Numerical Analysis of Mixed Fluid Jet Flows through Cutting Fluid Supplying Nozzle

S. CHUNG[a], B. SHIN[b]

Abstract

Metal cutting operation involves generation of heat due to friction between the tool and the cutting materials. This heat needs to be carried away otherwise it creates white spots. To reduce this abnormal heat cutting fluid is used. Cutting fluid also has an important role in the lubrication of the cutting edges of machine tools and the pieces, and in sluicing away the resulting swarf. As a cutting fluid, water is a great conductor of heat but is not stable at high temperatures, so to improve stability an emulsion type mixed fluid with water and oil is often used. It is pumped over the cutting site of cutting machines as a state of atomized water droplet coated with oil by using jet flow. In this paper, to develop cutting fluid supplying nozzle with ultra-thin oil film coating of water droplet, a numerical analysis of three dimensional mixed fluid jet flows through multi-stage nozzle was carried out by using a finite volume method. Jet flow characteristics such as nozzle exit velocity, development of mixing region, re-entrance and jet intensity were analyzed. Detailed mixing process of air, water and oil in multi-component fluid were also investigated. And the important flow information for advance design of cutting fluid supplying nozzle was drawn.

Keywords: Numerical analysis, Multi-stage nozzle, Oil film coating system, Jet flow characteristics, Multi-component fluid mixed jet

1. INTRODUCTION

During the cutting process of metal and nonferrous metals, high heat and pressure happen between cutting tool and material as well as cutting chips[1-2]. To bring down the high heat, cutting oil is used as a coolant. However it would give rise to serious problems of dermatitis, respiratory organ obstacle, oncogenesis and environmental pollution[3].

To reduce these problems, cutting fluid supplying apparatus which use extremely small amount of cutting oil by coating technology to make oil film over water particle is developed. Nozzle is used to jet cutting fluids in the apparatus[4]. In the development of high reliable and efficient cutting fluid supplying apparatus with jet technology, application and consideration of fluid dynamics are very important[5-8].

In this paper, to develop cutting fluid supplying nozzle with ultra-thin oil film coating to reduce environmental pollution, a CFD simulation of 3-D (three-dimensional) mixed fluid jet flows through multi-stage nozzle was carried out by using a CFX 10 Code[9]. First, through the calculation of top nozzle (the front part of multi-stage nozzle) hydromechanics behavior of simple shaped nozzle is investigated. Then, the whole complicated multi-stage nozzle is computed to investigate mixing status of air, water and oil and hydromechanics characteristics. Detailed jet flow mechanism such as nozzle exit velocity, development of mixing region, re-entrance, jet intensity and flow pattern in multi-stage nozzle are simulated.

a) Former Master Student, Dept. of Mechanical Engineering, CFD Analysis Group, Taisung Esene, Co. Ltd.
b) Professor, Dept. of Mechanical Design Systems Engineering

2. NUMERICAL ANALYSIS

The flow through the multi-stage nozzle was assumed by incompressible turbulence flow. Finite volume method with second order resolution scheme and k-ε turbulence model is used.

Fundamental equations are the continuity equation and the Reynolds Averaged Navier-Stokes equations as follows;

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i) = 0 \tag{1}$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_j} + \left[\mu_{eff}\left(\frac{\partial u_i}{\partial x_j}\frac{\partial u_j}{\partial x_i}\right)\right] = 0 \tag{2}$$

where ρ, u and p are fluid density, velocity and static pressure. t is time and μ_{eff} is effective viscosity considered molecular viscosity and turbulent viscosity. In this paper, the turbulent viscosity was estimated by solving the turbulence kinetic energy and dissipation equation.

Figure 1 shows a schematic diagram of the multi-stage nozzle which is actual model designed by

Hantech I & P Co., Ltd[10]. It consists of the top nozzle, water and oil spray nozzle, water drop chamber and oil forming chamber.

