International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-8 Issue-12, October 2019

Flow Simulation in Abrasive Fluid Jet Machining with Water as Carrier Medium using CFD

S.Ramanathan, E.Naveen, K.Vijay, M.J.Campbell Terrin, S.Anish Kisshore

ABSTRACT: ***Tremendous growth is seen in the field Machining of Materials where the engineering applications are highly demandable. Currently Abrasive Fluid Jet Machining is used to machine wide range of Engineering and Structural Materials including Aerospace, Defense and Aircraft Application which require high strength to weight and stiffness to weight ratios. In Abrasive fluid jet machining the abrasives are premixed with a suspended liquid to form slurry. The flow of the Abrasive Fluid mixture through the nozzle, results in rapid wear of the nozzle which degrades the cutting performance. This may lead to the divergence of the Abrasive Fluid Jet such that the angle of impingement varies. This jet impingement angles have more influence on the machining responses than the normal jet impingement angle as the depth of cut is improved thereby minimizing the shallow cuts within the same total cutting time. Nozzle replacement cost plays a vital 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. An analysis was carried out with constant nozzle taper angle and water as carrier medium. The objective of this work is to analyse the effect of inlet operating pressure on wall shear and exit kinetic energy with respect to Carrier medium. The two phase flow analysis was carried by using 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 provided better results.***

KEY WORDS: Abrasive Fluid Jet Machining (AFJM); Computational Fluid Dynamics (CFD)

I. INTRODUCTION

Rapid increase in industrialization and need for Eco-friendly machining generated a demand for machining technique which should be compatible in the area of advanced cutting applications. Machining has almost become part and parcel of every Material processing. Abrasive Water Jet Machining is one of the nontraditional manufacturing processes. In this process, the mechanical energy of water and abrasive is converted to kinetic energy thereby the metal removal takes place in the material.

Revised Manuscript Received on October 30, 2019.
***** Correspondence Author

S.Ramanathan*, Faculty-Department of Mechanical Engineering, Sri Sai Ram Engineering College, Chennai-44

E.Naveen , Faculty-Department of Mechanical Engineering, Sri Sai Ram Engineering College, Chennai-44

K.Vijay, UG Student- Department of Mechanical Engineering, Sri Sai Ram Engineering College, Chennai-44

M.J.Campbell Terrin, UG Student- Department of Mechanical Engineering, Sri Sai Ram Engineering College, Chennai-44

S.Anish Kisshore, UG Student- Department of Mechanical Engineering, Sri Sai Ram Engineering College, Chennai-44

In water jet machining process, only pure water is allowed to enter and it is used to convert kinetic energy thereby machining takes place. Water is pre pressurized upto 400 mpa and forced through a sapphire or tungsten carbide orifice to produce a narrow jet such that the velocity of the jet reaches the velocity of the sound.(1000m/sec). The cutting of soft materials such as rubber , leather, foams, wood, polymers etc are by the erosion of materials by water. While cutting harder materials abrasives are added to the high pressure water jet in the mixing chamber. The precise alignment of water orifice and mixing nozzle makes the abrasive water jet to be convergent but practically erosion of nozzle takes place thereby causing divergence of fluid flow ultimately leading to deterioration of geometrical dimensions. In order to visualize the fluid flow of the AFJM the computational techniques are incorporated before machining such that the flow characteristics can be analyzed in advance. The effect of inlet operating pressure on wall shear and exit kinetic energy plays a vital role in determining the Divergence of the Jet. Many researchers have carried out various Analysis for predicting the Flow characteristics of Abrasive Water Jet. Few of the interpretations from the various Research works are presented.

Liu et.al [1] performed CFD simulation that provided the information of particle velocities and trajectories that determined the particle impact angle, impinging speed and location. **Deepak et.al [2]** observed that abrasive particles moving with the water jet caused severe wall shear of the Nozzle material, thereby altering the nozzle diameter which in turn influenced the jet kinetic energy. By considering this aspect, the effect of inlet pressure on wall shear stress and jet kinetic energy were analysed. The increase in inlet pressure lead to significant increase in the wall shear stress induced. **Ray et.al [3]** observed that the Material Removal Rate (MRR) increases with increase of air pressure and ceases when the pressure reaches a threshold value. **Bhaskar Chandra et.al [4]** studied the effect of process parameters on the material removal rate (MRR) and Exit & Entry diameter of hole obtained were measured. **Andrej Lebar et.al [5]** observed that abrasive water jet (AWJ) machined surfaces exhibited superior surface quality in the upper region and rough surface in the lower zone with pronounced texture marks called striations. **Coray et.al [6]** claimed that there is a vast potential market for high precision parts manufactured by abrasive water jet (AWJ) machining, which calls for improvement of the current AWJ cutting methods. **Dewan et.al [7]** used abrasive water jet for precision cutting in modern technology. The granular abrasive particles were treated a continuous phase.

