ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

AERODYNAMIC CHARACTERIZATION OF SUBSONIC JETS IN CROSSFLOW

Pedro Ricardo Corrêa Souza
Odenir de Almeida
Federal University of Uberlândia, Campus Santa Mônica 2121, Santa Mônica, 38408-900, Uberlândia, MG, Brazil
pedro_ricardo@mec.ufu.br; odenir@mecanica.ufu.br

Carlos Roberto Ilário da Silva
EMBRAER S.A., Av. Brigadeiro Faria de Lima 2170, São José dos Campos, 12227-000, Brazil
carlos.ilario@embraer.com.br

Abstract. *The present work is aimed to the aerodynamics characterization of subsonic turbulent jets under the effects of crossflow by means of numerical simulations using Reynolds-averaged Navier-Stokes (RANS) approach. The simulations are conducted using the well-known CFD++ commercial code from Metacomp Inc. Following a step-by-step procedure, firstly an analysis of turbulence models and grid independence is performed. Secondly, detailed comparisons of mean flow properties from the numerical solution with experimental data available in the literature are illustrated. The results show the strong asymmetry of the jet's potential core and its spreading rate, especially for high crossflow velocities, which shows the capability of RANS as a good prediction tool for this type of flows. This study is part of an ongoing experimental and numerical research on subsonic jets in crossflow, currently being carried out in the Federal University of Uberlândia (UFU), and it will serve as basis for assembling the experimental facility. As a final target, the aerodynamic characterization of such jets will provide the flow field for further aeroacoustics investigations by using the LRT (Lighthill Ray-Tracing) method.*

Keywords: *Crossflow, Subsonic Jet, Aeroacoustics, CFD, Aerodynamics*

1. INTRODUCTION

A single flow jet subjected to an external crossflow is not an unknown problem. Indeed, many researches have been conducted in the last decades about such problem. From an engineering practical point of view, a jet under the effect of crossflow can be identified in several applications such as: plume dispersal from smokestacks and volcanoes; effluent dispersal for liquid disposal in streams; injection of fuel in jet engine combustors; reaction control jets, used on rockets and missiles; the jets from a vertical short takeoff and landing aircraft.

In the aeronautical context, subsonic jets under the cross-flow effect have been the focus of several types of researches. At the beginning, back on 60's, such studies were devoted to vertical takeoff aircrafts (V/STOL). In fact, many prototypes airplanes have been built at that time and, the Harrier Jump Jet is the most known example of industrial application. However, more recently, other applications for civilian aviation is becoming prominent, as the cross-flow effect in the operation of turbofan engines and auxiliary power units (APU). Figure 1 shows such industrial applications.


(a) (b)

Figure 1. Exhaust flow on aeronautical engines: (a) APU; (b) V/STOL airplane; source: www.airliners.net

In order to tackle this problem, fundamental works using experimental setups were devised through the years. Authors like (Keffer, 1963), (Ribner, 1946) and (Ruggeri et al, 1950) were able to brought a good level of understanding about the flow phenomena. Different experimental techniques have been used to investigate the flow, such as pitot tubes and hot-wire anemometry. More recent works point out the modern use of advanced techniques such as Particle Image Velocimetry (PIV), Laser Doppler Anemometry (LDA) and Laser Induced Fluorescence (LIF). All of those powerful setup allowed a general good description of the flow details, not only in terms of averaged properties as pressure and velocity fields, but also by investigating the turbulence behavior, discretizing and recognizing different flow structures.

On the other hand, unique numerical work is becoming another powerful tool for flow analysis. The great advances in numerical methods and computational power have given chance to unleash the capabilities of computational fluid dynamics applied to more complex industrial flows. In this direction, numerical methods for studying jets under cross-flow are found since RANS (Reynolds Averaged Navier-Stokes), URANS (Unsteady Reynolds Averaged Navier-Stokes) up to LES (Large Eddy Simulation) and DNS (Direct Numerical Simulation). In this research line, the works of (Rudman, 1996), (Cavar and Meyer, 2012), (Karvinen and Ahlstedt, 2005), (Chai & Mahesh, 2011) and (Yuan *et al*, 1999) stand out.

As regards of the acoustic study associated with the problem, just a little literature is available. As mentioned, some studies conducted in the NASA approach the subject, but these studies have reported aircraft applications in the 70s where V / STOL were in vogue, as mentioned in the work of (Cole, 1972) and (Camelier and Karamcheti, 1976).