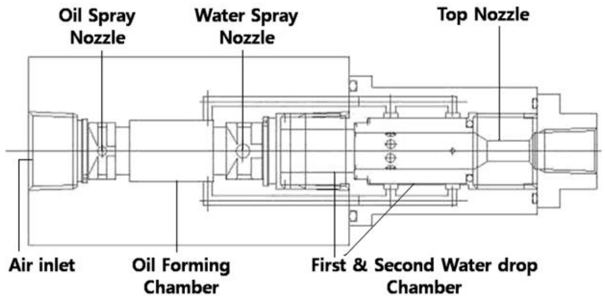


Fig. 1. Schematic diagram of multi-stage nozzle .

2.1 Computational grid

Figure 2 shows computational grid generated by prism and unstructured type mesh for simple circular shaped top nozzle flow. In this figure, the left-hand side represents top nozzle which is designed by convergent-divergent nozzle with consideration of jet intensity and the scattering of cutting fluid. The right-hand side shows a free-jet region to investigate development of jet flow and velocity distribution.

Figure 3(a) shows a solid model of whole multi-stage nozzle. Figure 3(b) shows an example of unstructured computational grid around the water drop forming chamber which is one of main mixing parts of cooling fluids. To obtain ultra-thin oil film to be coated on the surface of tiny water drop, it consists of very complicated ring type and circumferential several passages as seen these figures.

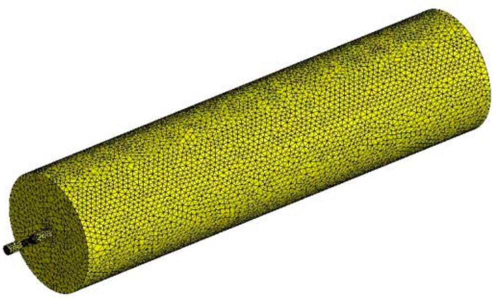
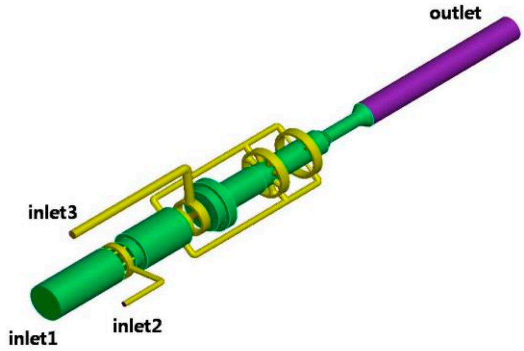


Fig. 2. Computational grid of top nozzle.

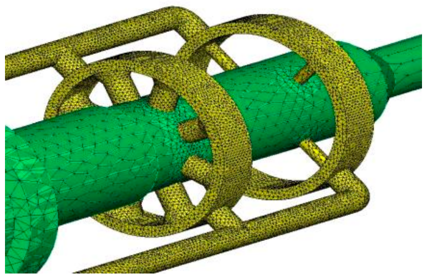
2.2 Boundary condition

A fixed inlet velocity condition based on the designed outlet velocity of top nozzle and atmospheric pressure at nozzle exit were imposed. The working fluid of water of 25°C is released by free jet flow in the exit of the nozzle.

On the other hand, regarding the boundary condition of the three inlets of air inlet (inlet 1 in Fig.3(a)), water spray nozzle (inlet 3) and oil spray



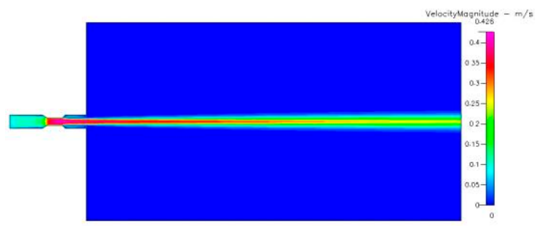
(a) Solid model of multi-stage nozzle.



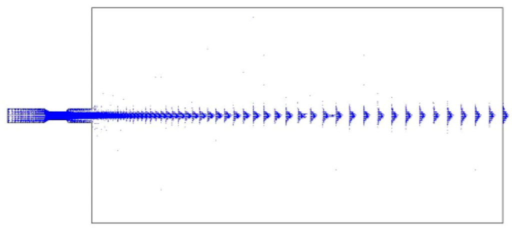
(b) Water drop forming chamber.