The air used for pumping the abrasive particles into the jet device was treated as a continuous phase and the water was treated as the principal continuous phase.
The solutions were obtained using the Inter- Phase Slip Algorithm (IPSA). **Kovacevic et.al [8]** developed 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. **Liu et.al [9]** observed that there is a minimum variation of the water pressure and velocity within the computational domain in the jet axial direction. The velocity and pressure variation in the radial direction were not significant within 80% to 90% of the jet diameter. It shown that the CFD simulation can provide the information of particle velocities and trajectories which made it possible to determine the particle impact angle as well as the impinging speed and location. **Ramanathan et.al [10]** made a study about analysis of 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 using CFD Analysis. Thus the above wrote literatures have been reviewed to access the flow characteristics of AFJM. From the literature the following points would be considered:

- Existing nozzle angle of 30° was compared with 15°, 45° and 60°and found that 45°C will be an optimized one.
- Water would be used as working fluid.

The objective of the work is as follows:

- To analyse the flow characteristics of abrasive water jet on the inside surface of the nozzle
- To optimise nozzle dimensions and process parameters to minimise the nozzle wear with

Inlet pressure of water jet.

II. CFD ANALYSIS

2.1 MODELING

Modeling was done using Pro E Wild Fire 2.0 and exported in IGES format. The models of nozzle heads with varying taper angle have been shown in the following figures 1 to 4.

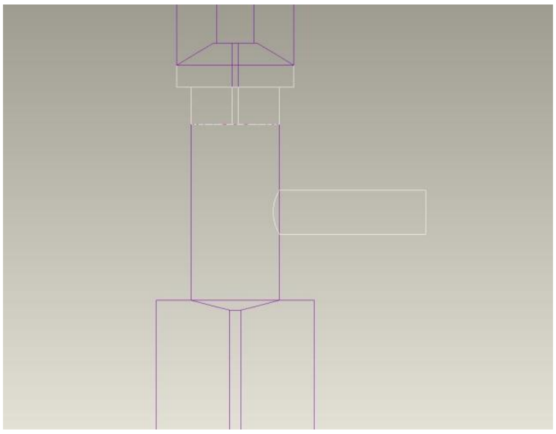

Fig 1. 15 deg. Taper Angle

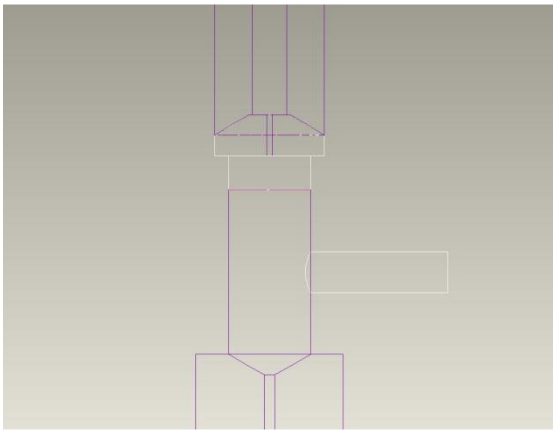

Fig 2. 30 deg. Taper Angle

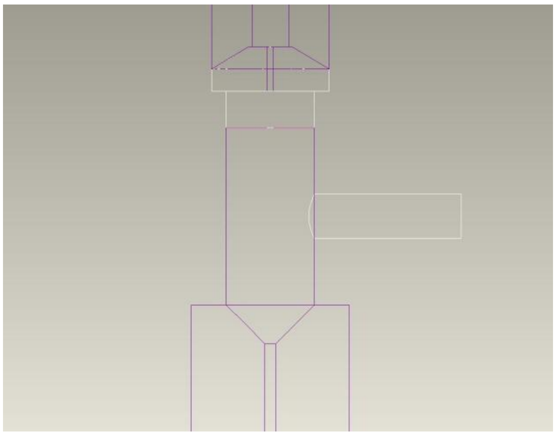

Fig 3. 45 deg. Taper Angle

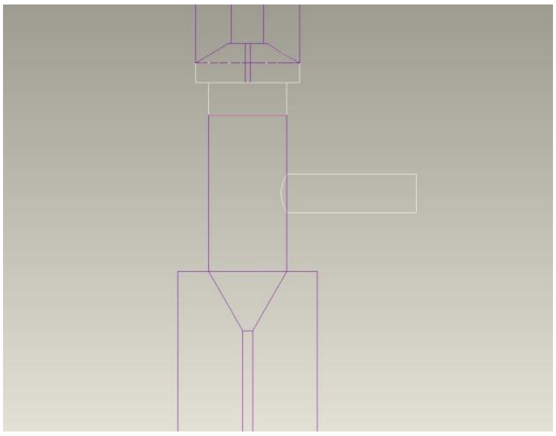

Fig 4. 60 deg. Taper Angle

The dimensions of the Nozzle head are as follows:

