 ISBN: 978-93-84339-38-8

Numerical simulation for hydrodynamic characteristics of nozzle used in water jet machining

Sourabh Jain
Assistant professor,
Navrachana university

Rachit Rathi
Assistant Professor
BITS, Vadodara

Bharat Sunar
Assistant professor
GCET, Vallabh Vidyanagar

Deepak S Jani
Lecturer
EME , Vadodara

Abstract*—For manufacturing of any product machining is required to give final shape. Conventional and Non-conventional machining process are available. But non-conventional machining technologies such as EDM, laser, plasma, water jet, abrasive water jet are very popular now a days because of its high machining versatility, minimum stresses on the work piece, high flexibility no thermal distortion, and small cutting forces. Water jet machining is one of non-conventional machining process used for production process. Performance of water jet machining depends on various parameter but material removal depends upon design of water jet nozzle. Practically, it is difficult to understand internal flow characteristics of liquid used for machining and hence with the help of numerical simulation parameters affecting machining process can be studied. Numerical simulation is CFD analysis, which plays a vital role to understand flow characteristics within nozzle geometry and outside nozzle, which helps to design pure water jet machining phenomena. The present study is to understand effect of high velocity jet on flat surface. Pressure, velocity and stream line has been obtained by simulating problem in commercial CFD tool ANSYS-Fluent. Results showed that a velocity of 570 m/sec can be reached for pressure up to 160 MPa.

Keywords—Water jet, CFD, Nozzle, Jet velocity

I. INTRODUCTION

The major problem using mechanical conventional cutting tool is their excessive tool wear due to the direct contact of the tool with work piece which resulted in quick rise of cutting temperature. Worn out tools produce high cutting forces, poor surface finish and eventually results in large processing costs. This problem can be eliminated by water jet. Water jet (WJ) is a generic term used to describe machining process in which a high velocity stream jet of water used for cutting or cleaning purposes. This process is similar as water erosion found in nature but highly accelerated and concentrated. High-pressure water is thrown through a small hole (generally called orifice to concentrate an extremely high amount of energy from a small area: so pressure energy is converted in kinetic energy in the capillary restriction of the tiny orifice, creating a high-velocity beam of water. Water jet cutting is simple and flexible technology, even if often it suffers a lack of high precision, especially if it is compared with laser cutting. In order to gain competitiveness and satisfy quality requirements, water jet cutting required more systematic insight of its process aiming to achieve awareness on the physics and the main causes of the disturbances which are systematically affecting the jet. Furthermore, because of the high velocities gained by the jet and the rapid dynamics in conjunction with the small characteristic dimensions of the process, quantitatively correct experiments and the empirical correlations are often very difficult: it is then obvious that numerical simulations would be an invaluable and promising mean to reach a better understanding of the process and optimize such a complex unsteady turbulent two-phase micro flow. The aim of the present work is to perform a CFD numerical simulation of a pure water jet in order to investigate its creation and stability achieving a better understanding of the process and its disturbances.

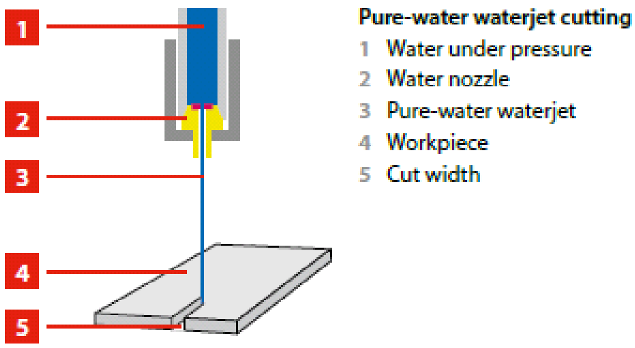

Fig. 1 Water Jet Machining

Research on numerical simulations of high speed water jets are limited; and also not good agreement with validation. Rajaratnam *et. al* [1] found the actual flow physics experimentally then after Liu *et al.* [2] taken problem of abrasive water jet (AWJ) and tried to solve, but their results were not validated with experimental results. Volume of fluid multiphase model used for simulation in FLUENT. Actually this model (or any multiphase model) is not able to capture the air entrainment process. So the results of Liu *et al.* [2] fail to simulate. It is worth mentioning here that numerical simulation of high speed jets faces significant challenges. Lin *et al.* [8] found that the process of jet breakup and subsequent mist formation not only depends on the thermodynamic states of liquid jet and the ambient air but it also depends on the

