THE FLUID FLOW SIMULATION THROUGH TO A VENTURI NOZZLE

IONEL OLARU*

"Vasile Alecsandri" University of Bacau, Calea Marasesti 157, Bacau, 600115, Romania

Abstract: In this work it was performed a fluid flow study of cooling systems for metal cutting. The study of forces resulting from the impact of fluid jets and when fluids are diverted round through the nozzle involves the application of pressure forces. The forces are determined by calculating the change of momentum of the flowing fluids. It was performed a simulation of the flow through a nozzle commonly used for pressure cooling of the cutting zone.

Keywords: fluid, pressure, velocity, jet, nozzle

1. INTRODUCTION

Fluid jets are used in various industries including the machine building industry. Nozzles that are sprayed jets of fluid can take various forms: convergent, divergent, and convergent-divergent. Convergent-divergent nozzles are used to provide a fluid jet cutting in the processing of the material by its appropriate dispersion [1].

Venturi nozzles are classified in tree types according to the manufacture method: rough-cast convergent, machined convergent and fabricated convergent. The advantages of the Venturi tube are its ability to transport materials in suspensions without clogging and its low overall energy loss. Traditional applications of this device have been in low-pressure gas lines and water and sewage mains [2].

The way the jet flow, we have: jet with laminar fluid flow or jet with turbulent fluid flow. After the nature of the fluid that forms the jet two types of jets can be distinguish: flooded jets - if the two fluid media are of the same nature (gas - gas, liquid - liquid) and none flooded jets - if the two fluid media have different natures.

After the interaction with solid walls of nozzle the jets can be classified in: free jets - have contours undisturbed by contact with solid walls of the nozzle, and forced jets - have contours disturbed by contact with solid walls of the nozzle. The pressure loss in terms of pressure developed by a Venturi is dependent primarily on the exit cone angle and the diffuser diameter to inlet diameter ratio.

Fluid jets are used in many industries; this paper aims to study the jets of fluid that pass through the convergent-divergent nozzle particularly used in machine building industry. With this type of nozzle is performed cooling of machining zone. In this sense have been made many specific studies to research how fluid leaves the nozzle and how it wet in the processing area to achieve an adequate cooling when is used a small quantity of lubricant. There are also important physicochemical characteristics of selected cooling fluids [2].

In this paper we propose to determine the flow of the cooling fluid through a Venturi nozzle used in efficient operation of cooling systems in the building machine industry. Study the way in which the fluid flows through

* Corresponding author, email: ionelo@ub.ro

the nozzle by computer simulation. In mechanical cutting process is very important to study the cooling fluid jet using minimal quantity of fluid.

2. EXPERIMENTAL PROCEDURE

Venturi nozzles consist essentially of a cylindrical inlet, convergent zone, diffuser and divergent outlet, the divergent outlet is present to reduce the overall loss of the meter [3].

None flooded jets liquid jet comprises three distinct areas:
- A compact area, which has a central core and the speeds are high;
- An area of disintegration, which under the influence of air friction and turbulence interior contribute to get into the flow of air bubbles, which breaks the jet;
- Splash zone, the fluid are completely disintegrate in the form of drops.

The driving fluid or motive (oil, liquid, emulsions or air) passes through the nozzle, converting the pressure energy into a jet. The result of this is that the fluid at the front of the jet is displaced, producing an area of low pressure at the exit of the nozzle (Figure 1) [4].

Fig. 1. Example of nozzle jet.

The conventional geometry of the Venturi nozzle is presented in Figure 2. That nozzle is used from building machine industry cooling systems.