Fig. 3. Solid model and computational grid around water drop forming chamber.

(a) Pressure distribution.



(b) Velocity magnitude.



(c) Velocity vectors.

Fig. 4. Computational results of top nozzle.

nozzle (inlet 2) in multi-stage nozzle, a fixed velocity of 0.03m/s, 0.04m/s and 0.0177m/s respectively are imposed according to the actual operation condition.

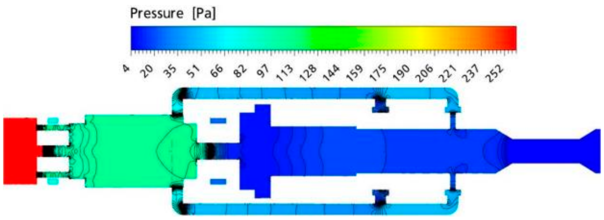
3. NUMERICAL RESULTS

3.1 Top nozzle

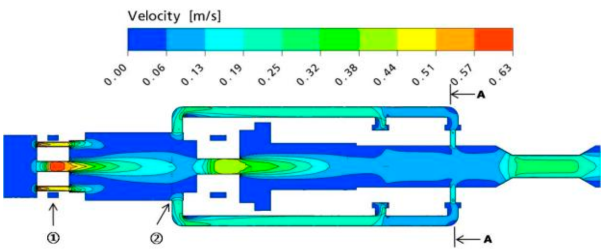
Figure 4 shows numerical results of pressure and velocity magnitude distribution and velocity vectors around top nozzle. Pressure inside nozzle is well propagated toward nozzle exit showing atmospheric pressure. Free jet velocity is decreasing at the downstream from the nozzle exit and circular jet flow phenomena such as development of boundary layer near core region, jet velocity distribution and re-entrainment is hydro dynamically well simulated.

3.2 Multi-stage nozzle

In Fig.5 numerical results of pressure and velocity magnitude contours on mid-section plane at air-inlet of multi-stage nozzle are shown.

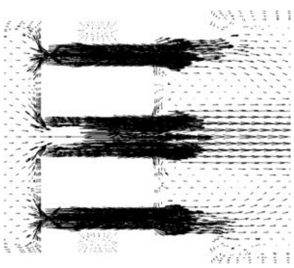


(a) Pressure distribution.

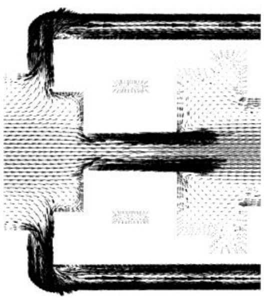


(b) Velocity magnitude distribution.

Fig. 5. Computational results in the middle section along the water spray nozzle.

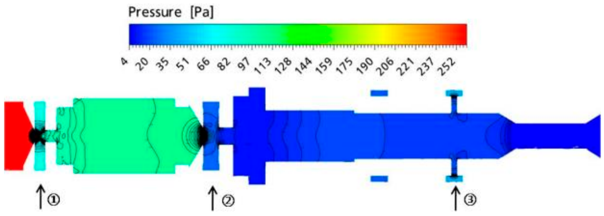


(a) Near oil spray nozzle.

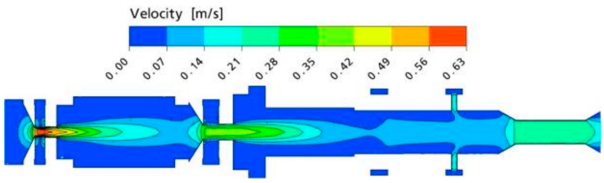


(b) Re-inflow passage.

Fig. 6. Velocity vectors around oil forming chamber

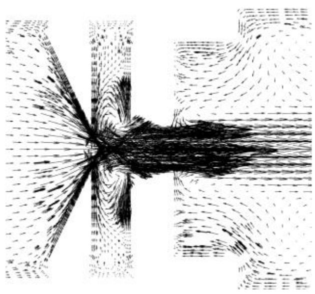


(a) Pressure distribution.

