional purpose of the water to carry both the abrasive material and the eroded material clear of the work area so that additional processing can be performed. Abrasive water jet machining offers the potential for the development of a tool which is less sensitive to material properties, has virtually no thermal effects, and imposes minimal stresses. This process was first introduced as a commercial system in 1983 for cutting of glass. Nowadays, this process is being widely used for machining of hard to machine materials like ceramics, ceramic composites, fiber-reinforced composites, and titanium alloys where conventional machining is often not technically or

iJETRM
International Journal of Engineering Technology Research & Management
Published By:
https://www.ijetrm.com/

economically feasible. The fact that it is a cold process has important implications where heat-affected zones are to be avoided. Several studies have addressed the modeling of taper angle of nozzle to improve wear characteristics:
H. Liu (1) There is a minimum variation of the water pressure and velocity within the computational domain in the jet axial direction. It also suggests that the cutting performance is relatively independent of standoff distance for the typical standoff distance used in AWJ cutting. It has also shown that the velocity and pressure variation in the radial direction is not significant within about 80% to 90% of the jet diameter. It has been shown that CFD simulation can provide the information of particle velocities and trajectories which make it possible to determine the particle impact angle as well as the impinging speed and location.
D.Kramar (2) monitored the interaction of the AWJ with the work piece, the generated sound, vibrations of the work piece and the back flow pressure of the jet from the cutting front was measured. Process parameters such as stand-off distance and abrasive mass flow rate were measured by the generated sound and acoustic emission respectively. By using the recently developed models and monitoring techniques in the Laboratory of Alternative technologies at the University, it was proved that by applying the semi empirical modeling it was possible to increase the machining quality and reduce the machining cost and time simultaneously.
D. Deepak (3) abrasive particles moving with the flow causes severe wall shear, thereby altering the nozzle diameter due to wear which in turn influences the jet kinetic energy. This will reflect on the life of the nozzle for effective machining. By considering this aspect, here the effect of inlet pressure on wall shear stress and jet kinetic energy is considered and analysed. It was found that an increase in inlet pressure resulted in significant increase in the wall shear stress induced.
P K Ray (4) when the Material Removal Rate (MRR) increases with increase of air pressure and the MRR ceases when the pressure reaches a threshold value. It is also observed that MRR is increased with increase in grain size and increase in nozzle diameter. The dependence of MRR on stand-off distance reveals that MRR increases with increase in standoff distance at a particular pressure.

2. DEFINITION OF PROBLEM

The typical flow through the abrasive fluid jet machining causes the nozzle to wear down quickly, which lowers the cutting efficiency. The abrasive fluid suspension's inlet pressure significantly influences the nozzle's internal surface erosion properties.

The economics of the machining process are heavily influenced by nozzle replacement costs; therefore, advancements in nozzle wear characteristics are essential to the development of this type of machining technology. The use of alternative fluids in place of water is still a topic of investigation.

3. BASICS OF COMPUTATIONAL FLUID DYNAMICS

3.1 Introduction

The equations regulating fluid motion can be approximated numerically with CFD. To investigate a fluid problem using CFD, the following stages must be followed. First, the fluid flow is described using mathematical equations. Usually, a collection of partial differential equations makes up these. After that, these equations are discretized to provide a numerical analogue. Next, the domain is split up into smaller components called grids. Ultimately, these equations are solved using the initial circumstances and the boundary conditions of the particular issue. Iterative or direct solutions are also possible. Furthermore, the method's accuracy, stability, and convergence are managed by a set of control parameters.
All CFD codes contain three main elements: (1) A pre-processor, which is used to input the problem geometry, generate the grid, and define the flow parameter and the boundary conditions to the code. (2) A flow solver, which is used to solve the governing equations of the flow subject to the conditions provided. There are four different methods used as a flow solver: (i) finite difference method; (ii) finite element method, (iii) finite volume method, and (iv) spectral method. (3) A post-processor, which is used to massage the data and show the results in graphical and easy to read format.

3.2 Fluid Dynamics

Fluid Dynamics is the study of fluids (liquids and gases) in motion. The basic equations governing fluid motion have been known for more than 150 years and are cumulatively called the Navier-Stokes equations which govern the motion of a viscous, heat conducting fluid.
Various simplifications of these Navier-Stokes' equations exist depending on which effects are insignificant. There are several dimensionless parameters which characterize the relative importance of various effects. Some of these are Mach number, Reynolds number and Prandtl number.