One of the practical applications is the exhaust gases of an aircraft's APU. It is known that such a system is a source of noise in the ground (with the aircraft parked). The presence of the wind changes the characteristics of acoustic propagation interfering with the perception of sound. Specifically, it is also shown that the emission of sound is affected by the shape, size and velocity of the fluid streams. Studies with this focus has been recorded recently in the academic community, however, this is a highly impactful subject in the aeronautical field.

The main objective of this work is to study the jet's noise propagation under the effect of crossflow, therefore, it becomes necessary a good characterization of the fluid flow properties. In the present work it's used the RANS simulation method, despite of its inherent limitations, to investigate if this type problem can be solved with appropriate accuracy when compared with a DNS simulation. RANS simulations are considerably less demand when compared with the DNS ones, so if it's possible to describe, reasonably well, the flow field of this problem, we could proceed to the implementation of the RANS-based LRT (Lighthill Ray-Tracing) method from (da Silva, 2011) to start the aeroacoustics investigations.

One of the motivations of this study is to develop the capability of correctly predicting the noise generation and propagation by the exhaust gases from the APU of an aircraft in a crosswind condition. It is known that the APU is an important source of noise in the ground. The presence of the wind changes the characteristics of acoustic propagation interfering with the perception of sound and it is also shown that the emission of sound is affected by the shape, size and velocity of the fluid streams. Such study can be useful for the aircraft disposal in airports where crosswind is a relevant factor and ramp noise is a concern to passengers and workers.

2. A REVIEW ON CROSSFLOW FLUID DYNAMICS

The JICF (Jet In Crossflow) fluid dynamics allow a rapid mixing of the jets due to the complex three-dimensionality of the interactions between the vortices formed. Thanks to works such as (Kamotani and Greber, 1972) and (Fearn and Weston, 1978), the results of these interactions are coherent structures, well defined and known today. Figure 2 illustrate a jet under a cross-flow, in which some main flow structures can be identified.

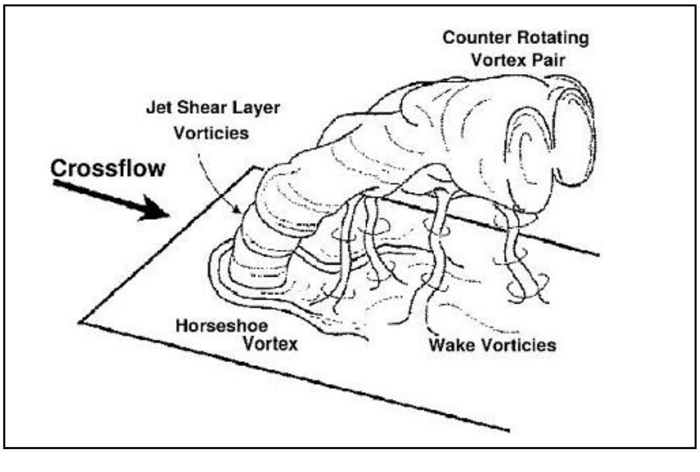


Figure 2. Schematization of the crossflow fluid dynamic, presented by (Jouhaud *et al*, 2007).

A vast number of investigations have been carried out regarding these structures and there is a common agreement of four major vortexes types as presented in (Jouhaud *et al*, 2007) and briefly described as follows:

- **Counter Rotating Vortex Pair**: it is the most dominant structure of the flow downstream of the nozzle and is formed by deflection of the jet and carried by the transversal flow. Is important to stress that this structure is responsible for most of the mixing between the jet and the crossflow wind. Experimental and numerical studies such as (Coelho and Hunt, 1989) and (Cortelezzi and Karagozian, 2001) prove such claims.

- **Horseshoe Vortex:** due to the adverse pressure gradient formed immediately in front of the jet, this structure is very similar to the vortex formed in the flow around a solid cylinder. It's shown in (Krothapalli *et al*, 1990) that even the detachment frequency of the vortices is very similar to the flow around a cylinder.

- **Shear layer ring vortices:** these vortices develop in the circumference of the jet and are generated in the shear layer by the encounter of the jet and the transversal flow, the instability take the form of a Kelvin-Helmholtz one but such vortices are distorted when the jet is deflected and its evolution was evaluated by (Kelso *et al*, 1996).

- **Wake vortices:** these structures have only been studied experimentally by (Moussa *et al*, 1977) and (Smith and Mungal, 1998), but are at least understood. However, the work of (Fric and Roshko, 1994) suggests that this vorticity originates in the boundary layer of the wall where it involves the jet and these flow patterns appear to be influenced by the jet trajectory.