Focus tube (Mixing tube) Diameter	: 0.76 mm
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

2.2 MESHING

ANSYS ICEMCFD's mesh generation tools offer the capability to parametrically create meshes from geometry in numerous formats:

i) Unstructured hexahedral Unstructured tetrahedral
ii) Cartesian with H-grid refinement Hybrid Meshes comprising hexahedral, tetrahedral, pyramidal and/or prismatic elements
iii) Quadrilateral and triangular surface meshes.

Figure 5 to 8 shows the meshed model of the Nozzle head with various angles.

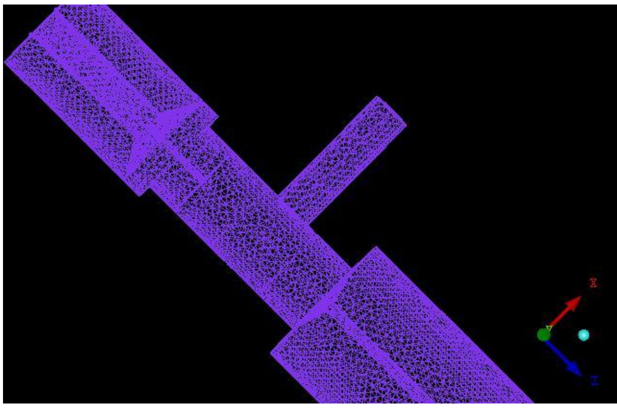

Fig 5. 15 deg. Taper Angle

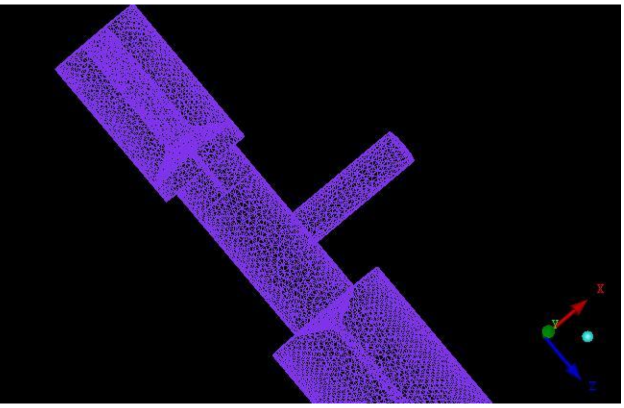

Fig 6. 30 deg. Taper Angle

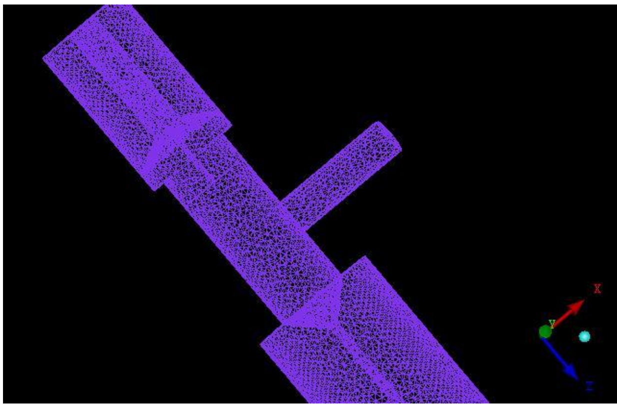

Fig 7. 45 deg. Taper Angle

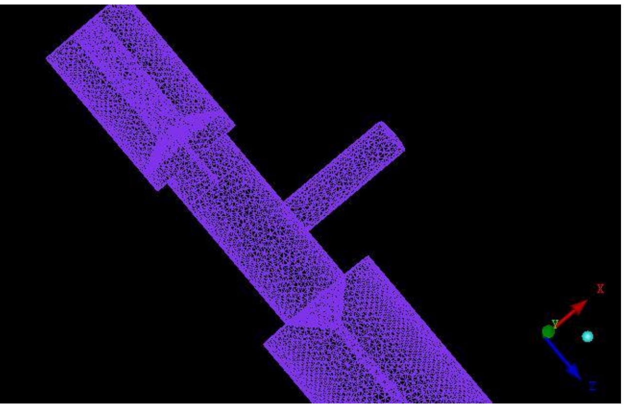

Fig 8. 60 deg. Taper Angle

III. Properties of the Carrier Medium

Table 1 shows the properties of water at various heads.

