SQUARE AND AXISYMMETRIC FREE JET FOR PROCESS SAFETY ANALYSIS: AN INITIAL INVESTIGATION

G. L. CABRIDE[1], E. S. FERREIRA JÚNIOR[1], S. S. V. VIANNA[1]

[1] State University of Campinas, Faculty of Chemical Engineering
E-mail contact: elmo@feq.unicamp.br, savio@feq.unicamp.br

ABSTRACT – Jet flow has several industrial applications like jet pumps, burners, ejectors as well as process safety. This work investigates square and axisymmetric free jet in order to verify the impact in the velocity profile. The geometry was built on ICEM software and discretized using unstructured mesh. The governing equation were solved numerically applying the finite volume method using upwind scheme for convective terms and central differentiation for diffusion terms. Initial results show that axisymmetric and square orifices have very similar free jet. This is not in accordance with the current understanding of high aspect ratio jets. On the other hand, the jet centreline decay is qualitatively well represented. Further investigation concerning the influence of mesh type and size is demanded.

1. INTRODUCTION

Free jet flow type occurs when fluid is expanded through a nozzle or orifice into an environment in which the flow is not directly affected by an object. In this circumstance, the velocity profile of a free jet is influenced by the orifice geometry, characteristic of the discharge and ambient fluid. It has been extensive studied in experimental and numerical ways (Chang, 2002; Klein et al., 2003) and industrial applications comprise jet pumps, burners, mixers or ejectors (Decker et al., 2011) as well as process safety (Ferreira Jr. et al, 2014).

This paper is focused on the application of ANSYS-CFX code to conduct square and circular free jet simulation in order to compare and verify the impact of orifice shape in the near and far field jet behaviour. Comparison with experimental free jet obtained in a three dimensional Laser-Doppler-Anemometry system, which is described by Miltiner et al., 2014, is performed. Numerical findings are compared with the experimental and it serves as an initial indication of the numerical parameters which need to be addressed to improve the results.