3. PROBLEM DESCRIPTION

3.1 Flow Conditions

Table 1 presents the flow conditions for the jet flow investigated in this work. The data were taken from (Rudman, 1996) in order to validate the results obtained by the simulations presented in this work. The conditions were applied to a 3D domain with the same size as in (Rudman, 1996), as shown in next the sections.

Table 1. Flow conditions simulated.

U_j/U_c	U_j/c_∞	T_∞	U_j (m/s)	U_c (m/s)	c_∞ (m/s)	P_∞ (Pa)	ρ_∞ (Kg/m³)	P_0 (Pa)	T_0 (K)
5	0.4	291.2	135.1	27	337.75	102345	1.225	114273.5	300.55

3.2 Computacional Domain

The computational domain is an extension for the RANS simulation and is illustrated in Figure 3. The size of the domains in x, y and z coordinates was selected based on the work of (Rudman, 1996). The values shown below are considered suitable for such subsonic simulations and are used consistently through this whole work.

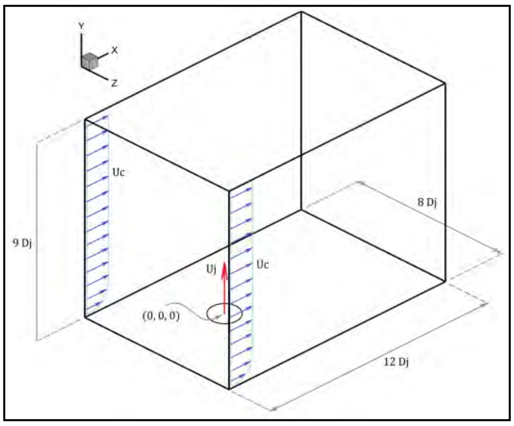


Figure 3. Computational domain based on the jet's diameter.

3.3 Boundary Conditions

The boundary conditions are depicted in Fig. 4. At the inlet (jet exhaustion), total pressure and total temperature of the jet are specified. At the inflow (plane $x = -3Dj$) the total pressure and total temperature of the crossflow are specified. The wall part (plane $y = 0$) has a no-slip adiabatic wall function applied. For the sides part (planes $z = -4Dj$ and $z = 4Dj$) is used a symmetry condition. The top part (plane $y = 9Dj$) is defined using a static pressure and temperature using inside velocity and finally, at the outflow (plane $x = 9Dj$), static pressure is specified. Values for pressures and temperatures were applied in accordance to Table 1.

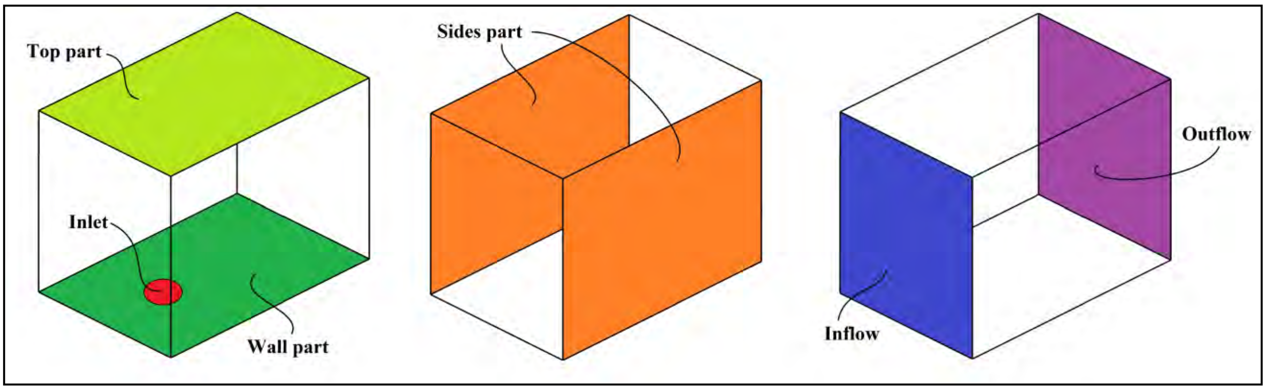


Figure 4. Boundary Conditions applied in the domain.

3.4 Computational mesh

The commercial software Ansys® IcemCFD was used to generate the structure meshes used on the simulations, which are shown in Fig.5. The discretization consisted of a block structured mesh with 20 blocks with 1,964,430 elements and 1,990,560 nodes, where can be seen the clustered of points on the desired high gradients region of the flow.