Table 1: Properties of Water at various pressures:

S.No	Inlet Pressure of water , MPa	Velocity of water, m/s	Mass flow rate of water, kg/min	Mass flow rate of abrasive particles kg/min	Velocity of water abrasive mixture, m/s
1	40	282.84	0.533	0.45	153.36
2	80	400	0.753	0.45	250.49
3	120	490	0.918	0.45	328.81
4	200	632.45	1.192	0.45	459.13
5	300	774.59	1.46	0.45	592.1

IV. SOLVER STAGE

After defining all the conditions the model is imported in CFX- Solver Module in .def format for doing iterative calculations and to generate result file.

The following solver control parameters have been specified in CFX- Pre Module.

Solver Control parameters

Number of iterations : 100
Time scale control : Auto Time scale

Residual Target : 1e-4

V. RESULTS AND DISCUSSION

5.1 Effect of velocity variation of Water with respect to inlet pressure:

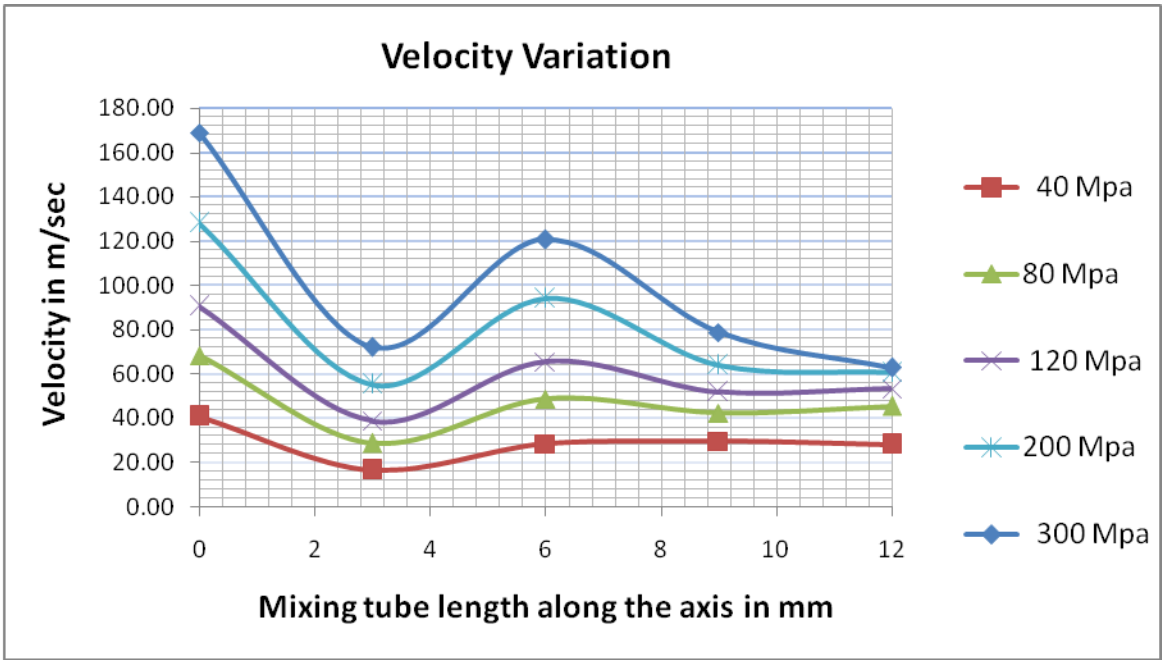

Fig 11 Mixing Chamber

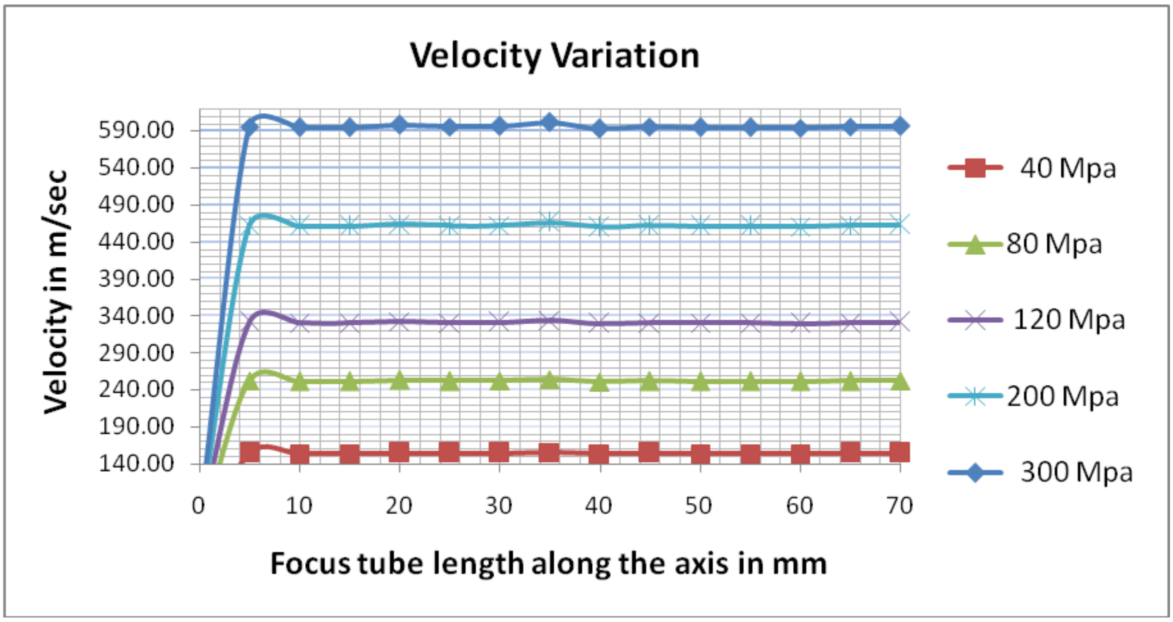

Fig 12 Focus tube

Figure 11&12 shows the velocity variation along the mixing and focus tube length. In the case of mixing tube the velocity reduces gradually and it reaches maximum in the mixing region and then gradually decreases when it reaches at the end of tube. The magnitude of velocity is high at higher water inlet pressure. In the case of focus tube length, the gain in velocity is observed when the flow past the nozzle. The velocity gain is more for 300 MPa pressure. Further kinetic energy lost is observed when the flow is along the focus tube for all the cases. This may be due to some of the abrasive particles do collide with the focusing tube wall.