ISBN: 978-93-84339-38-8

nozzle internal flow characteristics like nozzle cavitation, turbulence, etc. Guha *et al.* [3] created a semi-empirical model to capture the process of air entrainment. Based on empirical relations, this model predicts the interaction between air and water phases. This interaction term is then used into the governing Navier-Stokes equations as source term. They validated their simulations with the experimental work of Rajaratnam *et al.* [9] Yoon *et al.* carried out experiments with low, moderate and high speed jets. They found that the hydrodynamic instability and subsequent break-up mechanism of high speed jets is significantly changing from the better understood Rayleigh type mechanisms of the low/moderate speed jets. So this is theoretical understanding of mechanisms related to high speed water jets is still not well developed.

II. PROBLEM DEFINATION

A. Geometry

Computational fluid dynamic is very strong tool for understanding of flow behavior of internal or external flow phenomena. Various flow properties can be evaluated by using commercial available CFD tool. In present study ANSYS-Fluent used to simulate water jet problem. Since aim of this study is to understand the effect of high velocity jet on flat surface. Formulation of problem has been done such that it involves internal flow within jet nozzle and outer flow near to the flat surface. Information obtain can be utilized to find out machining capability of water jet for particular case. Water jet mainly consist nozzle with small orifice at outlet. Fig. 2 show simplify water jet nozzle with all dimension.

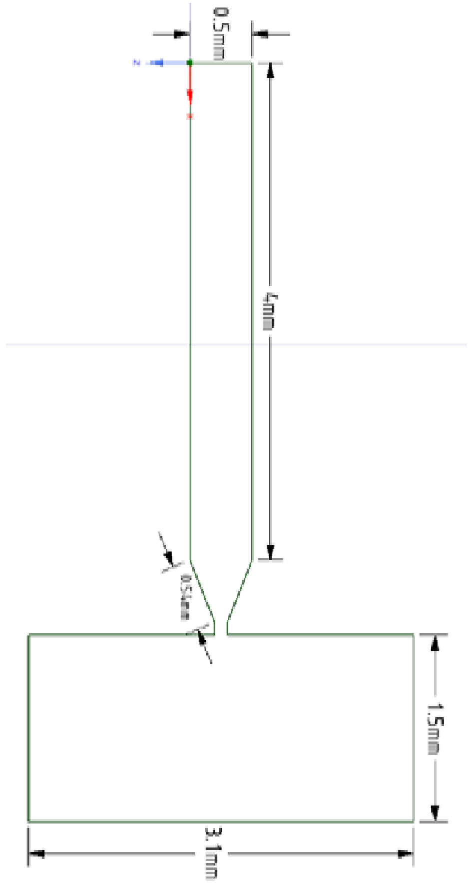

Fig. 2 Schematic diagram of Water jet nozzle

B. Mesh generation & Boundary conditions

Computational model has been generated in commercial available tool called Space claim part of ANSYS-workbench. After generate model, entire fluid domain should be divided in small number of finite volume. Solution has been obtained by applying continuity and momentum equation to each finite volume. However computational results depend on grid size of the finite volume. But fine grid has computational cost. It is very much required to perform grid independence test before going for parametric study shown in Table-1. The mesh is generated in a more uniform way, and it has been refined in Fluent with mesh adaption method. The purpose of mesh adaption is refining the mesh in the regions of the domain where important flow features take place. The most interesting regions to be adapted are the capillary region.

The total amount of mesh elements at the end of the simulation is around 38725, which is almost the triple than the elements of the first original mesh. This obviously results in a long CPU time which is however unavoidable in order to obtain precise enough solutions.