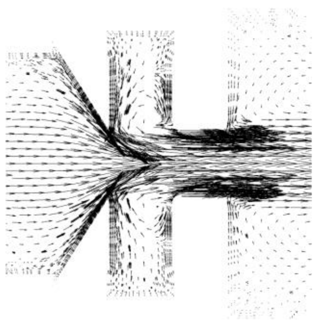


(b) Velocity magnitude distribution.

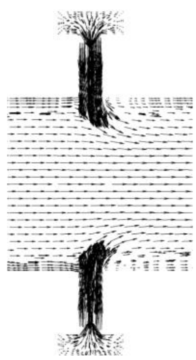
Fig. 7. Computational results in middle section along the oil forming nozzle.



(a) Junction of oil spray nozzle (①).

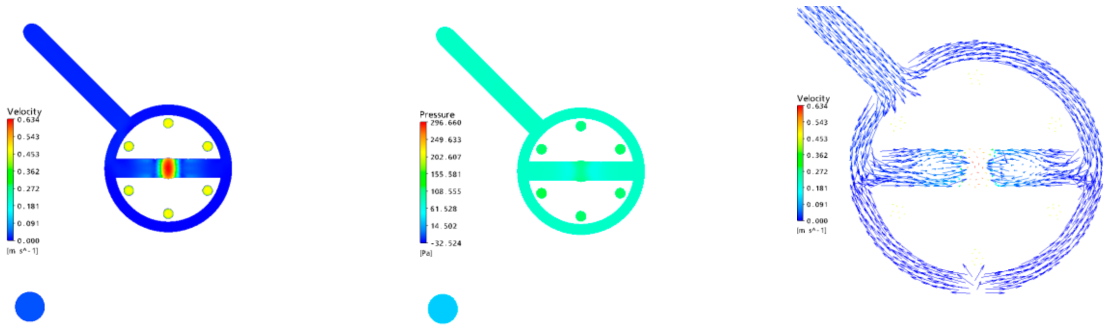


(b) Junction of water spray nozzle (②).

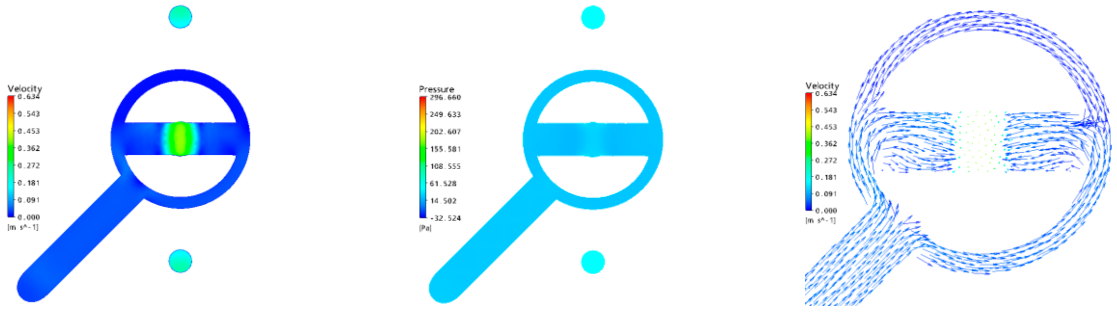


(c) Re-inflow junction (③).

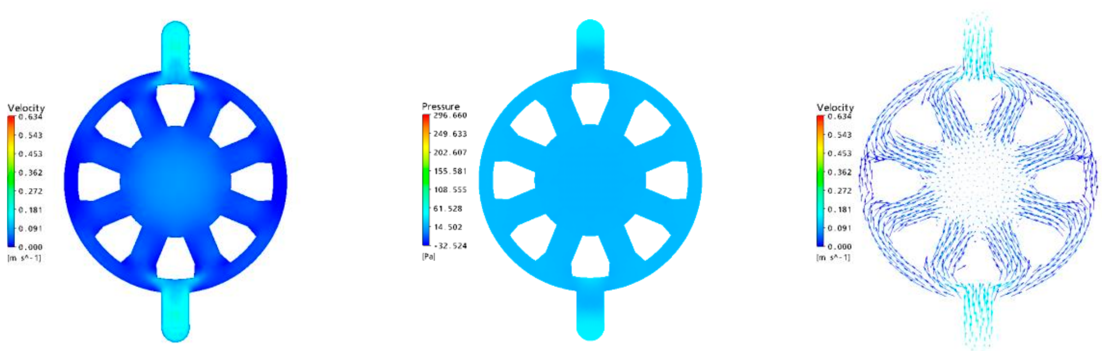
Fig. 8. Velocity vectors near spray nozzle junctions