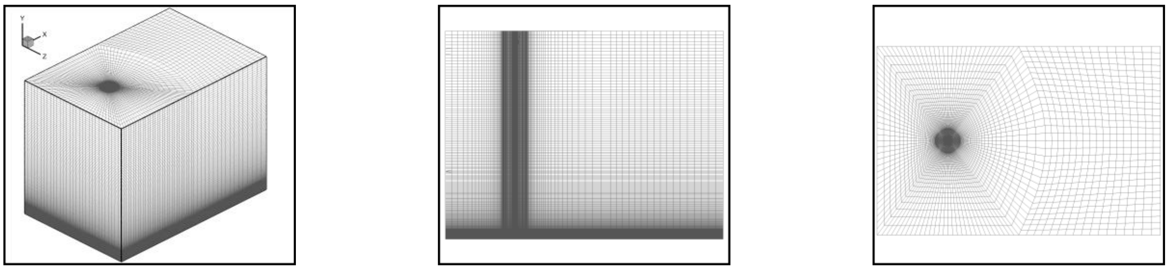

(a) 3D View of the mesh (b) XY plane in $Z = 0$ (c) XZ plane in $Y = 0$

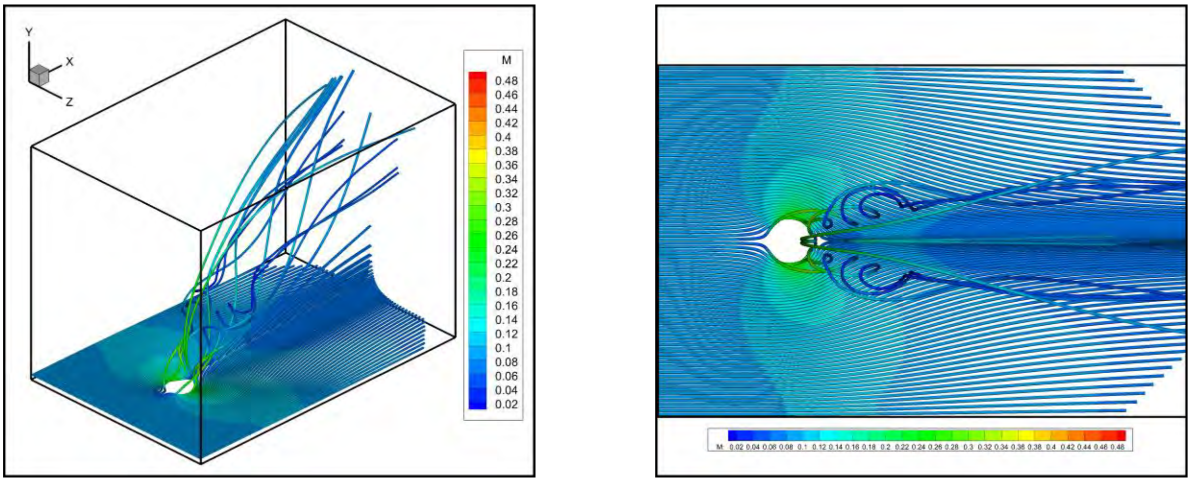
Figure 5. Mesh details in different views.

3.5 Numerical Scheme

A Reynolds Averaged Navier-Stokes (RANS) approach is used in this work. The compressible steady-state equations of motion are solved for the three dimensional domain. The governing equations were solved with a second order accuracy through a Finite Volume formulation employed in the CFD++ software. For all flows at Mach number 0.4, a preconditioning approach was necessary to stabilize the solution. The final result was obtained by running 5000 iterations, or when the residual dropped 5 orders of magnitude. For the turbulence modeling, the realizable k-ε model was chosen, based on experiences acquired in the past works with RANS equations applied to shear layer flows, (Souza and Almeida, 2011).