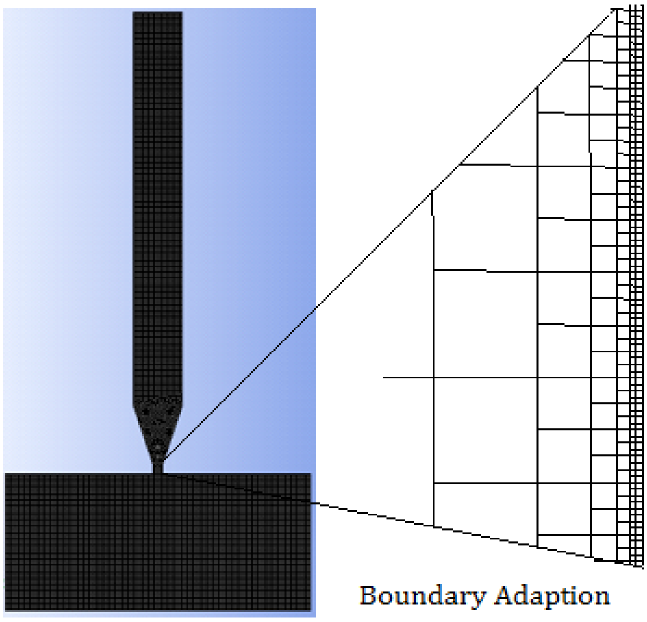

Fig. 3 Mesh elements with grid adaption technique

TABLE-1 Effect of mesh elements on maximum velocity

ISBN: 978-93-84339-38-8

Sr. No	Number of Elements	Maximum velocity (V) (m/s)
1	12354	579.5
2	25444	571.5
3	38725	570.6
4	54004	570.2

After generate mesh, boundary condition has been assigned to problem. At inlet of nozzle pressure inlet is assigned. In present case there are pressure outlets are defined shown in Fig. 4. It is defined as pressure out let boundary condition, which is nothing but atmospheric pressure. Remaining all boundary defined as no slip boundary i.e. wall. Solution of the problem has been investigated in FLUENT. The flow is assumed steady and incompressible. Simulation has been carried out using water with injection pressure of 160MPa. Outlet pressure is considered as zero gauge or atmospheric pressure.

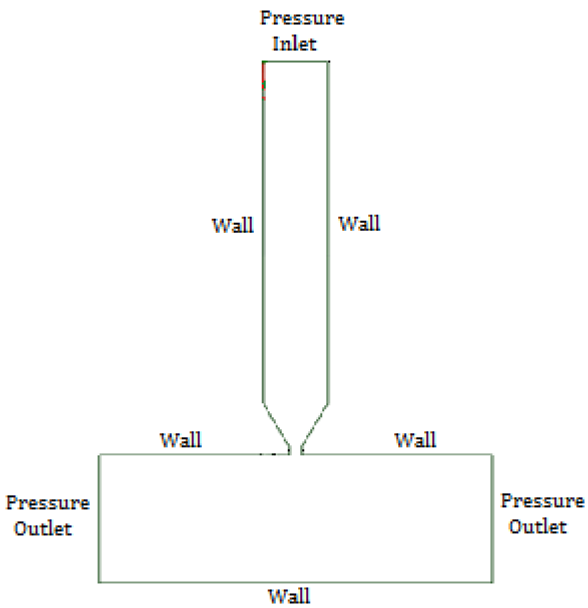

Fig. 4 Boundary conditions

C. *Solver setting & Governing equations*

The present section collects all the settings related to the numerical methods employed by the solver. Simulation has been carried out with pressure base solver. Problem is assumed to be steady state flow. Only continuity and momentum equation are taken in to account to solve present problem. For incompressible flow density is constant. Accordingly we have,

Continuity equation:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad (2)$$

X-momentum equation:

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) + \rho g_x \qquad (3)$$

Y-momentum equation

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) + \rho g_y \qquad (4)$$

TABLE-2 Solver setting

Solver Setting:	
Pressure-Velocity coupling	PISO
Neighbor correction	3
Skewness correction	0
Skewness-Neighbor coupling	No
Discretization schemes:	
Pressure	PRESTO!
Momentum	2nd order
Volume Fraction	Geo reconstruct
Turbulent kinetic energy (k)	2nd order
Turbulent dissipation rate	2nd order

III. Results and Discussion

In present study effort has been made to validate simulation results with analytical or experimental results. Validation of numerical models is always challenging when dealing with WJ applications: experiments on this subjects are very rare and, if existing, very limited concerning detailed quantitative data. Quantitatively correct experiments are almost impossible since the physics and the dynamics inside a WJ orifice involve phenomena characterized by small length scales (in the order of few millimeters or less) together with very high velocity scales (from 500 up to 900 m/s) and rapid time scales (in the order of milliseconds). Since water is considered incompressible in the numerical model developed, the theoretical velocity of the formed water jet can be calculated from Bernoulli's equation:

$$v_{th} = \sqrt{\frac{2\,p_{up}}{\rho}} = \sqrt{\frac{2 \cdot 160E{+}06}{998.20}\,\frac{\text{Pa}}{\text{kg/m}^3}} = 566.20\ \text{m/s} \qquad (5)$$