5.1.2 Wall Shear Distribution

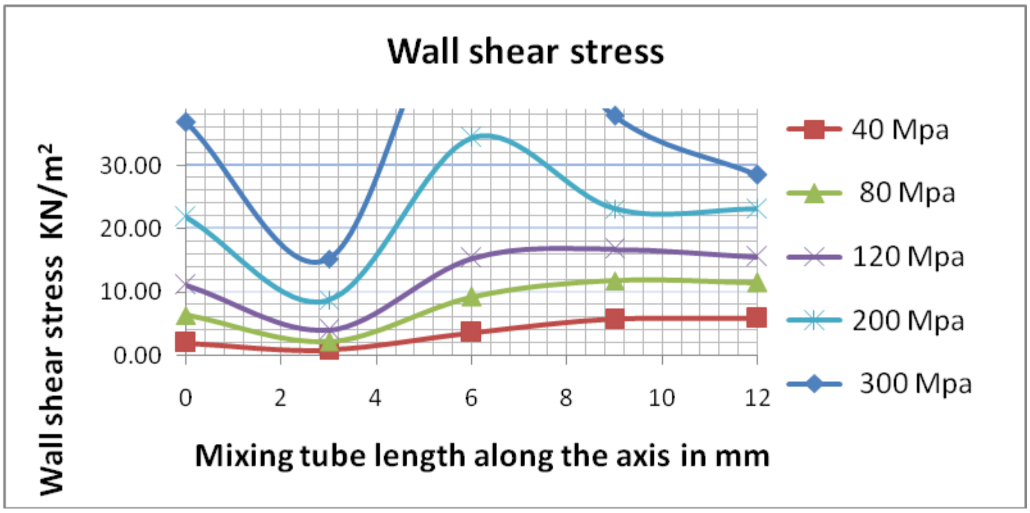

Fig 13 Mixing Chamber

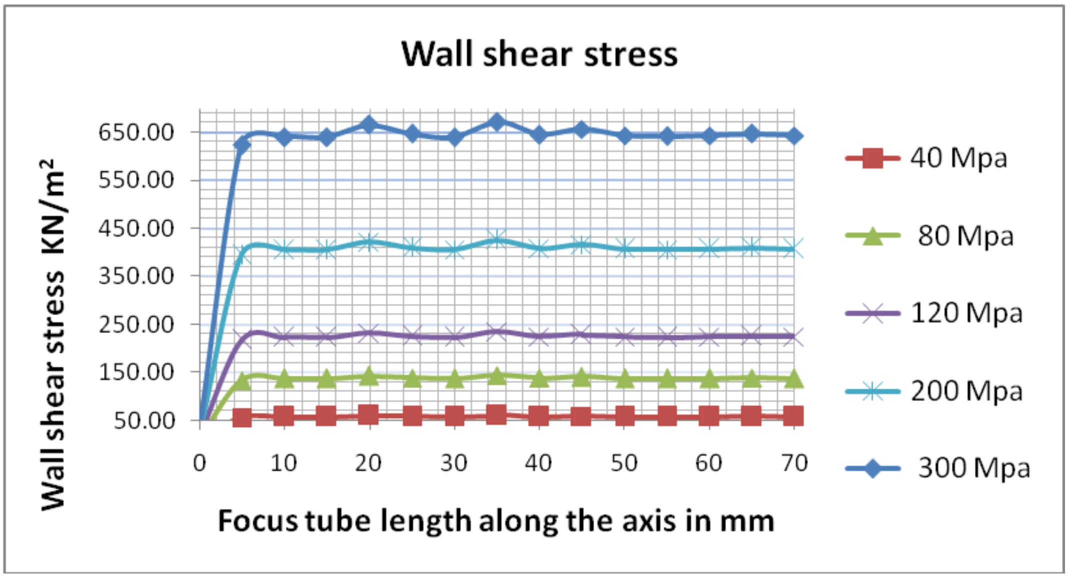

Fig 14 Focus tube

Figure 13&14 shows the wall shear stress distribution along the mixing and focus tube length. Figure 13 shows that increased wall shear for at higher water inlet pressure. Wall shear increases maximum at the mixing region and it decreases along its flow. It has been observed from the graph that when the water pressure increases the magnitude of wall shear increases sharply in the mixing region. The wall shear is relatively low in the case of 40 MPa water pressure. Fig shows 14 that the wall shear increases when the inlet water pressure increases. The wall shear is relatively uniform along the focus tube length at water pressure of 40 MPa and it varies when the inlet pressure increases.

