24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-2150
TURBULENT STRUCTURES IN A FLOW VISUALIZATION INSIDE A CONICAL DIFFUSER

José Leandro Cardoso Rivera Vila[1]
Lucas Henrique Vieira Dias[2]
Pedro Henrique de Carvalho de Mattos[3]
Pedro Paulo Silva de Almeida[4]
Tiago de Melo[5]
Centro Universitário do Distrito Federal – UDF, SEP/SUL EQ704 / 904 Conj.A – Brasília / DF - CEP 70390-045
[1]jl.vila@hotmail.com, [2]lucashenriquevd@hotmail.com,[3]phmattos97@gmail, [4]almeidappsa@gmail.com, [5]tiago.melo@udf.edu.br

Abstract. *The flow's analysis to high Reynolds numbers is performed aiming the determination of both qualitative and quantitative parameters of the wake's vortex turbulence. Running tests on transparent conical diffusers made of plexiglass, one with a conical geometry, one with a circular and the other with a square inlet. The turbulence in the diffusers was generated by a secondary faster flow, in which was injected to the major flow perpendicularly. For this research both experimental setups and numerical simulations were run with the intent of analyzing how the flow behaves when subjected to such variables; the variables involved were diffuser geometry, flow velocity and the number of Reynolds, this last ranging from $4.224x10^3$ to $23.251x10^3$. The experimental tests were conducted on a experimental setup with didactic natures in order to obtain the experimental visualization of the turbulent flow; the numerical simulations were executed in the Ansys 18.1 software by making the use of the CFX analysis system, with the intent of analyzing how the involved conditions influence the flow inside the diffuser.*

Keywords: *Flow,Turbulence, Vortex, Visualization, Setup*

1. INTRODUCTION

The study of the flow in diffusers of hydro-generators arouses attention from uncountable researchers for quite a few time now, Chang and Tavoularis (2012), being object of many experimental and numerical works due the importance this kind of flow has in many characteristic applications of engineering. Diffusers are elements comparable to channels and ducts, these are important structures for engineering due its unique attributes of energy, velocity and pressure conversion with wide application in various mechanical equipments and projects as for example exhausters, gas turbine combustors and rocket nozzles.

Studies on these simple structures are essential to analyze and comprehend the processes involved, so that the gathered data on these enable the project of more complex and greater scale systems. For Angulo (2016), this kind of structure is one of the hardest parts to describe in a flow, due the interaction of many complex phenomena such as instability, turbulence, lines of current, secondary flow, swirl and cavitation.

The detachment's visualization of vortices and lines of current are areas with such importance to the determination of a profile to a fluid's flow behavior. Nonetheless, how certain characteristics such as the geometry aspect where the flow is passing through and the Reynolds numbers affects these vortices are a vast field of research as well. This way, understanding the flow behavior and the turbulent structures that are generated on the inside of these channels and how its geometry has a part in it can lead to better designs and more efficient equipment. Authors like Choueri and Tavoularis (2014), have dedicated attention to investigate experimentally the flow development, so to have a better understanding on how the flow behaves.

Turbulence in turn, apparently is a kind of chaotic flow, but as studied by authors such as White (2000), and Panofsky and Dutton (1984), it is proved that it's the case of a regular, static and periodic flow. By Johnson and Patel (1999), although the great progress achieved around this kind of flow, obtained fundamentally by experimental methods, there are several phenomenological aspects not comprehended in its totality, mainly in relation to the transition of the turbulent wake and vortex generating mechanisms.

The objective of this research is to visualize the formation of the vortices and the turbulence suffered in a conical diffuser in a macroscopic scale. To do so, tests were run in both experimental setups, as done by authors like Vila, *et.al.*, (2017), and Chen, *et.al*., (2014), and numerical analysis, as done by Coelho, *et.al*., (2006), and Neiva, *et.al*., (2007), and where the analysis were made in CFX commercial package with the purpose of comparing the data obtained in these numerical tests with the visualizations realized in the experimental setup. While conducting the tests properties such as the diffuser's geometry, flow's velocity and Reynolds numbers, this was used from $4.224x10^3$ to $23.251x10^3$, were altered to provide various conditions of analysis, generating data from these tests to furthermore have a better understanding on a flow's behavior when subjected to such conditions.