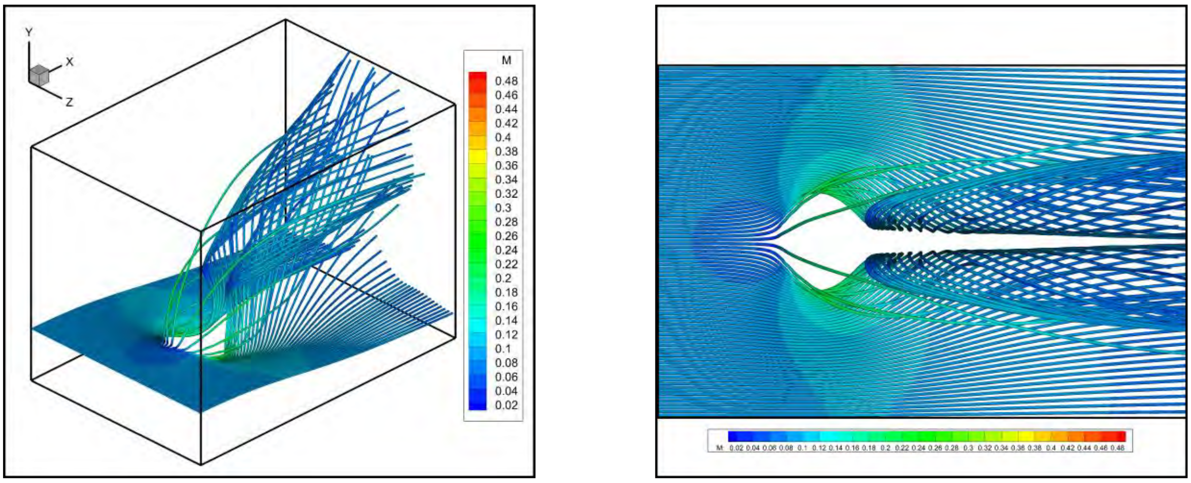
4. AERODYNAMIC RESULTS

This section is dedicated to present the results obtained for the simulation with the conditions as described in Table 1, and the comparison of these results with the work of (Rudman, 1996). The Figures 5 and 6 show the streamtraces entering the domain from the crossflow direction in $x = -3Dj$ in two different positions along the y axis ($y \approx 0$ and $y = 2Dj$).



(a) 3D view (b) Top view (Plane XZ)

Figure 6. Streamtraces from the domain's entrance at the inflow condition, in $y \approx 0$



(a) 3D view (b) Top view (Plane XZ)

Figure 7. Streamtraces from the domain's entrance at the Inflow condition, in $y = 2Dj$

The Counter Rotating Vortex Pair can be easily identified from the numerical results presented, especially in Figure 7. It's important to point out that a RANS simulation isn't capable of capturing the other vortices structures listed in the early sections of this paper, due to the time dependency of these other vortices types.

In order to compare the results from (Rudman, 1996) the same planes analyzed by the author are used herein. In his work, Rudman presents the planes $y = 2Dj$ and $x = 1.2Dj$ with velocity vectors, which length obey the velocity magnitude. Contour plots of pressure distribution was taken in the same planes and plot under the Rudman's velocity vectors so that it was possible to visualize the differences between both simulations.

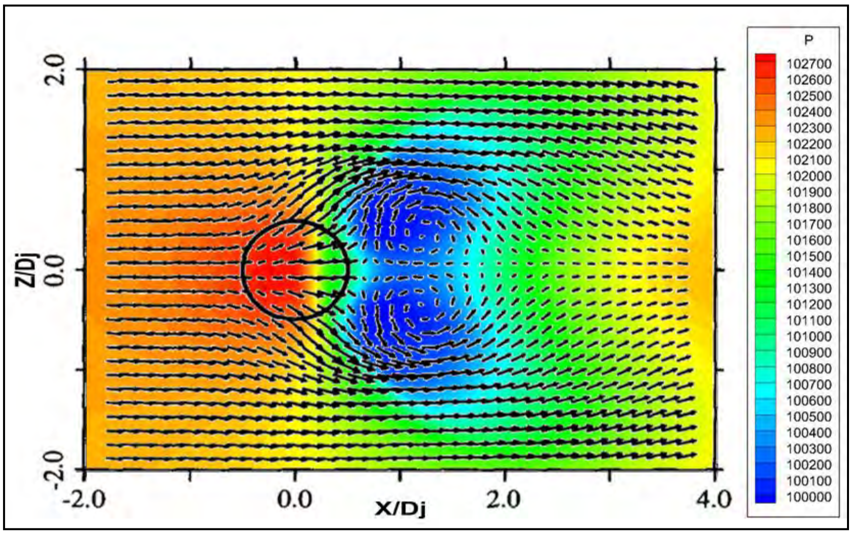


Figure 8. Pressure contour by the present RANS simulation made under velocity vectors generated by Rudman's DNS simulation, in *y = 2Dj*

