COBEM-2017-1752
EXPERIMENTAL AND NUMERICAL ANALYSIS OF UNDEREXPANDED JET

Derek Jorge
Marcelo Mendes Vieira
Universidade Federal de Mato Grosso, Mechanical Engineering, Rondonopolis-MT, Brazil.
derekjorge@live.com
mmendes@ufmt.br

Thais K. Albernaz de Moraes
Daniela de Oliveira Maionchi
Universidade Federal de Mato Grosso, Physics Institute, Cuiaba-MT, Brazil.
thais@fisica.ufmt.br
dmaionchi@fisica.ufmt.br

Robson Leal da Silva
Universidade Federal da Grande Dourados, Mechanical Engineering, Dourados-MS, Brazil.
rlealsilva@hotmail.com

Abstract. *Compressed air equipments are widely used in research and industry. The present contribution deals with numerical and experimental studies of the compressibility phenomena that occurs in fluid jets with the formation of shock waves. Numerical simulations based on finite volume integration were performed for a compressible gas flow over a nozzle using the CFD software OpenFOAM. The effect of working parameters, such as nozzle diameter and flow rate for an air jet was analyzed. The smaller the nozzle diameter the greater the occurrence of shock waves. Higher flow rates also favor the appearance of shock waves. The Schlieren method of visualization was adopted to obtain the images of the shock waves with a CCD camera. The experimental results presented a good accuracy to the numerical ones, what validates the model proposed to describe the jets.*

Keywords: *compressible flow, CFD, underexpanded jets, Schlieren visualization method.*

1. INTRODUCTION

Compressible phenomena can take place in many compressed air devices depending on upstream conditions of pressure and temperature. When the downstream pressure is approximately the half of the stagnation pressure, it is possible to verify that the flow is chocked. If the downstream pressure decreases, the flow remains chocked and a supersonic flow can be identified in the fluid jet at the nozzle exit. A test bench for flow visualization using the Schlieren method can be used to observe the supersonic effects that occur in such jets (Vieira *et al*., 2000 and Staschower *et al*., 2002). This method is similar to the shadowgraph technique and relies on the fact that light rays are bent whenever they encounter changes in density of fluid. Parallel to this experimental setup, CFD (Computational Fluid Dynamics) has been a widely adopted numerical analysis tool to study such systems, enabling to reveal the parameters that influence mostly the flow features.

There are works that show underexpanded jets (Anderson, 1990 and Munson *et al.*, 2002). In others studies there are authors that investigated this subject and obtained a correlation of any gases of the rate of upstream and downstream pressure with the distance of Mach disk that is a frontal shock wave of jet (Crist *et al*., 1966). The structure of jet have a complex format because in exit nozzle there are expansions waves that coalesce in the oblique shock wave of the compression that follows of the interior jet (Crist *et al*., 1966). This oblique shock wave maintains supersonic velocity after wave in which undergo a new compression with other oblique shock wave.

The numeric simulation must have capability to capturing shock wave in supersonic flow (Anderson, 1995). The OpenFOAM software has several examples to validate the code and any are supersonic flow examples that appear the occurrence of shock waves (OpenFOAM, 2017). Wüthrich (2007) utilizes OpenFOAM to simulate any cases of the supersonic flow. Additionally, Fuszko and Olšiak (2016) obtained numeric results of the behavior coaxial flow. Their model is axisymmetric and uses OpenFOAM code and is included the turbulence term in simulation. In the study of CFD and OpenFOAM, Samel (2011), Tridal (2015), Taludkar (2015) and Zang *et al*. (2017) report studies with good solutions that analyze underexpanded jets.

2. OPTICAL SYSTEM

The optical system was designed with the Schlieren method in the “Z” configuration. Parabolic mirrors were used for the subject of obtaining a parallel light beam covering a 10 cm diameter circular region. There are spatial filters or pinholes that have diameters between 0.3 and 1.0 mm. It was found that a regular 40W automotive light bulb was suitable as a lighting source. A monochromatic CCD camera (Coastar, model CV-M50) imaged the region of interest obtaining frames at a rate of 30 fps, as usual. However, the electronic shutter aperture operated at 0.1 ms. An image acquisition system (Data Translator, model DT-3152) captured the CCD images. The shock waves appear the high gradient of density in the flux of the stream line. (Liepmann and Roshko, 1957 and Thompson, 1972) and the Schlieren method is the good system to visualize the shock waves position in the jets. This rig test is the same used in the work of the Vieira *et al.* (2000) and Staschower *et al.* (2002).