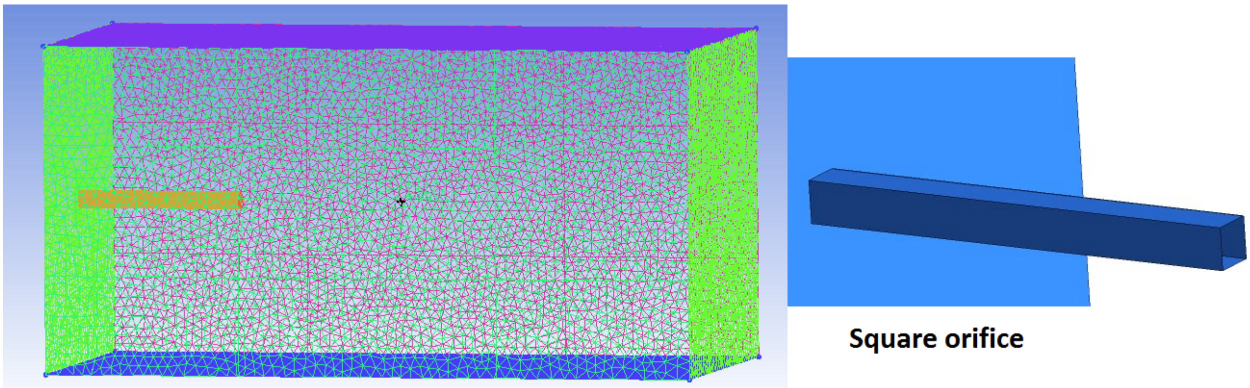
2. MATERIALS AND METHODS

2.1 Numerical set up and simulation

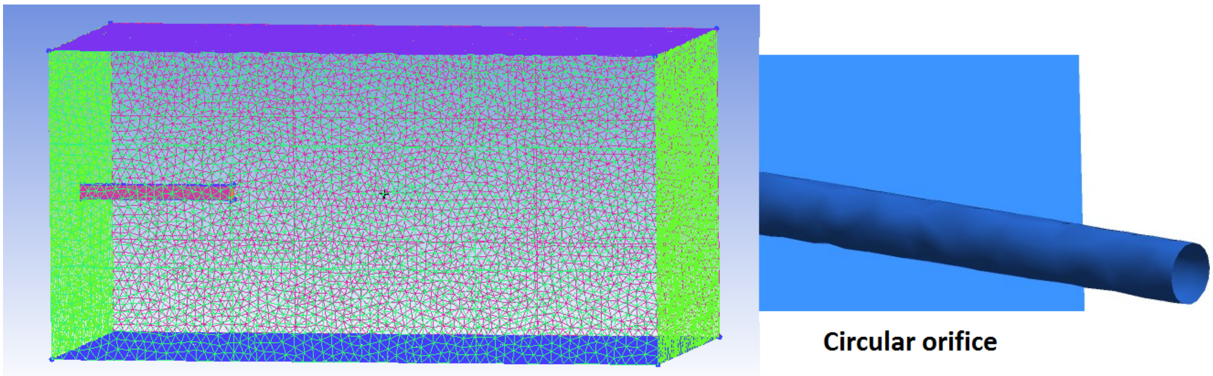
CFD simulations have been performed applying the commercial solver ANSYS-CFX© 14.0. Computational mesh sensitivity has been performed and independent mesh set up is applied. It comprises $52.7x10^4$ and $44.8x10^4$ volume cells with a strong structural refinement around the nozzle outlet, for circular and square orifice, respectively. Outlet boundaries have been set up to pressure of 0 Pa. A uniform velocity inlet of 5.22 m/s with temperature of air at 35°C has been applied in a circular diameter of 104 mm and in an equivalent square diameter of 92.14 mm. Solid walls have been set up applying the non-slip wall condition. Turbulence was modelling using k-epsilon approach and second order upwind scheme was chosen.

The geometry and the set up in the CFD code were based on experimental and numerical investigation performed by Miltiner et al., 2014. Figure 1 shows the geometric characteristics modelled in ICEM software for square (a) and circular orifice (b).

Figure 1 – Geometric characteristics for numerical investigation. (a) Square orifice. (b) Circular orifice.



(a)

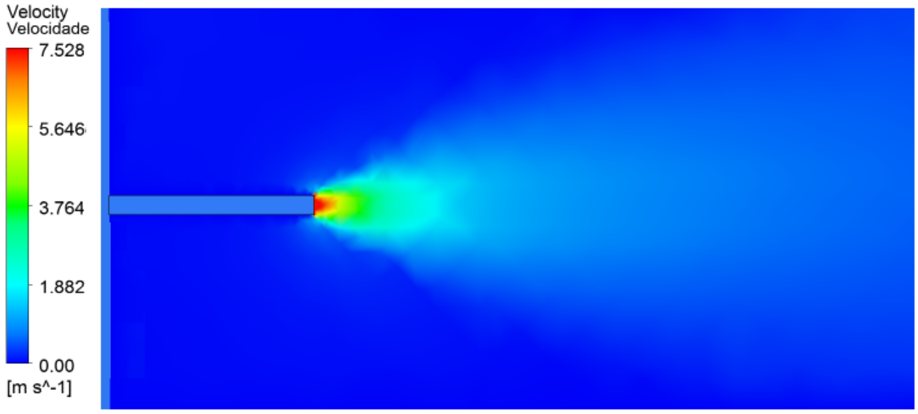


(b)