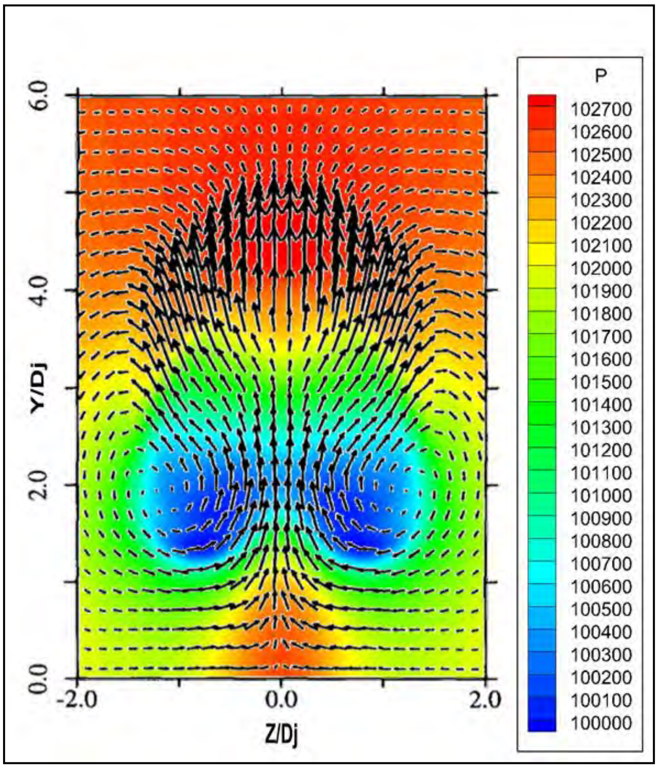


Figure 9. Pressure contour by the present RANS simulation made under velocity vectors generated by Rudman's DNS simulation, in *x = 1.2Dj*

The same method is used for the comparison in the plane *z = 0* in the next figure, but instead of a pressure contour it is used a Mach number contour, in order of better evaluation of the results.

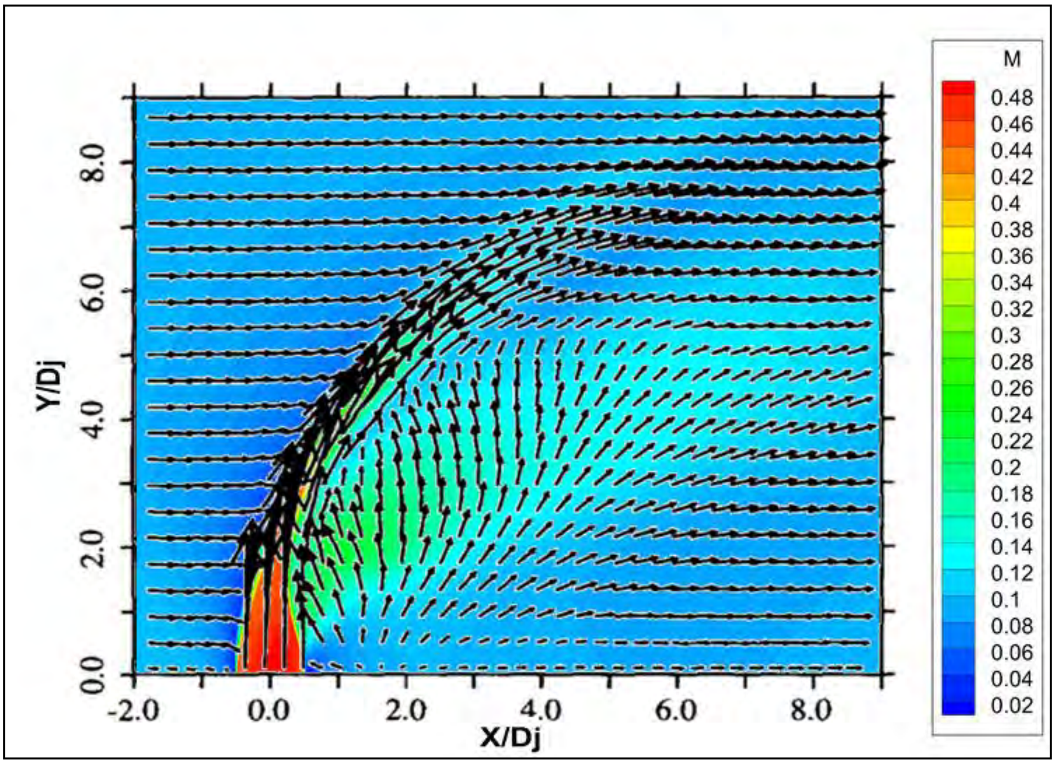


Figure 10. Mach number contour by the present RANS simulation made under velocity vectors generated by Rudman's DNS simulation, in $z = 0$

In a more quantitative result, Rudman present the jet's centerline, which is defined to be the locus of maximum velocity on the center plane ($z = 0$) of the jet in crossflow. The same centerline was measured for the present simulations and the result is shown in the figure below.