3. NUMERICAL SIMULATION

For Wolfram (2002) the Navier Stokes equations are differential equations describing the flow of fluids and are the partial derivatives that allow determining the fields of velocity and pressure in a flow. They were named after Claude-Louis Navier and George Gabriel Stokes developed a set of equations that would describe the movement of fluid substances such as liquids and gases. These equations establish that changes in momentum and acceleration of a fluid particle are simply the result of changes in pressure and viscous dissipative forces (similar to friction) acting on the fluid. This viscous force originates in molecular interaction.

These equations are obtained from basic principles of conservation of mass, momentum and energy. To make it easier to apply these principles it is useful to consider a finite arbitrary volume, called control volume. The control volume remains fixed in space or can move like fluid, which leads to special considerations.

For a simulation in CFD, we have to do a modeling compatible with the physics of the problem, having to take the necessary actions and thus know the limitations of the simulation, as well as the possible errors and the degree of precision of the results compared with the experimental one. This is the only way to validate the simulation and the results obtained. First, it is chose an equation or equations that will serve as a mathematical model of the physical problem, and from this model to perform discretization of the geometric model. The mesh should always be compatible with the problem physics to be simulated. To solve the problem, we adopted a two-dimensional and axisymmetric mesh, which is a structured mesh and qualitative classification, with a better organization in its appearance and the desired format. Given the axisymmetric geometry, the mesh has an inclination of 5 degrees around the longitudinal axis or axis of revolution (x-axis). For mesh generation, it is used “blockMesh” module of the OpenFOAM.

With the mesh characterized, next step is the discretization of the differential equations, that is, the numerical method to be performed, this step is performed automatically in the chosen "solver". This discretization has already been done by the developers who implemented in the source code of OpenFOAM, what is necessary to know are the equations and the transitions of them. In this step, the numerical method is chosen and as it is OpenFOAM, it is the finite volume method, which is the most suitable for fluid mechanics. From this point on, the appropriate solver is chosen through the User's Guide (OpenFOAM UserGuide), which provides the "solvers" and their available equations. For each case, the relevant boundary conditions are determined as well as certain physical properties, such as viscosity of the constant fluid or non-slip condition in the wall of the conduit.

3.1 Mathematical equations

The constitutive equations in Fluid Mechanics. In this initial approach, the equations are shown in a Cartesian system of three dimensions (three-dimensional), which will later involve replacing these differential equations with a system of algebraic equations, represented by a matrix conservation system of mass, Navier-Stokes equations, and conservation of energy.

As the mass of the system remains constant with the variation of time, assuming the constant volume of an element, we have represented in “Eq. (1)”.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \tag{1}$$

By the conservation equation of linear momentum (Newton's 2nd Law) in differential form, for Cartesian coordinates can be written as “Eq. (2)”.

$$\frac{\partial(\rho\vec{V})}{\partial t}+\nabla\cdot(\rho\vec{V}\vec{V})=-\nabla p+\nabla\cdot\tau+\rho g \tag{2}$$

The principle of energy conservation (1st Law of Thermodynamics) applied to a fluid element that moves with the flow is given by the equality of the energy variation in the element and the sum of the resulting heat flow into the element plus the rate of work performed on the element by field and surface forces. The following equation is given in the conservative form, adding the equation of continuity “Eq. (3)” multiplied by the internal energy, *e* (Fortuna, 2012). The viscosity of fluid was considered null which is other simplification this numerical solution.

$$\frac{\partial\rho.e}{\partial t}+\nabla[V(\rho V)]+\nabla(V.p)+\nabla(\tau V)=0 \tag{3}$$

The flow is considered adiabatic and therefore it is possible to simplify hypotheses for the energy conservation equation with the absence of external and internal sources of heat.

3.2 First Structured Mesh

The creation of the mesh requires some attention since its language must be well elaborated and plausible in the considerations. The model that will be shown below was used in the simulation of this work. “Fig 1” shows a schematic drawing of the vertices of the simulation mesh.