3.3 Computational Fluid Dynamics

iJETRM
International Journal of Engineering Technology Research & Management
Published By:
https://www.ijetrm.com/

Prior to the era of computers, the availability of analytical solutions to the various industrially relevant Fluid dynamics problems were rendered dormant due to the unavailability of the requisite computing power to solve the millions of mathematical equations that led to the results.
However, with the arrival of personal computing, the immense number-crunching power of the digital devices were put to use by programmers to develop codes that infused the equations of fluid flow into the memory of the computers by means of algorithms and programming methods. This marked the arrival and extensive use of Computational Fluid Dynamics or CFD in the solution of industrial problems.
Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions.
Even with high-speed supercomputers only approximate solutions can be achieved in many cases. Ongoing research, however, may yield software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is often performed using a wind tunnel with the final validation coming in flight test.

3.3.1 Considerations in CFD

The most fundamental consideration in CFD is how one treats a continuous fluid in a discretised fashion on a computer. One method is to discretise the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion (Euler equations for in viscid and Navier–Stokes equations for viscous flow.
This is necessitated by the fact that computational methods can only be applied to finite number of elements and not to a continuous domain as this would translate to practically infinite nodes for computation and thus put a tremendous load on the processing hardware and render the solution impossible.
In addition, such a mesh can be either irregular (for instance consisting of triangles in 2D, or pyramidal solids in 3D) or regular; the distinguishing characteristic of the former is that each cell must be stored separately in memory. Where shocks or discontinuities are present, high resolution schemes such as Total Variation Diminishing (TVD), Flux Corrected Transport (FCT), Essentially Non-Oscillatory (ENO), or MUSCL schemes are needed to avoid spurious oscillations (Gibbs phenomenon) in the solution.

4. Discretization Methods

Some of the common discretization methods used in CFD packages are

4.1 Finite Volume Method

This is the "classical" or standard approach used most often in commercial software and research codes. The governing equations are solved on discrete control volumes. FVM recasts the PDE's (Partial Differential Equations) of the N-S equation in the conservative form and then discretise this equation. This guarantees the conservation of fluxes through a particular control volume. Though the overall solution will be conservative in nature there is no guarantee that it is the actual solution. Moreover this method is sensitive to distorted elements which can prevent convergence if such elements are in critical flow regions. This integration approach yields a method that is inherently conservative (i.e. quantities such as density remain physically meaningful)

$$\frac{\partial}{\partial t}\iiint Q\, dV + \iint F\, d\mathbf{A} = 0,$$

Where Q is the vector of conserved variables, F is the vector of fluxes (see Euler equations or Navier–Stokes equations), V is the cell volume, and $\mathbf{A}$ is the cell surface area.

4.2 Finite Element Method

This method is popular for structural analysis of solids, but is also applicable to fluids. The FEM formulation requires, however, special care to ensure a conservative solution. The FEM formulation has been adapted for use with the Navier–Stokes equations.
Although in FEM conservation has to be taken care of, it is much more stable than the FVM approach. Consequently it is the new direction in which CFD is moving. Generally stability/robustness of the solution is better in FEM though for some cases it might take more memory than FVM methods. In this method, a weighted residual equation is formed:

$$R_i = \iiint W_i Q\, dV^e$$

iJETRM
International Journal of Engineering Technology Research & Management
Published By:
https://www.ijetrm.com/

Where R_i is the equation residual at an element vertex i , Q is the conservation equation expressed on an element basis, W_i is the weight factor and V^e is the volume of the element.

4.3 Finite Difference Method

This method has historical importance and is simple to program. It is currently only used in few specialized codes. Modern finite difference codes make use of an embedded boundary for handling complex geometries making these codes highly efficient and accurate. Other ways to handle geometries are using overlapping-grids, where the solution is interpolated across each grid.

$$\frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = 0$$

5.1 Modeling

Modeling was done using Pro E Wild Fire 2.0 and exported in IGES format. The models of nozzle heads have been shown in the following figures