When high pressure water flow through converging nozzle pressure energy converts in to kinetic energy, it leads to very high velocity at the outlet of nozzle. Figure 5 and 6 shows velocity and pressure contour inside Water jet nozzle. High velocity jet strikes on the top surface which ultimately acts as cutting force for machining process. Referring to Fig. 5, the numerical model gives a simulated velocity $V_{simulation}$ of 570.6 m/s. The percentage error is around 0.8%, which means that the two data seem to be in perfect accordance. Fig. 6 clearly indicate high pressure water jet impinge on flat surface with pressure value of 155 MPa. Due to high density with high pressure create large amount of energy impinge surface which can be used for machining purpose. Fig. 7 shows path line of water jet. High velocity jet coming out from nozzle orifice and becomes zero at center of flat surface. Large recirculation is appear on both side of water jet.

 ISBN: 978-93-84339-38-8

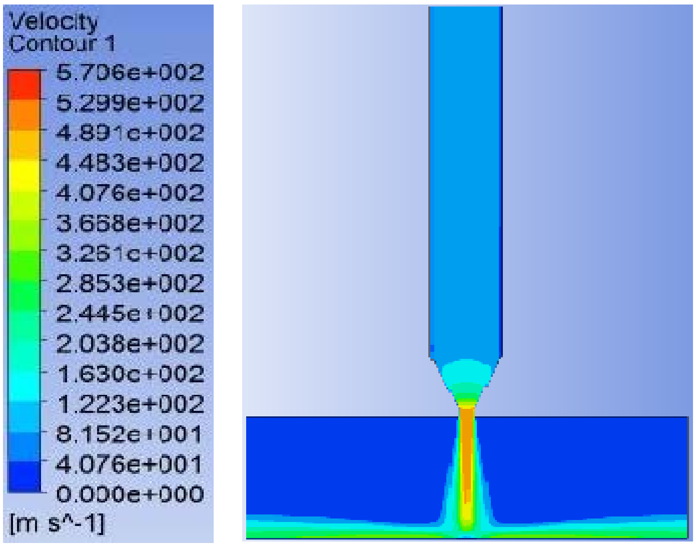

Fig. 5 Contours of velocity magnitude

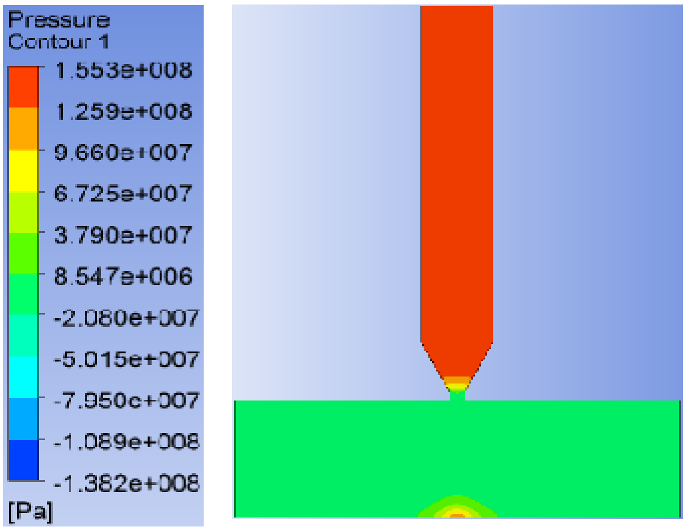

Fig. 6 Contours of pressure magnitude

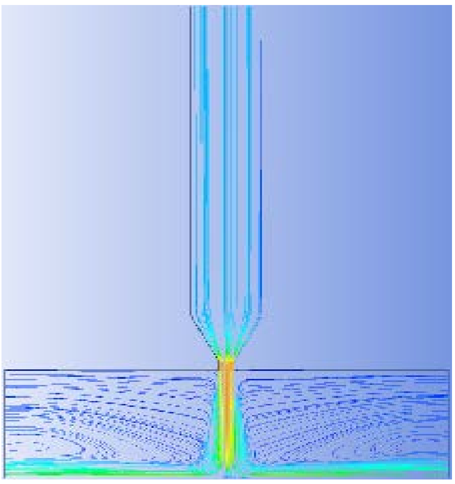
Fig.7 Path line of water jet

IV. Conclusion

Numerical simulation using CFD is the best tool to capture any physical phenomena which occurs at micro level to understand the physics of problem. In actual phenomena where measurement of parameter such as pressure, velocity need measuring instruments. Here, one can able to identify the actual pressure and velocity at each and every point through color graphs. Results obtained using CFD are in good agreement with analytical equations with error of not more that 0.8%. This shows about accuracy with which CFD works. Flow velocity reach up to 570 m/s, at inlet pressure of 160 MPa resulting into removal of material at the impingement of water jet with work piece. As there is no tool involved in machining of work piece, removal of material can be continuously carried out without any break for replacing tool. At such high velocity one should study about stability of fluid flow and its behavior.

References

[1] Rajaratnam, N., Steffler, P.M., Rizvi, S.A.H., Smy, P.R., Experimental Study of Very High Velocity Circular Water Jets in Air. Journal of Hydraulic Research, 32(3), 461-470,1994.

[2] Liu, H.,Wang, J.,Kelson, N.,Brown, R.J. A study of abrasive waterjet characteristics by CFD simulation. Journal of Materials Processing Technology, 153-154, 488-493, 2004