The “Fig. 2” and ”Fig. 3” represent underexposed flow with the occurrence of shock waves, where “Fig. 2” represents the image of the simulation and “Fig. 3” represents the image of the experimental analysis.

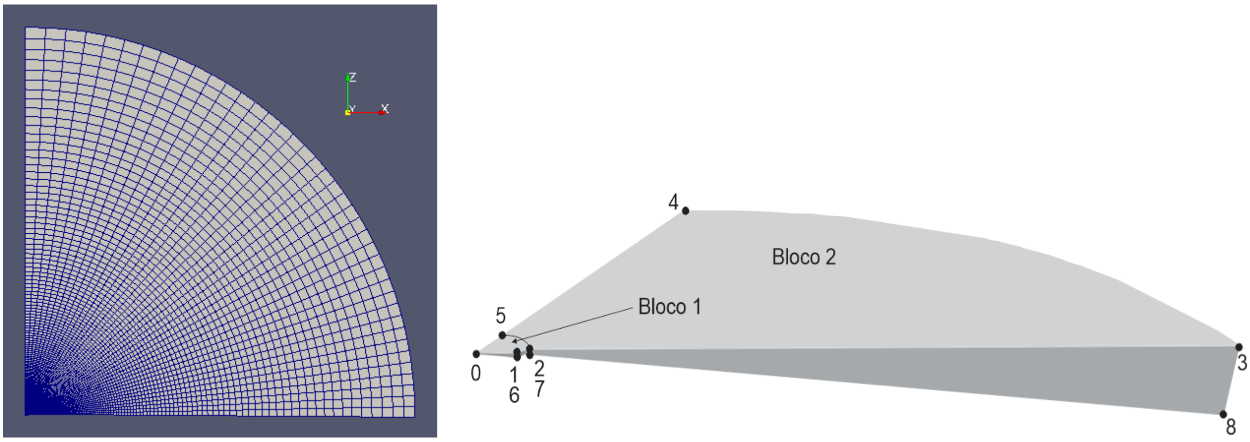

Figure 1. Schematic drawing of the structured mesh, elaborated in blockMesh. In first image, it is showed the discretization and other image is viewed the inferior face with fifty degree angle.

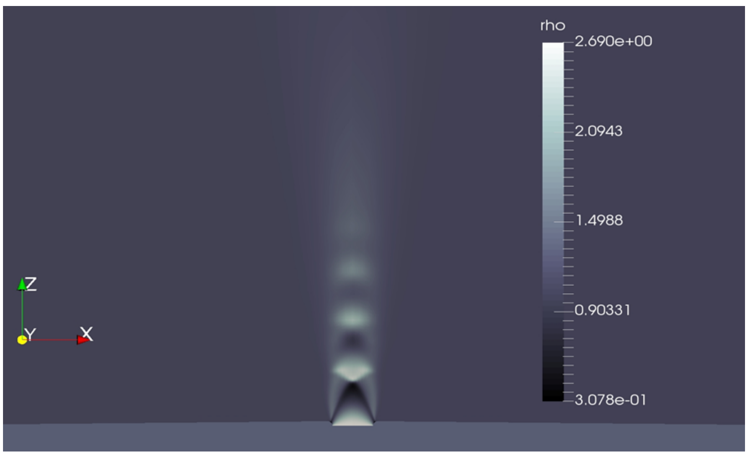

Figure 2. Simulation about fluid density that is an image obtained by Paraview.