2. EXPERIMENTAL METHODOLOGY

The tests were conducted on a experimental setup with educational purposes, projected by Vila, *et.al.*, (2017). This setup, see Fig.01, manages to have the fluid primarily undisturbed, laminar flow like, so a pump produces axial flow which passes through the diffuser, a transparent conical part so the macroscopic analysis is possible. This setup is powered by two 34 watts pumps and two 20 liters water tanks, the velocity and the Reynolds number are manipulated by monitoring the flow rate with a flow water sensor of ½ programmed in Arduino and controlled by registers and dimmers connected to the pumps, and also by utilizing two different geometries for the diffuser, one being conical with circular inlet, and the other being conical with a square inlet; the diffuser utilized has an inlet diameter of 20 mm and outlet diameter of 32 mm and the structure has a length of 250 mm.

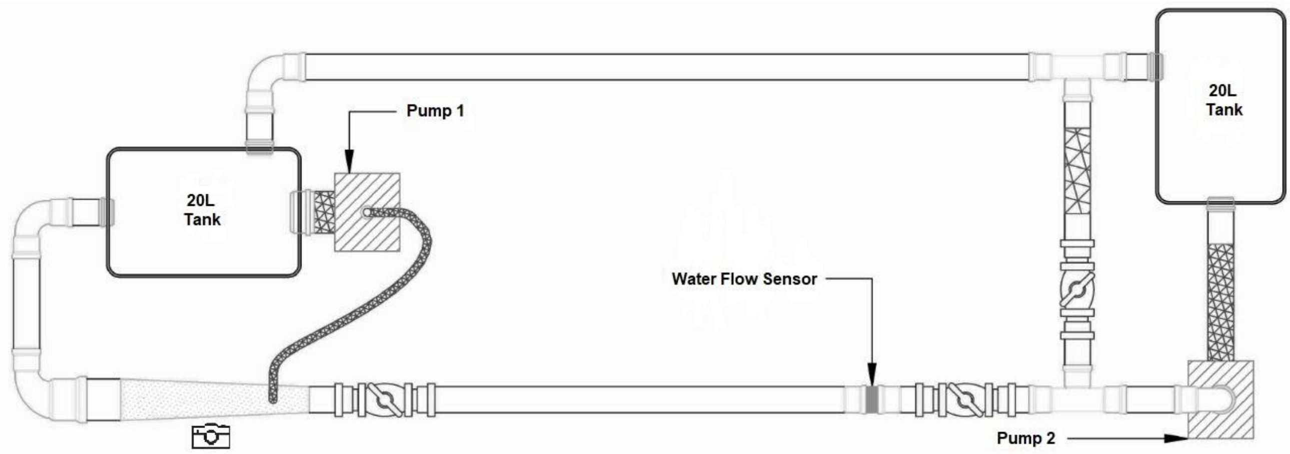

Figure 1. Schematic representation of the experimental setup

To generate the turbulence inside the diffuser, the injection of a second flow of the same fluid was used, this second flow strikes the diffuser perpendicularly, this way the main laminar flow is disturbed by this added flow inducing the formation of a turbulent movement downstream. To make the visualization possible the dye wash technique, see Fig.02, was used, this being a technique used by many authors like Price, *et.al*., (2002) and Chen, *et.al*., (2015), this consists in the injection of a second different fluid to the first, this being responsible of highlighting the lines of current present in the movement, for this matter a red PVA dye was used. The visualization was photographed with the assistance of a camera, Nikon Coolpix P510 of 16 megapixels, supported by a tripod so the images produced keep the same focus, making the analysis more precise, detailed and simple for the visualization. The velocity of the major flow used in the tests ranged from $0.212\ ms^{-1}$ to $1.167\ ms^{-1}$, in the case of the secondary flow it was fixed at $3.464\ ms^{-1}$; the tests were conducted with different Reynolds ranging from $4.224x10^3$ to $23.251x10^3$.