[3] Guha, A., Barron, R. M., and Balachandar, R., Numerical Simulation of High Speed Turbulent Water Jets in Air. Journal of Hydraulic Research, 48(1), 119-124, 2010.

[4] Hashish, M., and duPlessis, M. P.,Theoretical and Experimental Investigation of Continuous Jet Penetration of Solid. ASME Journal of Engineering for Industry, 100, 88–94, 1978.

[5] Hashish, M., and duPlessis, M. P. Prediction Equations Relating High Velocity Jet Cutting Performance to Standoff Distance and Multipasses. ASME Journal of Engineering for Industry, 101, 311–318, 1979.

[6] Leach, S. J., Walker, G. L., Smith, A. V., Farmer, I. W., Taylor, G., Some Aspects of Rock Cutting by High Speed Water Jets. Philosophical Transactions of the Royal Society of London, 260 (1110), 295-310, 1966.

[7] Leu, M.C., Meng, P., Geskin, E.S., Tismeneskiy, Mathematical Modeling and Experimental Verification of Stationary Water Jet Cleaning Process. Journal of Manufacturing Science and Engineering, 120(3), 571-579, 1998.

[8] Lin, S.P., Reitz, R.D, Drop and Spray Formation from a Liquid Jet, Annual Review of Fluid Mechanics, 30, 85–105, 1998.

[9] Rajaratnam, N., Albers,C., Water Distribution in Very High Velocity Water Jets in Air. Journal of Hydraulic Engineering, 124(6), 647-650, 1998.

[10] A. Tomiyama A. Sou, S. Hosokawa, “Effects of cavitation in a nozzle on liquid jet atomization”, International Journal of Heat and Mass Transfer, (50):3575–3582, 2007.

[11] B. Pourdeyhimi N. Anantharamaiah, H. Vahedi Tafreshi, “Numerical simulation of the formation of constricted water jets in capillary nozzles: effects of nozzle geometry”, Chemical Engineering Research and Design, 84(A3):231–238, 2001

[12] Francesco Arleo, “Numerical simulation of a pure water jet inside an orifice: jet stability and effects of droplets collisions”, Master of Science Thesis, San Sebastian – Milano Academic year 2009/2010.

[13] Mehul Bambhania, Nikul Patel, “Numerical simulation to predict cavitation inside diesel engine fuel injector nozzle”, International conference on Multidisciplinary Research Approach for the accomplishment of academic excellence in Higher technical education through industrial practices, Pattaya-Bangkok, June 2016.

[14] Jignesh T. Dave, Mehul P. Bambhania “Numerical simulation for formation of water jet in pure Water jet machining process”, International conference on futuristic trends in engineering,science,pharmacy & management, organised by A D Publication, vadodara, december-2016.