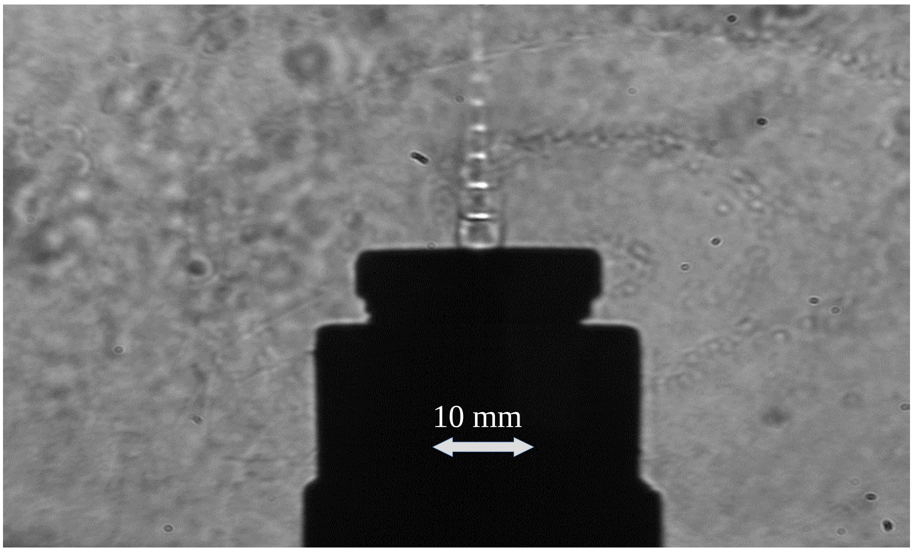

Figure 3. Experiment of fluid flow through the nozzle.

3.3 Second Structured Mesh

This was projected with refined mesh along the axis jet. This way, the result was nearest of the experimental jets.

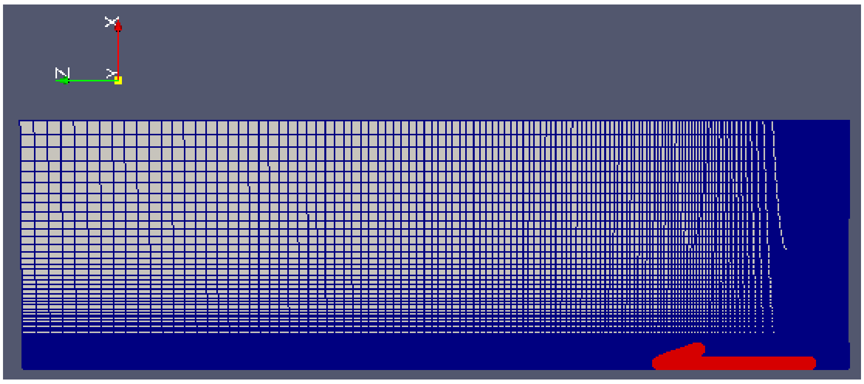

Figure 4. Schematic drawing of the second structured mesh, elaborated in blockMesh. The arrow illustrates the direction and position of central line of jet.

4. NUMERIC AND EXPERIMENTAL RESULTS

The stagnation pressure and temperature of the atmospheric air was measured at upstream jet. The air jet was discharged into the atmosphere and the environment pressure is equal to 98+/-1 kPa. Thus, it is showed the collected data of runs in Tab. 1, which P_0 is gauge stagnation pressure and T_0 is stagnation temperature.

Table 1. Experimental results for underexpanded air jet.

Test	Exit diameter nozzle	P_0	T_0
A	2.5 mm	4.0 +/- 0.2 kgf/cm²	31.9 +/- 0.2 ^{o}C
B	3.2 mm	2.8 +/- 0.2 kgf/cm²	31.8 +/- 0.2 ^{o}C
C	3.2 mm	3.0 +/- 0.2 kgf/cm²	32.0 +/- 0.2 ^{o}C
D	5,0 mm	1.2 +/- 0.2 kgf/cm²	31,6 +/- 0.2 ^{o}C

In the sequence is viewed the image obtained from optical schlieren method for the first case which the exit diameter nozzle is 2.5 mm (Fig. 5). The other image in Fig. 5 is illustrated the simulation from OpenFOAM. The maximum pressure this jet has the value of 258.3 kPa, approximately, that is equal the critical pressure the compressible flow. Also, this figure shows the pressure establishing to the atmospheric environment. Under the frequency of shock wave forwards of the jet, the Fig. 6 illustrates the graphic that compare the simulation and photographic image this run. Under the spatial shockwave frequency along the jet, Fig. 6 illustrates the graph that compares the simulation and the photographic image of that run.