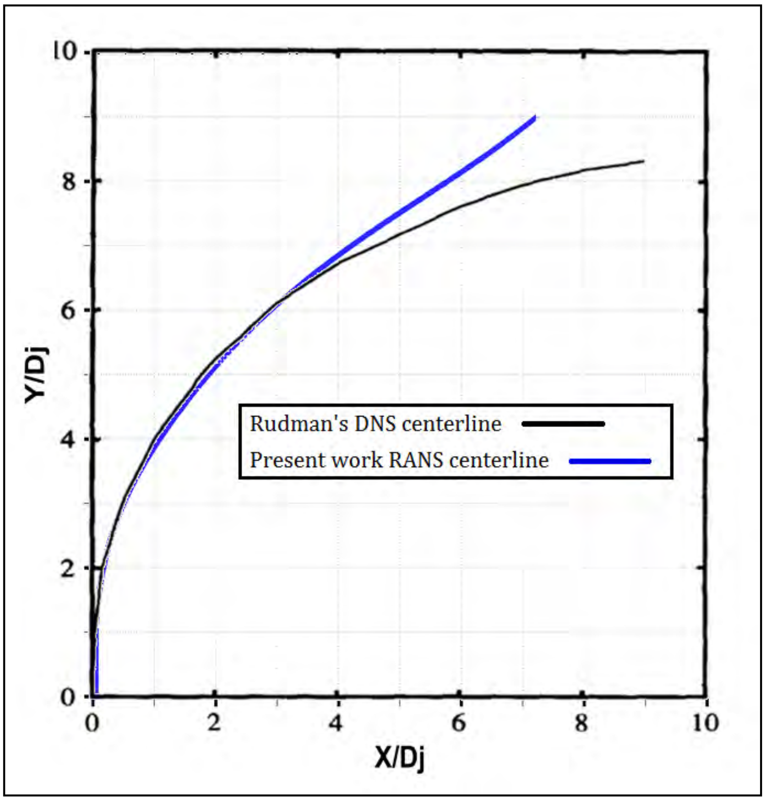


Figure 11. Jet's centerline, comparison between Rudman's DNS results and the present work

It can be noticed that above seven jet diameters in the *y* axis the curves start to differ from each other. It is believed that these discrepancies from the RANS simulations are due to the boundary condition proximity. The "static pressure and temperature using inside velocity" condition used in the top part (plane $y = 9Dj$) is too close from the jet exhaust. It's recommended that this condition be applied to the domain when it reaches the farfield.

5. CONCLUSION

This work showed the aerodynamics characterization of subsonic turbulent jets under the effects of crossflow by means of numerical simulations by using Reynolds-averaged Navier-Stokes (RANS) approach. The main objective of this work was to evaluate the capability of RANS to correctly predict the jet's potential core and its spreading rate, especially for high crossflow velocities. Such analysis is important and necessary for further aeroacoustics investigations by using the LRT (Lighthill Ray-Tracing) method. The numerical results have been compared with experimental and numerical data in Rudman (1996). It is reasonable to conclude that RANS showed very promising results and further improvements and analysis are currently underway. Finally, this work is part of an ongoing experimental and numerical research on subsonic jets in crossflow, currently being carried out in the Federal University of Uberlândia (UFU), and it will serve as basis for assembling the experimental facility for JICF analysis.

6. ACKNOWLEDGEMENTS

The authors of this work would like to thank Fundação de Amparo a Pesquisa do Estado de Minas Gerais – FAPEMIG.

7. REFERENCES

Chai, X. and Mahesh, K., "Simulations of high speed turbulent jets in crossflow", 49th AIAA Aerospace Sciences Meeting, AIAA-2011-650, 2011.

Coelho, S. L. V. and Hunt, J. C. R., "The Dynamics of the Near Field of Strong Jets in Crossflows," Journal of Fluid Mechanics, Vol. 200, 1989, pp. 95-120.

Cole, J.E., "The influence of a crossflow on jet noise", NASA CR-2169, November 1972.

Cortelezzi, L. and Karagozian, A. R., "On the Formation of the Counter-Rotating Vortex Pair in Transverse Jets," Journal of Fluid Mechanics, Vol. 446, 2001, pp. 347-373.