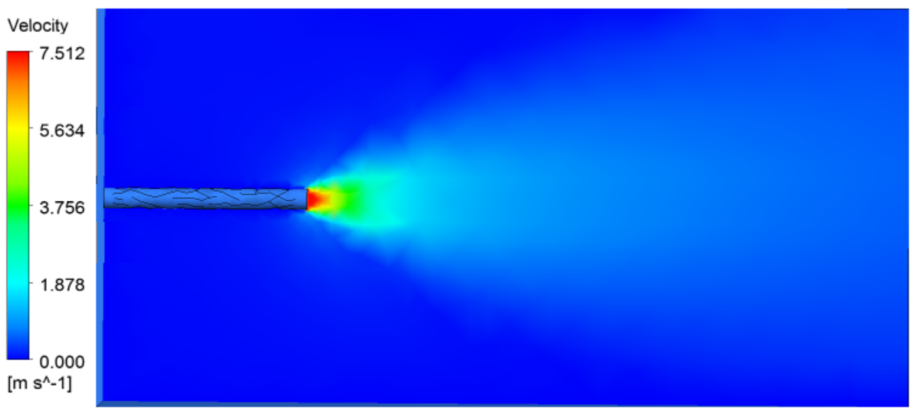
3. RESULTS AND DISCUSSIONS

Figure 2 shows a slice at 1000 mm in the y direction to the velocity profile of the turbulent free jet. Square and axisymmetric jet release orifices are shown in Figure 2 (a) and Figure 2 (b). Qualitative comparison between both releases shows very similar centreline velocity profile. It can be seen by gradient colours results. The red colour means the greater velocity and the blue colour means the lower velocity. It is important to say that square and circular jet have not the same characteristics (e.g. streamwise and crosswise rms velocities, turbulence intensity ratio), however, there are similarity in the centreline velocity decay in both shapes (HASHIEHBAF, 2014).

Figure 2 – Slice at 1000 mm in the y direction to the velocity profile of the turbulent jet. (a) Screenshot of the square jet release. (b) Screenshot of the circular jet release.



(a)



(b)

Quantitative analysis confirmed the similar profile for both square and circular orifices. This behaviour is not in accordance with the current understanding of high aspect ratio jet as far as numerical and experimental investigations are concerned (Wakes, 2011, Godoy 2014). Figure 3 shows the jet centreline velocity decay for both cases. Analysis of the plot shows no significant difference in the numerical findings.

Figure 4 shows the near field numerical results and compare them with experimental data. The difference between numerical findings and experimental data is reduced at distances further down the jet leak. In general the agreement is not satisfactory. On the other hand, the qualitative trend is well captured as reported by (Birth et el., 1987).

Analysis of the results indicates that further mesh assessment is required. There seems also to be indication that no significant difference between the square and circular leak is noticed. This is due to small difference among the dimensions considered in the hole size (square case). As a consequence the characteristics of the hight aspect ratio jet are obscured by the approximation of the effective leak area considered. Previous numerical simulations have shown good agreement with experimental data using the same CFD tool (Godoy, 2014). It is therefore demanded further investigation in the meshing process.

Figure 3 – Velocity profile of a turbulent jet. Comparison between jet velocity profile using square and circular orifice.

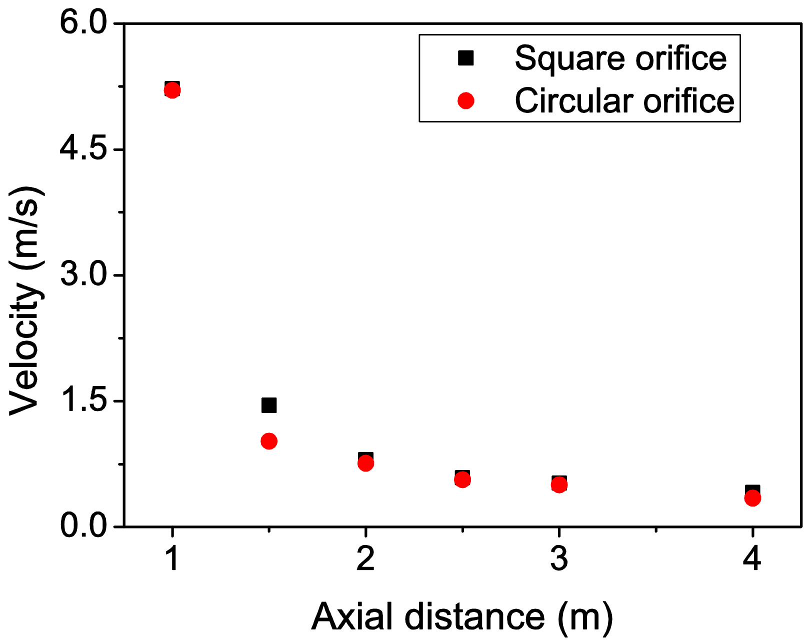
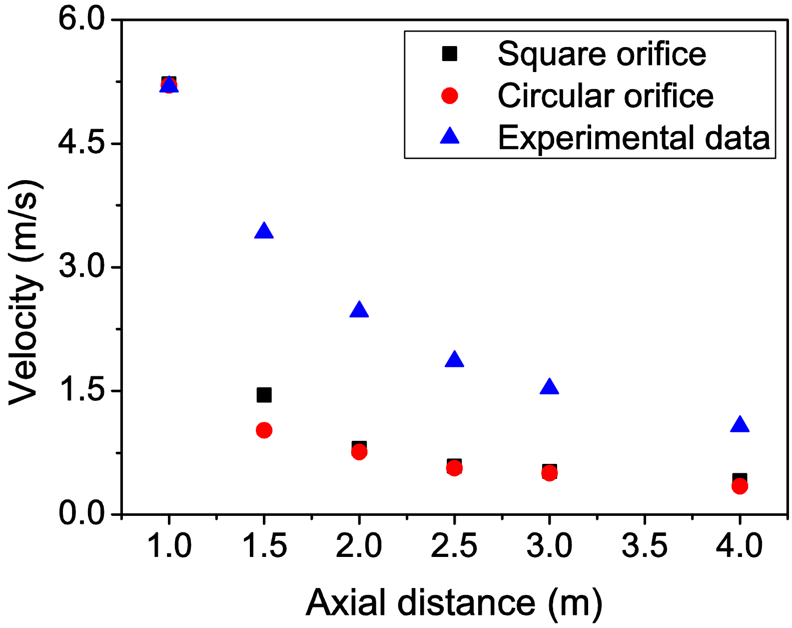