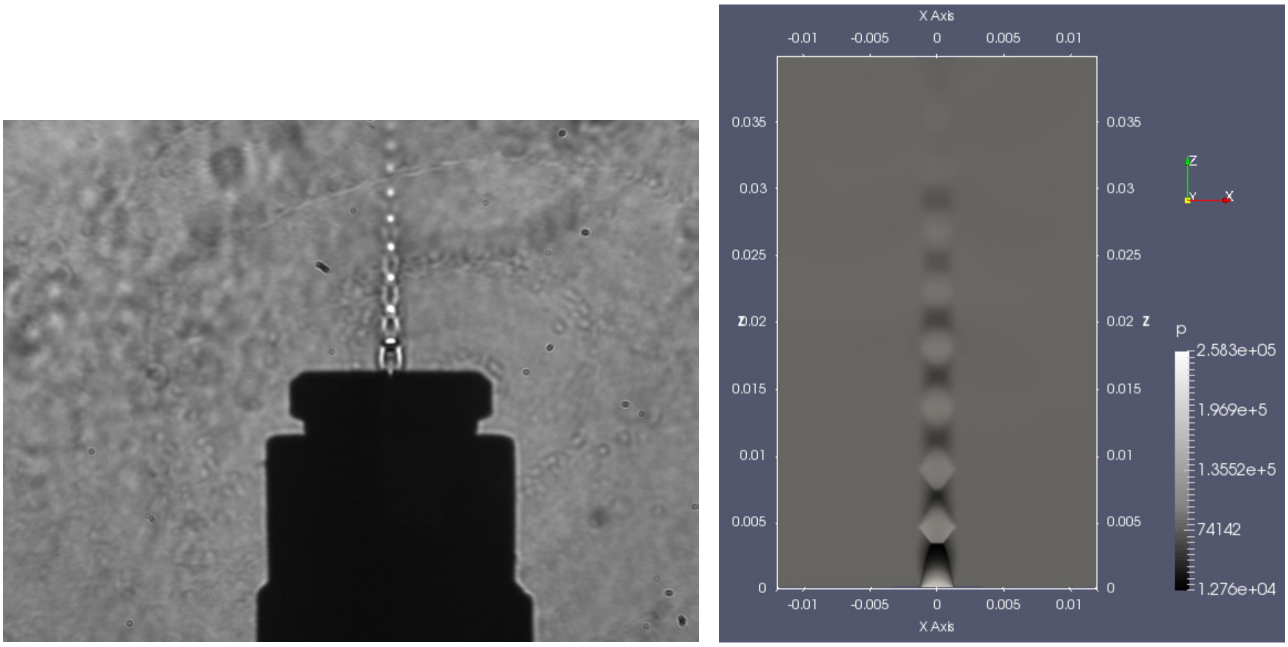

Figure 5. In left side is the photographic register of the air jet that used of 2.5 mm exit diameter nozzle (case A). The other , it is viewed the pressure field in the flow.

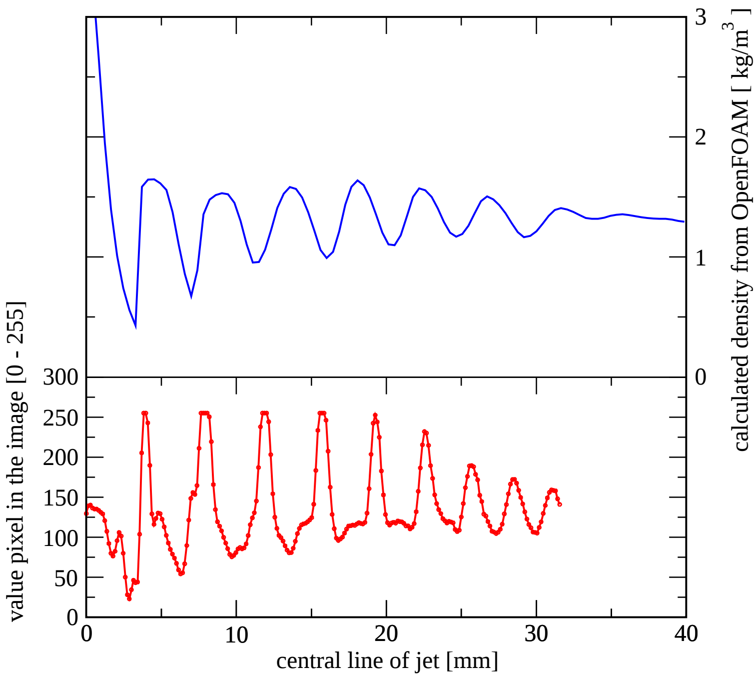

Figure 6. This graph compares the density changes along the central line jet with the collected data of pixels obtained from photographic image (Fig. 5).