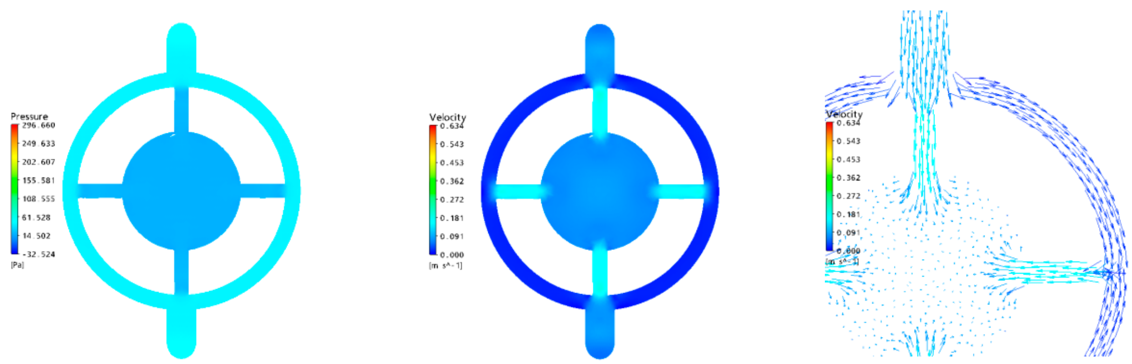
(a) Near oil spray nozzle.



(b) Near water spray nozzle.



(c) Near second water drop forming chamber (1st re-inflow passage).



(d) Near second water drop forming chamber (2nd re-inflow passage).

Fig. 9. Computational results of velocity distribution, pressure, velocity vectors at several inflow sections.

Flow is flowing toward right-hand side, and relatively low pressure is distributed near oil spray nozzle (① in Fig.5(b)) and water spray nozzle due to contraction of flow passages. Then the pressure gradually decreases toward nozzle exit. On the other hand in Fig.5(b), relatively high velocity at oil forming nozzle influences on the velocity near water forming nozzle. By this influence, most of fluids from oil forming chamber flows into water forming chamber, so that film coating is accelerated. Eventually the ultra-thin oil film coating for water droplet is obtained. At the exit, oil coated water droplets (coolant) vigorously jet out due to the decrease of flow passage. In details of velocity vectors in Fig.6, a small circulation zone is observed at main nozzle near the junction areain Fig.6(a). After main stream of oil, the remains of oils reenter to the water forming chamber through re-inflow passage(② in Fig.5(b)) in Fig.6(b) and they help to mixing water and oil.

Figure 7 shows numerical results at another middle section plane along the oil forming nozzle. As investigated in Figs.5 and 6, it is easy to understand the velocity and pressure distributions. Especially, pressure near junctions of spray nozzles propagates like a wave of sound from its source. Nozzle areas are easily detected from the velocity contours. The main stream form the air and oil spray nozzle is well flow down through the main nozzle. In Fig.8(a) and (b), the flow has somewhat large circulation in the passages both nozzle junctions before it flows into main nozzle. This circulation plays an important role of mixing the small amount of oil and air. Then, fluid from the bypass is merged into main stream without circulation, so that it makes smooth mixing and flowing of cutting fluid. In this figure, numbers of ①, ② and ③ in circle represent the numbers in Fig.7(a).

Figure 9 shows pressure, velocity magnitude contours and velocity vectors on several inflow sections of the oil spray nozzle, water spray nozzle and water drop chamber in multi-stage nozzle. It is well simulated that from each nozzle inlet located at circumferential direction of main nozzle, mixing flows are coming into main nozzle. Pressure is gradually decrease along the flow direction and flow velocity is increase at the nozzle. Several pathway and jet flows help making tiny water droplet and thin oil film coatings.