5.1.3 Energy Dissipation

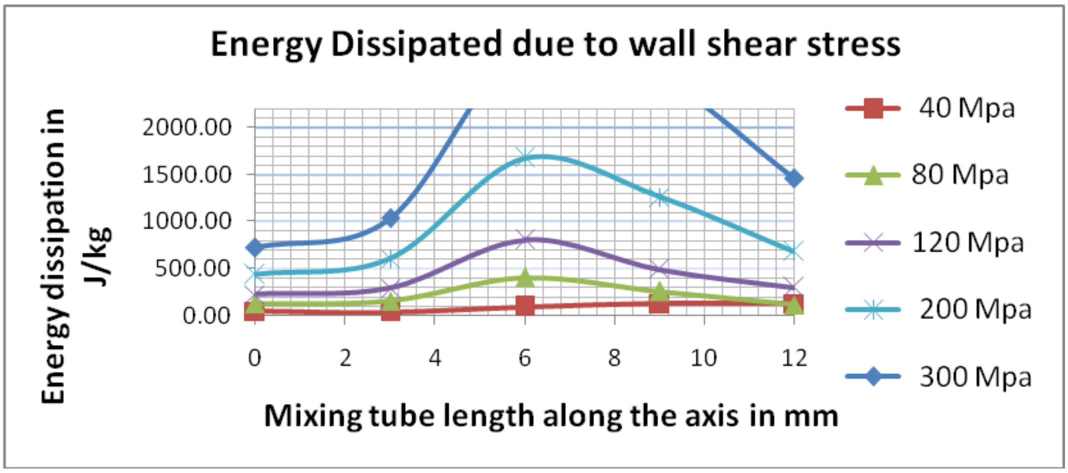

Fig 15 Mixing Chamber

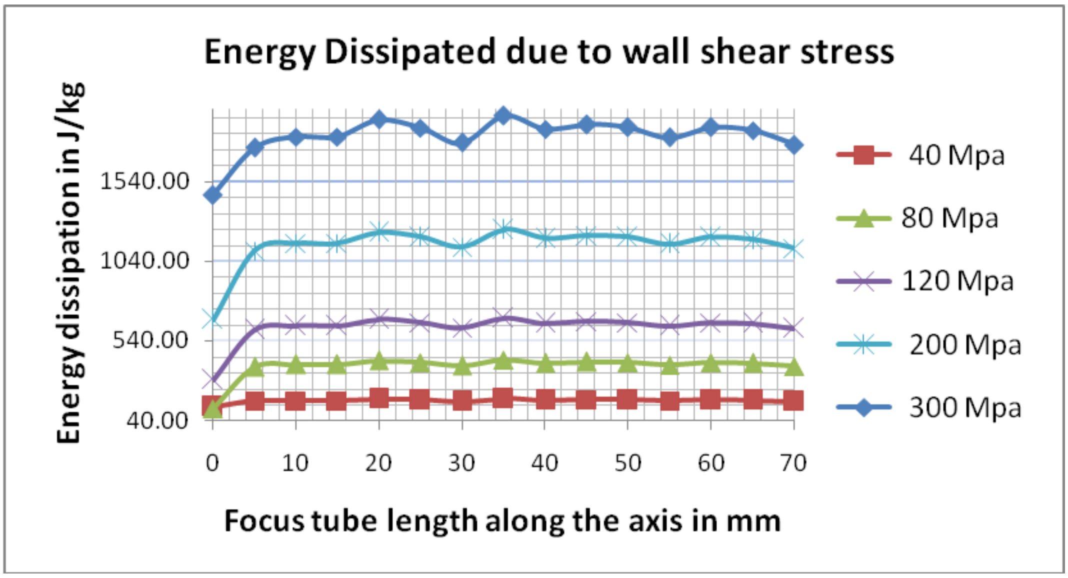

Fig 16 Focus tube

Figure 15&16 shows the energy dissipation due to wall shear along the mixing and focus tube length. Figure 15 tube shows that the magnitude is low at the entry of mixing tube and it increases till the mixing region for all the cases. It decreases gradually when it reaches the end of the tube. The energy dissipation is significantly high at higher water inlet pressure.Figure 16 shows energy dissipation increases when the flow past the nozzle it is relatively uniform for 40 Mpa and 80 Mpa water pressures. When the water pressure increases the variation (fluctuation) is high.