In other tests, the results are similar. In the Fig. 7, it is illustrated the jet of nozzle with 3.2 mm of diameter (case B). How the case C uses the same nozzle, in the Fig. 8 is illustrated the comparative between case B and case C. It is possible to conclude that the pressure increasing also leads a increment in distance of the axial shockwave. In Fig. 9, it is indicated the vortex formation to the case D. This is highlighted because the propagating of the vortex overlaps on the shockwaves and in this way, the shockwaves have not sharpness in photographic register.

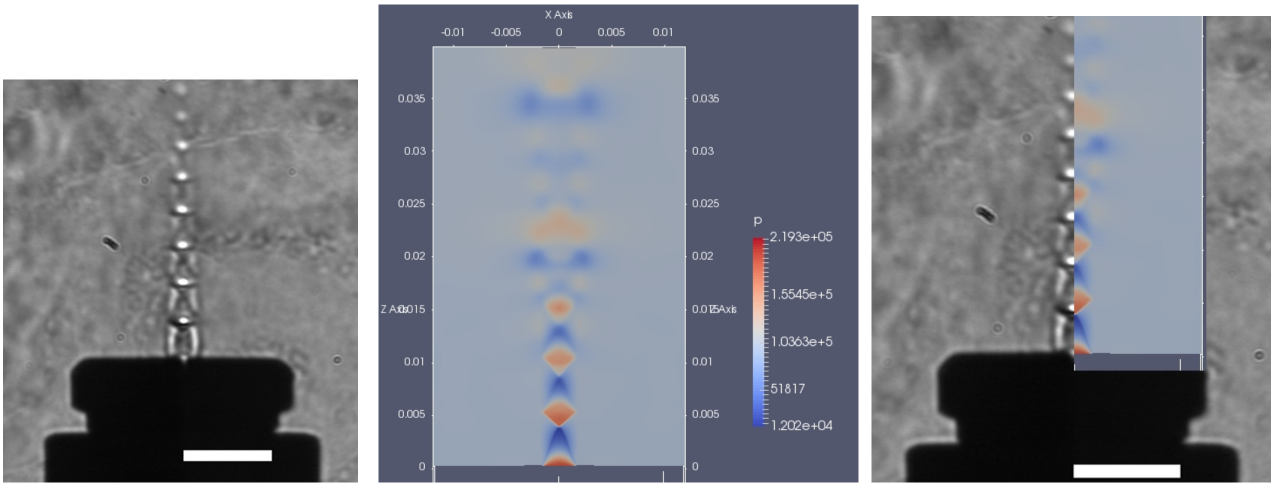

Figure 7. The first image is the experimental test, the second image is the numerical result with pressure indicating in flow and the last compares the previous images (case B).