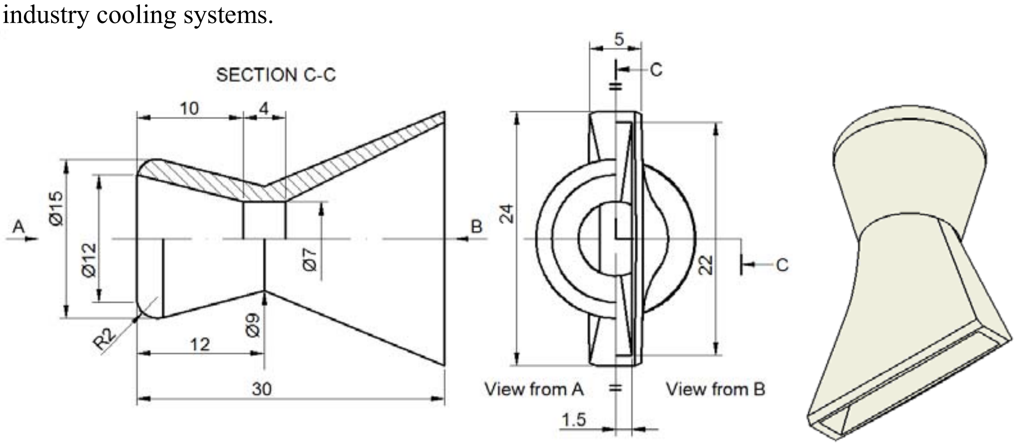

Fig. 2. Geometrical dimensions of nozzle.

Mathematical calculation is made on the same Venturi nozzle. Nozzle will a flow rate and pressure at the inlet and atmospheric pressure at outlet. The working fluid is water; the jet is dispersed in air.

First for pressure force calculation we must identify the control volume on which to conduct a force balance. The nozzle is filled with fluid with pressure p_1 at inlet and p_2 at outlet. There will be forces on the outer surface of the volume due to atmospheric pressure because the nozzle is small volume. The approach to be used here is to determine the forces to the directions x and y to find the resultant force.

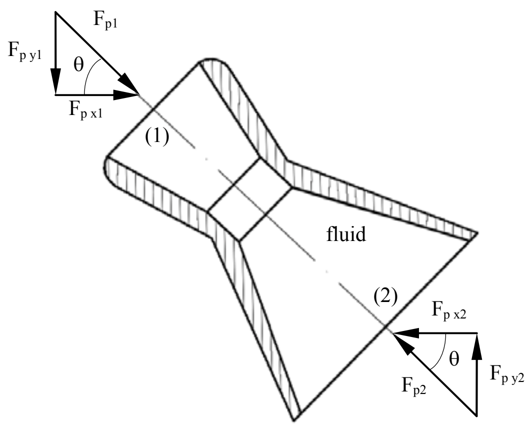

Fig. 3. Nozzle pressure forces.

Pressure forces are the force normal to the surface and have expression:

$$F_p = p \cdot S \ \text{[N]} \tag{1}$$

where:

p is pressure, N/m^2;
S - area which act the pressure force, m^2.

Pressure at the inlet of the fluid is 2 bars (19.99 · 10^{-4} N/m^2), nozzle inlet area is S_1 = 37,7 mm^2 and nozzle outlet area is S_2 = 66 mm^2.

For the nozzle shown in Figure 3, we have pressure forces at the inlet (1) and at the outlet (2). The total horizontal and vertical forces (F_H, F_V) can be written as system:

$$\begin{cases} F_H = F_{p_1} \cos\theta - F_{p_2} \cos\theta \\ F_V = F_{p_1} \sin\theta - F_{p_2} \sin\theta \end{cases} \tag{2}$$

where:

$F_{p\,1}$, $F_{p\,2}$ are pressure forces at the inlet and the outlet of nozzle, N;
θ - angle of the jet of fluid (nozzle).

3. FLOW SIMULATION PROCEDURE

In order to achieve the simulation we used the module Flow Simulation from Solidworks program. Initially was developed 3D model of the nozzle. After the realization of the model were determined by analysis of flow conditions. The finite element method initiated by establishing computation domain, fluid subdomains, boundary conditions and mesh realization [5].

Water was used as working fluid, nozzle inlet pressure 2 bars and at outlet atmospherically pressure. After running the simulation could be viewing mode through which the flow in the nozzle (Figure 4).

As expected in convergent nozzle zone will register lower pressures with increasing fluid velocity. The exit zone of the fluid from nozzle (in the divergent section) pressure increases with increasing of surface. In terms of speed, the latter having higher values the center of the nozzle decreasing toward nozzle walls.