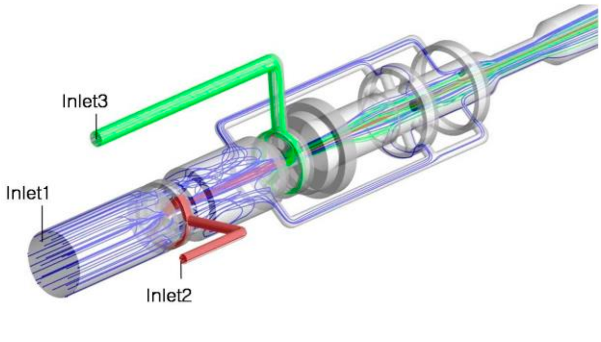


Fig. 10. Streamlines of multi-stage nozzle.

Figure 10 shows streamlines of internal flow in entire multi-stage nozzle from nozzle inlet to outlet. It reads off the complicated flow structure and mixing process in recirculation regions, inflow and outflow channel, where recirculation regions are designed for smooth mixing among air, water and oil. Actually the present multi-stage nozzle was made and tested, and showed its excellent performance in the rate of reuse of cutting oil, the evaluation of air and water pollution, roughness of cutting surface and the replacement period of cutting fluid[10]. However, due to the computational problems, it is still difficult to predict the size of water droplet and the thickness of oil film coating.

4. CONCLUSION

For the purpose of the advance design of cutting fluid supplying nozzle with ultra-thin oil film coating of water droplet, numerical simulation of three dimensional mixed fluid jet flows through multi-stage nozzle was performed. As the results, internal flow in complicated multi-stage nozzle was well simulated at the entire flow regions including water droplet forming chamber, oil and water spry nozzle, and top nozzle, so that it is well known the mixing process of air, water and oil in multi-component flow and their complicated flow pattern and structure. In the computation of top nozzle, jet flow characteristics such as development of mixing flow, re-entrainment and jet intensity were intelligibly analyzed. As far as this simulation is concerned, the present model of multi-stage nozzle system is favorable especially in the jet flow structure, atomization of water droplet and mixing of working fluids.

REFERENCES

1) Jerry P. Byers: “Metalworking Fluids,” Marcel Dekker, Inc, 1994.
2) S. Kalpakjian: “Manufacturing Processes for Engineering Materials,” 2nd ed., Addision-Welsley, 1991.
3) E. O. Bennett: “Water based cutting fluids and human health,” Tribology international, 1983.
4) H. Yoshimura, F. Itoigawa, T. Nakamura, N. Kosaburo: “Development of Nozzle System for Oil-on-Water Droplet Metalworking Fluid and Its Application to Practical Production Line,” JSME Int'l J., Ser.C, Vol.48, pp. 723-729, 2005.
5) Y, Yue, J. Sun, K.L. Gunter, D.J. Michalek, J.W. Sutheland: “Character and Behavior of Mist Generated by Application of Cutting Fluid to a Rotating Cylinderical Workpiece, Part 1: Model Development,” ASME, J. of Manuf. Sci. Eng., Vol.126, pp. 417-425, 2004.
6) G.W. Rankin, K. Sridhar, M. Arularja, K.R. Kumar: “An Experimental Investigation of Laminar Axisymmetric Submerged Jets,” J. Fluid Mech., Vol.133, pp. 217-231, 1983.

7) M. Arularja, G.W. Rankin, K. Sridhar: “Maximum Velocity Decay in a Submerged Laminar Jet Issuing from a Long Tube,” Trans. Canadian Society of Mech. Eng., Vol.7, pp. 41-43, 1983.
8) V.N. Dmitriev, N.A. Kulesova: “The Calculation of a Laminar Jet in the Surrounding of the Supply Nozzle,” Proc. 5th Jablonna Fluidics Conf., pp. 83-91, 1974.
9) ANSYS, CFX User’s Manual, Version 11, 2007.
10) S. Ahn: “Development of Cutting Fluid Supplying System by Using Ultra-thin Oil Film Coating Technology,” MCIE Report, 2006.