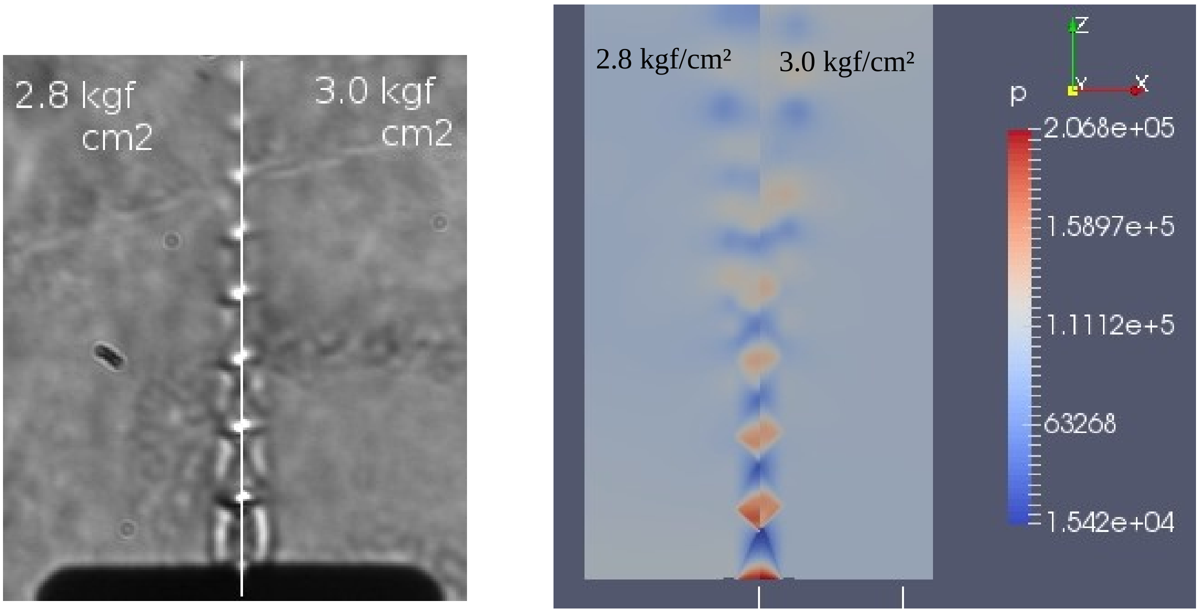

Figure 8. Comparative between case B and C.

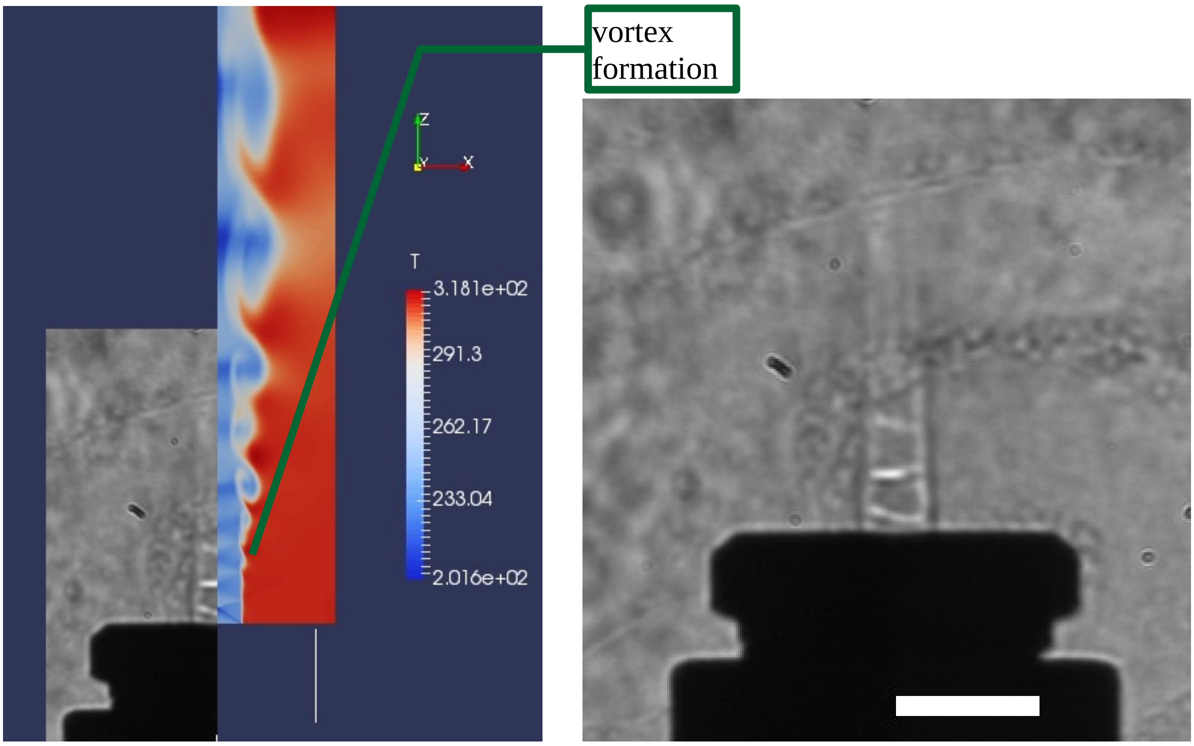

Figure 9. Comparative between experimental and numeric solution from case D. In right image, the photographic register not presents the sharpness shockwave formation.

5. CONCLUSIONS

The OpenFOAM demonstrated to be excellent to simulations that there are compressible phenomena. It was possible to observe shockwave formation and the shape was consistent. To case B and C that there was a increment of the pressure in upstream, the experimental and numerical solution obtained results that was sensible this change. In case D, the numerical analysis demonstrated the propagation of vortexes that overlap on shockwave of the jet and thus optical system could not register the sharpness of the shockwave shape.