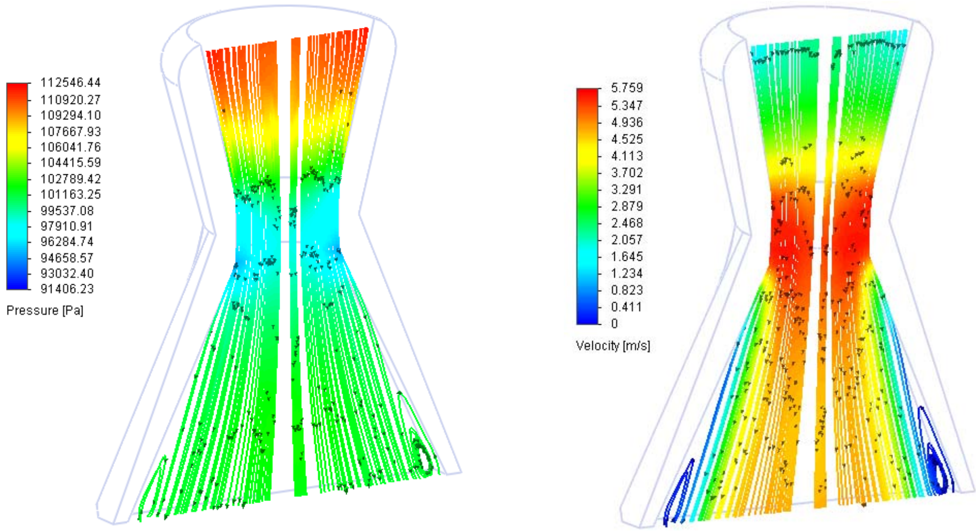

Fig. 4. Pressure and fluid velocity variation through to the nozzle.

As can be seen on pressure scale we have values between $9.1 \cdot 10^{-4}$ and $11.2 \cdot 10^{-4}$ N/m^2 inside the nozzle. Flow velocity with a range between 0 and 5.75 m/s.

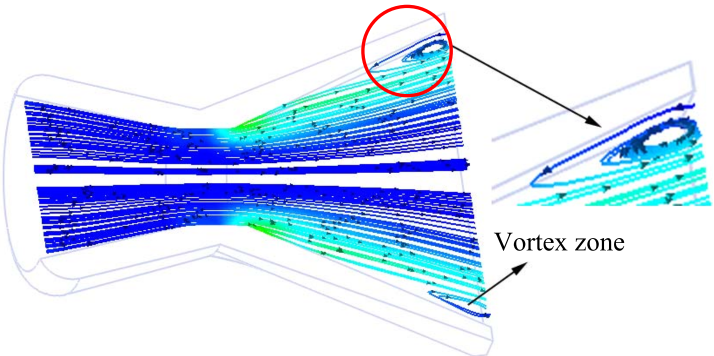

Fig. 5. Vorticity analysis through to the nozzle.

From the vortices analysis we can see that the nozzle exit section of small vortices are formed which lead to a turbulent fluid motion at the exit (Figure 5). Same area vortices generate a more pronounced dispersion of fluid flow. So this leads to a large area of wet processing area which can only be beneficial.

4. CONCLUSIONS

Following this study it may be established that the realization of efficient cooling systems for machine tools have studied the shape of pulverization cooling fluid in processing area. Furthermore are important input parameters of these fluids so that they use a small quantity as the better results.

In terms of flow through a Venturi nozzle, this study shows some important aspects regarding the parameters used for cooling fluids. Fluid flow is important, such as the flow at the exit of the nozzle, as jet of fluid is dispersed on the surface to perform a proper wetting and also the proper cooling.

A better correlation with the cooling and mechanical machining itself requires taking into consideration the temperature of these fluids. In this study was used a fluid at room temperature.

REFERENCES

[1] Frank, M. W., Fluid Mechanics, Fourth Edition, McGraw-Hill Series in Mechanical Engineering, University of Rhode Island, 2001.
[2] Bloomer. J. J., Practical Fluid Mechanics For Engineering Applications, Marcel Dekker Inc., New York, 2000.
[3] Florescu, I., Florescu, D., Olaru, I,, Mecanica fluidelor şi maşini hidraulice. Îndrumar de laborator, Editura Tehnica Info Chişinău, 2003.
[4] Krause, E., Fluid Mechanics, Springer Berlin Heidelberg New York, 2005.
[5] Lewis, W. R., Nithiarasu, P., Seetharamu, N. K., Fundamentals of the Finite Element Method for Heat and Fluid Flow, John Wiley & Sons, 2004.