6.1.4 Pressure Gradient

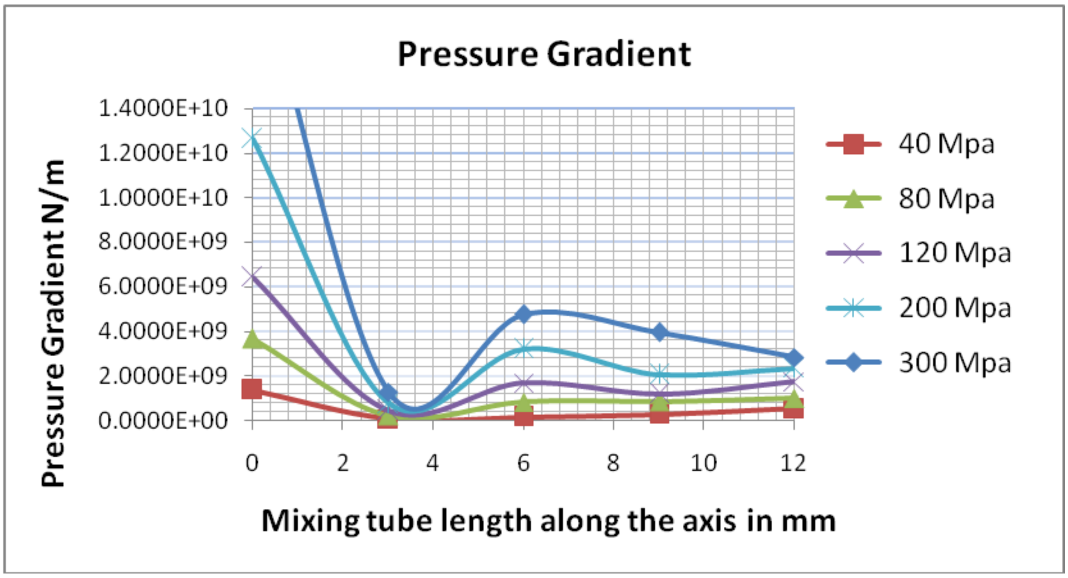

Fig 17 Mixing Chamber

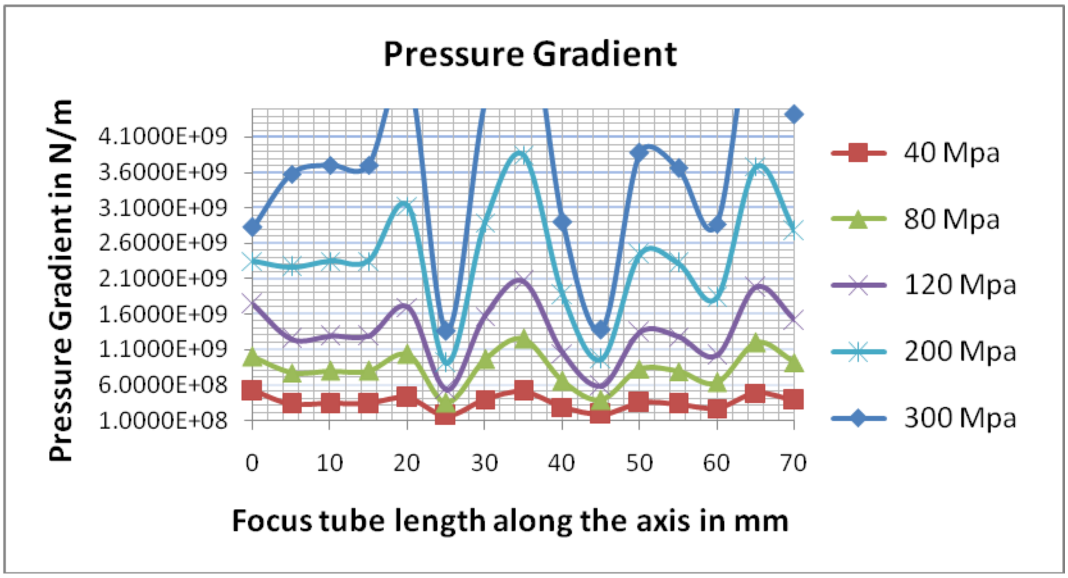

Fig 18 Focus tube

Figure 17& 18 shows the pressure gradient along the mixing and focus tube length. The pressure gradient in the flow causes the axial velocity to decrease and cause for eddies which will consequently increase the energy dissipation. Figure 17 shows that the pressure gradient decreases sharply towards the mixing region and it increases gradually towards outlet of mixing tube. It has been observed from the figure 18 that the pressure gradient is comparatively very high at the entry for 300Mpa. Figure 18 that increased pressure gradient along the focus tube for 300 Mpa. It is also observed from fig (focus) that the pressure gradient is relatively low at lower water pressure.

VI. CONCLUSION

Thus CFD analysis of flow through nozzle of abrasive fluid jet Machining has been carried out and the following conclusion has been drawn.

Effect of Nozzle Angle

- 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.

Effect of Inlet Water Pressure

- The velocity gain is more at higher water pressure. Further kinetic energy lost is observed when the flow is along the focus tube for all the cases. This may be due to some of the abrasive particles do collide with the focusing tube wall
- The wall shear increases when the inlet water increases. The distribution of wall shear is relatively uniform along the focus tube length at lower water pressure and it varies significantly when the inlet pressure increases.
- The energy dissipation and shear strain rate is significantly high at higher water inlet pressure.
- Pressure gradient is relatively low at lower water pressure

REFERENCE

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. 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
3. 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
4. Bhaskar Chandra, Jagtar Singh, A Study of effect of Process Parameters of