Figure 4 - Comparison between jet velocity profile using square and circular orifice and experimental data.



4. CONCLUSION

This work has presented the comparison between square and circular orifice in order to verify their impact on the jet centreline velocity decay. Taking into account the current understanding of high aspect ratio jet it is clear that the dimension considered in the square geometry must be reviewed. It is also important to verify the meshing process as well as the influence of the mesh when capturing the effects of the geometry shape.

As far as the circular leak is considered, it was also noticed that the numerical modelling was not able to capture the phenomena as it is observed in the experimental data. Further investigation concerning the CFD set up is needed.

Overall, this preliminary attempt was very important as the comparison with experimental data indicated key points that must be tackled in order to improve the CFD modelling. Based on the literature review it is clear from this work that the modelling of the shape of the leak for square releases must be addressed with care.

Concerning the axisymmetric, jet further investigation of the mesh size is demanded. It is also of paramount importance to consider an additional set of experimental data as a matter of benchmarking.

5. REFERENCES

CHANG, I. Unsteady-State Underexpanded Jet Flows, *AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit.* p. 1-12, 2002.

DECKER, R.; BUSS L.; WIGGERS, V.; NORILER, D.; REINEHR, E.; MEIER, H.; MARTIGNONI, W.; MORI M. Numerical Validation of a Coaxial and Confined Jet Flow, *Chem. Eng. Trans*, v. 24, p. 1459-1464, 2011

FERREIRA JR, E. S.; VIANNA, S. S. V. A novel and free advanced large eddy simulation computational fluid dynamics tool for gas dispersion. *Int. J. of Model. and Simulation. for the Petrol. Ind.,* v. 8, p. 1-6, 2014.

HASHIEHBAF, A.; ROMANO G. P. Experimental investigation on circular and non-circular synthetic jets issuing from sharp edge orifices. International Symposium on Applications of Laser Techniques to Fluid Mechanics, 2014.

KLEIN, M., SADIKI, A., JANICKA J. Investigation of the influence of the Reynolds number on a plane jet using direct numerical simulation. *Int. J. Heat and Fluid Flow*, vol.24, pp.785–794, 2003.

MILTNER, M.; JORDAN, C.; HARASEK, M. On the Applicability of RANS-Turbulence Models in CFD Simulations for the Description of Turbulent Free Jets during Biomass Combustion. C*hem. Eng. Trans*., v. 39, 2014.

BIRCH, A . D., HUGHES, D. J., SWAF F. F. Velocity decay of high pressure jets. 1563-521X. *Combus. Sci. Tech*., v.52(1), p.161-171. 1987.

WAKES, S. J.; HOLD Ø, A. E.; MEARES , A. J. Experimental investigation of the effect orifice shape and fluid pressure has on high aspect ratio cross sectional jet behavior. Journal of Hazardous Materials, A.89, p.1-27, 2001.

GODOY, C. T.D. FERREIRA., S. S.V. VIANNA. O Efeito da Geometria do Vazamento na Nuvem de Gás Inflamável: Uma Análise sob a Ótica da Segurança de Processo. Anais do COBEQ 2014. Florianópolis. SC. Brasil

7. ACKNOWLEDGMENT

Thanks for CAPES, UNICAMP, FEQ, L4R1S4 laboratory.