Cavar, D. and Meyer, K. E., "LES of turbulent jet in cross-flow: Part 1 – A numerical validation study", International Journal of Heat and Fluid Flow 36: 18-34, 2012.

Fearn, R. L. and Weston, R. P., "Induced Velocity Field of a Jet in a Crossflow", NASA, TP-1087, 1978.

Fric, T. F. and Roshko, A., "Vortical Structure in the Wake of a Transverse Jet", Journal of Fluid Mechanics, Vol. 279, 1994, pp. 1-47.

Gordier, R. L., "Studies on Fluid Jets Discharging Normally into Moving Liquid." St. Anthony Falls Hyd. Lab., Tech. Paper, No. 28, Series B, Aug. 1959.

Hasselbrink, E. F. and Mungal, M. G., "Transverse jets and flames Part 1: Scaling laws for strong transverse jets", J. Fluid Mech. 443, 1 2001

Huang, R. F. and Hsieh, R. H., "An experimental study of elevated round jets deflected in a crosswind", Experimental Thermal and Fluid Science 27, 77-86, 2002.

Camelier, I. and Karamcheti, K., "An experimental study of the structure and acoustic field of a jet in cross stream", NASA CR-162464, January 1976.

Jordinson, R., "Flow in a Jet Directed Normal to the Wind". R. and M. No. 3074, Aero. Res. Comm. (Great Britain), Oct. 1956.

Jouhaud, J. C., Gicquel, L. Y. M., Enaux, B. and Esteve M. J., "Large-Eddy-Simulation Modeling for Aerothermal Predictions Behind a Jet in Crossflow", AIAA Journal, vol. 45, no. 10, 2007, pp. 2438-2447.

Kamotani, Y. and Greber, I., "Experiments on a Turbulent Jet in Crossflow," AIAA Journal, Vol. 10, No. 11, 1972, pp. 1425-1429.

Karvinen, A. and Ahlstedt, H., "Comparison of turbulence models in case of jet in crossflow using commercial CFD code", Engineering Turbulence Modelling and Experiment 6, Elsevier, 2005.

Keffer, J. F. and Baines, W. D., "The Round Turbulent Jet in a Cross-Wind". J. Fluid Mech., Vol. 15, Pt. 4, 1963, pp. 481-496.

Kelso, R. M., Lim, T. T. and Perry, A. E., "An Experimental Study of Round Jets in Cross-Flow", Journal of Fluid Mechanics, Vol. 306, 1996, pp. 111-144.

Krothapalli, A., Lourenco, L. and Buchlin, J. M., "Separated Flow Upstream of a Jet in a Crossflow", AIAA Journal, Vol. 28, 1990, pp. 414-420.

Moussa, Z. M., Trischka, J. W. and Eskinazi, S., "The Near Field in the Mixing of a Round Jet with a Cross-Stream", Journal of Fluid Mechanics, V ol. 80, 1977, pp. 49-80.

Ribner, H. S., "Field of Flow About a Jet and Effects of Jets on Stability of Jet-Propelled Airplanes". NACA War. Rep. L 213, 1946.

Rudman, M., "Simulation of the Near Field of a Jet in a Cross Flow", Experimental Thermal and Fluid Science, Vol. 12, 1996, pp. 134-141.

Ruggeri, R. S., Callaghan, E. E. and Bowden, D. T., "Penetration of Air Jets Issuing from Circular, Square and Elliptical Orifices Directed Perpendicularly to an Air Stream". NACA TN 2019, Feb. 1950.

Silva, C. R. I., "Development of a novel RANS-based method for the computational aeroacoustics of high speed jets", Escola Politécnica da Universidade de São Paulo, Tese de Doutorado, 2011.

Smith, S. H. and Mungal, M. G., "Mixing, Structure and Scaling of the Jet", Journal of Fluid Mechanics, Vol. 357, 1998, pp. 82-122.

Souza, P.R.C. and Almeida, O., "Aerodynamics Characterization of a Subsonic Jet: Boundary Conditions Influence", Iberian Latin American Congress on Computational Methods in Engineering - CILAMCE XXXII, 2011.

Su L. K. and Mungal M. G., "Simultaneous measurements of scalar and velocity field evolution in turbulent crossflowing jets", J. Fluid Mech.513, 1 2004.

Yuan, L. L., Street, R. L. and Ferziger, J. H., "Large-eddy simulations of a round jet in crossflow", J. Fluid Mechanics, vol. 379, pp. 71-104, 1999.

8. RESPONSIBILITY NOTICE

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