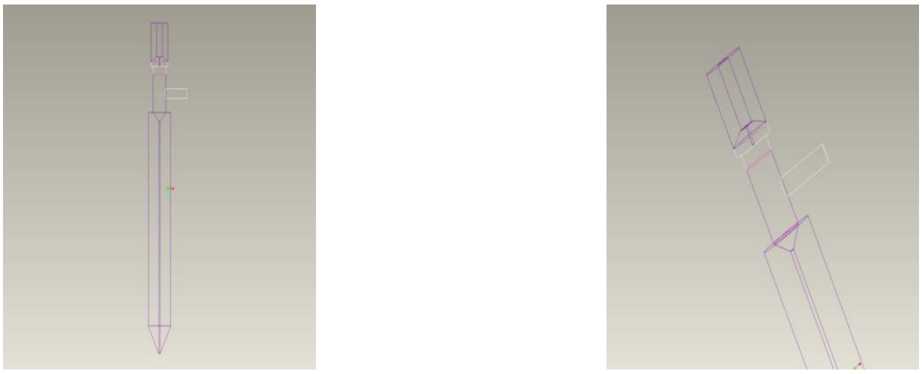


Fig 5.1 Pro E Model of Nozzle Head

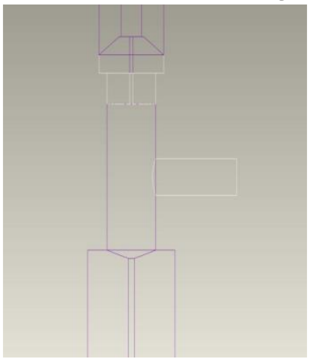


Fig 5.2 (a) 15°Taper Angle

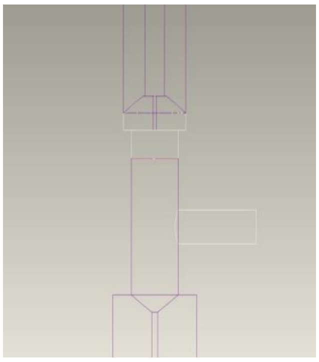


Fig 5.2 (b) 30^0 Taper Angle

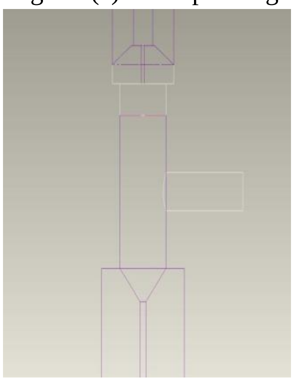


Fig 5.2 (c) 45^0 Taper Angle

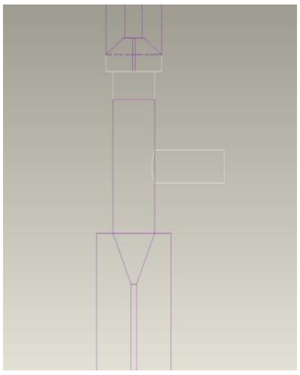


Fig 5.2 (d) 60^0Taper Angle

Focus tube (Mixing tube) Diameter : 0.76 mm

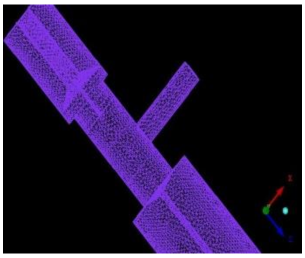
iJETRM
International Journal of Engineering Technology Research & Management
Published By:
https://www.ijetrm.com/

Focus tube length : 76 mm
Taper angle of nozzle : 45 deg
Mixing chamber diameter : 6 mm
Mixing chamber length : 12 mm
Orifice diameter : 0.2 mm
Water inlet diameter : 2.5 mm
Abrasive inlet diameter : 3 mm

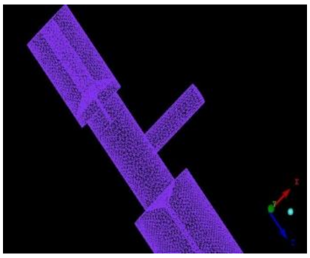
5. Result and Discussion

5.1 Modified Design of taper angle of nozzle.

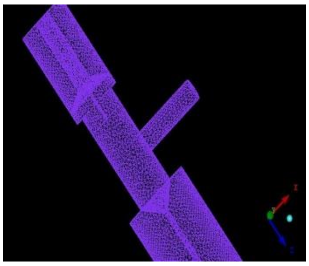
The nozzle angle model modified with angle of 15^0, 30^0, 45^0, and 60^0 respectively as shown. The volume mesh of the model has been generated with tetrahedron element.



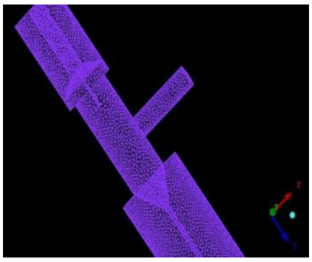
(a) Taper angle 150



(b) Taper angle 300



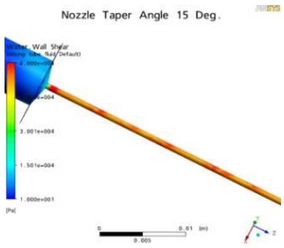
(c) Taper angle 450



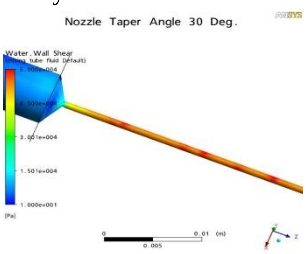
(d) Taper angle 600

5.2 Effect of modified nozzle angle.

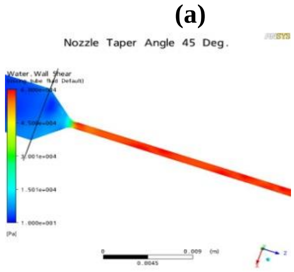
The models were analyzed for the wall shear stress conditions. The analyses of results for 30^0 show that reduced wall shear stress. The wall shear stress distribution is relatively low and evenly distributed for the entire length at 30^0 taper angle.