6. ACKNOWLEDGEMENTS

We express gratitude to SISEA – Alternative Energy System Laboratory of the Polytechnic School of USP which provided the optical system of this work.

7. REFERENCES

Anderson, J. D., 1990, “*Modern Compressible flow*”, second edition, McGraw-Hill

Anderson, J. D., 1995, “*Computational fluid dynamics - The basis with applications*”, McGraw-Hill.

Brunetti, F., 2008, “*Mecânica dos Fluidos*”. 2 ed. São Paulo: Pearson Prentice Hall, Revised.

Buchmann, N. A.; ATKINSON, C.; SORIA, J., 2012, “*Ultra-high-speed tomographic digital holographic velocimetry in supersonic particle-laden jet flows*”. Measurement Science and Technology.

Christopher, J. G., 2011-2016, “*User Guide OpenFOAM*”, Foundation Ltd., CFD Direct Ltd.

Crist, S., Sherman, P. M., and Glass, D. R., 1966, “*Study of the Highly Underexpanded Sonic Jet*”, AIAA Journal, Vol. 3, nº 1, pp. 68-71.

ESSS. www.esss.com.br, 11 March 2016.<www.esss.com.br/blog/2016/03/simulacao-computacional-por-que-investir/>

Fortuna, A. D. O., 2012, “*Computational Techniques for Fluid Dynamics*”. Second ed. São Paulo: publishing company University of São Paulo.

Fox, R. W.; McDonald, 2001, “*Introduction to Fluid Mechanics*”. Fifth ed. [S.l.]: LTC.

Fuszko,Z. and Olšiak, R., 2016, “*Design and CFD Analysis of an Axisymmetric Supersonic Plug Nozzle for an Air-Air Ejector*”, AIP Conference Proceedings <http://dx.doi.org/10.1063/1.4963055>

Liepmann H. W. and Roshko, A., 1957, “*Elements of Gasdynamics*”, New York, John Wiley & Sons, Inc.

Lopes, P. M. B., 2013, “*Free-surface flow interface and air-entrainment modelling using OpenFOAM*”, Thesis Project in Hydraulic, Water Resources and Environment Doctoral Program in Civil Engineering, Coimbra.

Munson, B. R., Young, D.F., B. R., Okiishi, T. H., 2002, "*Fundamentals of Fluid Mechanics*", Fourth Edition, John Wiley and Sons.

OpenFOAM software, 2017, "*Tutorial Guide*", July/2017, <http://www.openfoam.com/documentation/tutorial-guide/>.

Staschower, M.; Vieira, M. M.; Simões-Moreira, J. R., 2002, "*Didactic Film for Visualization and Study of Compressible Flow in Laval Nozzles*", ENCIT.

Samel, M. A., 2011, "*Numerical Investigation of Gas-Particle Supersonic Flow*", Thesis of Master, University of Massachusetts Amherst

Talukdar, M. I. H., 2015, "*Numerical simulation of underexpanded air jet using OpenFOAM*", Thesis of Master, University of Stavanger, 15 June 2015.

Thompson, P., 1972, "*Compressible-Fluid Dynamics*", Rensselaer Polytechnic Institute Press

Trydal, J., 2015, "*CFD analysis of temperature development due to flow restriction in pipeline*", Thesis of Master, University of Stavanger.

Van Wylen; Borgnakke, C.; Sonntag, R. E. "*Fundamentals of Thermodynamics*". eighth ed. [S.l.]: Edgard Blucher, 2013.

Vieira, M. M., Guimarães, M. S. and Simões-Moreira, J. R., 2000, "Estudo experimental de jatos evaporativos", ENCIT 2000.

Wolfram, S. A, 2002, "*New Kind of Science*". [S.l.]: LLC.

Wüthrich, B., 2007, "*Simulation and validation of compressible flow in nozzle geometries and validation of OpenFOAM for this application*", Master Thesis, ETH Swiss Federal Institute of Technology Zurich

Zang, B., US, V., New, T. H. D., 2017, "OpenFOAM based numerical simulation study of an underexpanded supersonic jet", In 55th AIAA Aerospace Sciences Meeting, pp. 747.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.