a. Abrasive jet machining , International Journal of Engineering Science and Technology, Vol. 3 No. 1 Jan 2011

5. Andrej Lebar, Mihael Junkar Simulation of abrasive water jet cutting process:Part 1. Unit event approach, Modeling And Simulation In Materials Science And Engineering, (2004).
6. Coray, Jurisevic, Junkar, Heiniger, Measurements on 5:1 Scale Abrasive Water Jet Cutting Head Models, University of Applied Sciences, Switzerland
7. Dewan Ahmed, Elias Siores, Numerical Simulation of Abrasive Water Jet for Taper Inlet Angles, 14th Australasian Fluid Mechanics Conference Adelaide University, Adelaide, Australia, (2001)
8. Kovacevic Turbulent solid–liquid flow through the nozzle of premixed abrasive water jet cutting systems, Southern Methodist University, USA
9. Liu,Wang, Brown and Kelson, Computational Fluid Dynamic (CFD) Simulation of Ultrahigh Velocity Abrasive Water jet, Key Engineering Materials, pp 233-236 (2003).
10. S. Ramanathan, J. Jeykrishnan, J. Sylvester Diaz, M. Sarath, N. Sriram, V. G. Ramanujam, C. Pratheep, S. Caleb Stevenson and B. Karthik, Analysis of Taper Angle in Nozzle in Abrasive Fluid Jet Machining using CFD, International Journal of Mechanical Engineering and Technology 9(2), 2018. pp. 271–279.

AUTHOR PROFILE

Mr. S. Ramanathan working as Assistant professor at Sri Sairam Engineering College having a vast working experience of 10 years in the field of teaching. CFD, ANSYS, FEA, Robotics are the area of specializations.

Mr. E. Naveen working as Assistant Professor at Sri Sairam Engineering College having a working experience of 4 years in the field of teaching and research. Manufacturing, Materials Science and Industrial engineering are his area of specializations.

Mr.K.Vijay is an Undergraduate Student of Mechanical Engineering at Sri Sairam Engineering College

Mr.M.J.Campbell Terrin is an Undergraduate Student of Mechanical Engineering at Sri Sairam Engineering College

Mr.S.Anish Kisshore is an Undergraduate Student of Mechanical Engineering at Sri Sairam Engineering College