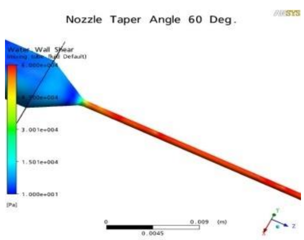
(a)



(b)



(c)



(d)

iJETRM

International Journal of Engineering Technology Research & Management
Published By:
https://www.ijetrm.com/

Fig.Wall shear stress (a) Nozzle angle 15^0 (b) Nozzle angle 30^0 (c) Nozzle angle 45^0 (d) Nozzle angle 60^0

The models were analyzed for the pressure gradient conditions. The analyses of results for 45^0 show that reduced pressure gradient.

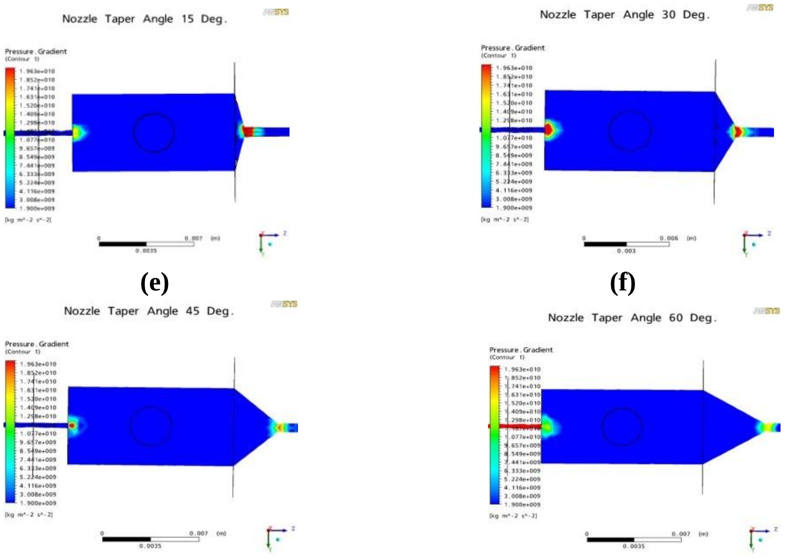


Fig. Pressure gradient (e) Nozzle angle 15^0 (f) Nozzle angle 30^0 (g) Nozzle angle 45^0 (h) Nozzle angle 60^0

CONCLUSION

Thus CFD analysis of flow through nozzle of abrasive fluid jet Machining has been carried out and the following conclusion has been drawn. Loss in kinetic energy has been observed when the flow is along the focus tube .This may be due to some of the abrasive particles do collide with the focusing tube wall.

The kinetic energy loss is relatively less for 45° taper angle. The magnitude of wall shear stress increases when the taper angle increases. The wall shear in the mixing chamber increases sharply after the mixing region.

The energy dissipation due to wall shear is relatively low for 30° taper angle. The pressure gradient is comparatively less for 45° taper angle.

ACKNOWLEDGEMENT

We thank our staffs of Karpaga Vinayaga College of Engineering and Technology, who provided insight and expertise that greatly assisted the research and for assistance and for comments that greatly improved the manuscript. We are expressing our gratitude to our families for being an inspiration. Above all, to God.

REFERENCES

[1] H. Liu, J. Wang, R.J. Brown and N. Kelson , Computational Fluid Dynamic (CFD) Simulation of Ultrahigh Velocity Abrasive Water jet, Key Engineering Materials, Vols. 233-236 (2003), 477-482

[2] B.Jurisevic, D.Kramar, A.Lebar, H.Orbanic and M.Junkar, Modelling and Monitoring Abrasive Water Jet Cutting for Better Process Control, Proceedings of the 37th CIRP International Seminar on Manufacturing Systems, Budapest, Hungary, May 19-21, 2004, pp. 409-415.

[3] Deepak D, Anjaiah D and N Yagnesh Sharma, 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. 2011 Vol III WCE 2011, July 6-8,2011, London, U.K

[4] P K Ray, Member, and Dr A K Paul, Studies on Abrasive Jet Machining, Journal of the Institution of Engineers (India) Vol 68 part -2 November 1987

[5] Bhaskar Chandra, Jagtar Singh, A Study of effect of Process Parameters of Abrasive jet machining, International Journal of Engineering Science and Technology, Vol. 3 No. 1 Jan 2011.