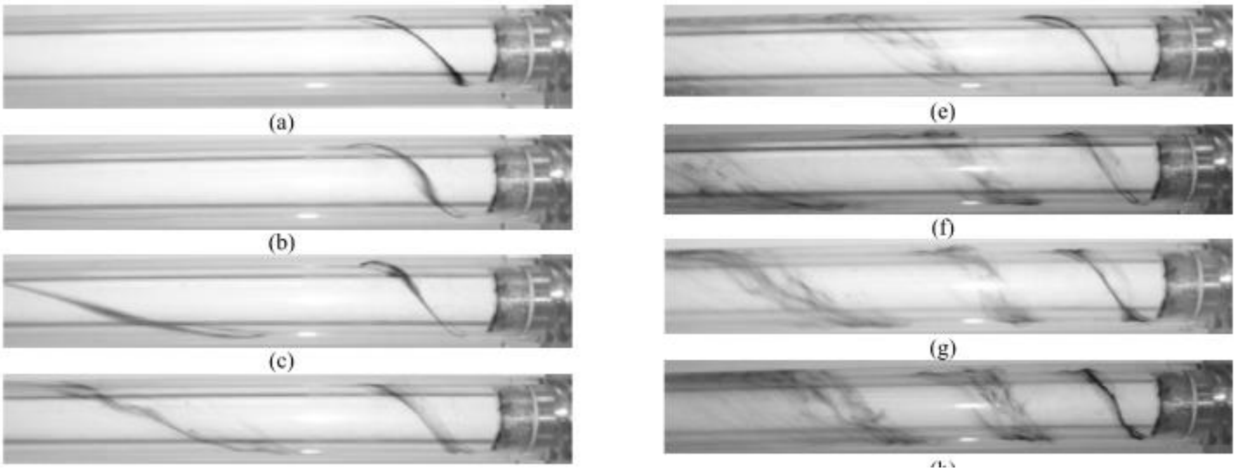

Figure 2. Flow visualization images by Chen, *et.al.*, (2015)

3. GOVERNING EQUATIONS AND NUMERICAL ANALYSIS

The turbulent flow generated in the workbench, is a kind of flow's behavior which is originated by instabilities in a laminar flow to high Reynolds numbers, this new regime is inevitable and it is a fluctuating and disorderly motion, White (2000). This type of movement has no consensus for its characterization around all of its aspects, Moller and Silvestrini (2004), although there is the classic way of qualifying this flow where the factors taken in consideration are the irregularity, diffusivity, high Reynolds numbers, three-dimensionality of vorticity, dissipation and the continuous environment, Tennekes and Lumley (1972). The irregularity refers to the matter that the tools available at the moment make a deterministic analysis of the turbulence something impossible, however there is a useful and promising solution, which consists in analyzing the displacement equations or experimental data of statistic manner, Cant and Mastorakos (2008).

3.1 Governing Equations

On the mathematical side, the basic equations which govern the turbulent movement, for incompressible fluids, which is the case of the water, are the continuity equations, which are simply developed by the law of conservation of mass to a small element, and the momentum equations, Coelho, *et.al.*, (2006), or Navier-Stokes equations, respectively given by Eq. (1) and Eq. (2); there is also the equation for the conservation of the passive scalar, which is Eq. (3).

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \cdot \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \cdot \frac{\partial p}{\partial x_i} + \nu \cdot \frac{\partial^2 u_i}{\partial x_j \partial x_j} \tag{2}$$

$$\frac{\partial \theta}{\partial t} + u_j \cdot \frac{\partial \theta}{\partial x_j} = k \cdot \frac{\partial^2 \theta}{\partial x_j \partial x_j} \tag{3}$$

Where, in Eqs. (1,2,3), u_i, p, θ, ν, k and ρ are respectively the components of velocity, pressure, passive scalar, molecular viscosity, molecular diffusivity and specific mass. Furthermore, these equations shall be complemented with the purpose of restricting the values, those being the initial and contour conditions. In equation (2), the viscous term, τ_{ij}, answers to the viscous tensions divergent, which when simplified by using the equation (1), incompressibility condition, results in Eq. (4).

$$\tau_{ij} = \mu \cdot \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \rightarrow \frac{\partial \tau_{ij}}{\partial x_j} = \mu \cdot \frac{\partial^2 u_i}{\partial x_i \partial x_i} \tag{4}$$

In Eq. (4), μ is the dynamic viscosity. The same way done for equation (4), to equation (3), the diffusive term, q_j, answers to the scalars flux divergent, given by Eq. (5). In equation (5), *k*, is the thermal conductivity.

$$q_j = -k\left(\frac{\partial \theta}{\partial x_j}\right) \rightarrow -\frac{\partial q_j}{\partial x_j} = k \cdot \frac{\partial^2 \theta}{\partial x_j \partial x_j} \tag{5}$$

In a turbulent movement there is the appearance of swirls in the flow, that in fact cause an agitation of mixing the fluid, being this one of the diverse applications of the turbulent flow, in this swirls there is there presence of the turbulence wake and vortices. The vortices, which are related to the turbulence in the flow, as studied by Schlichting (1979), analyzing a immerse cylinder with the purpose of describing its formation, these formation are conceded by the separation of the limit layer above the immerse body, where this disconnection has the function of distributing the accumulated pressure on this layer. This way, analyzing the flow when passing through a cylinder it was shown that when the detachment occurs due to the frictional force causing the kinetic force to be consumed, this way not remaining enough energy to the increase of pressure, in function of the extern pressure some particles then start moving in an opposite way to the flow; this reverse movement is where vortices are generated. By a mathematical analysis, the vorticity is given by Eq. (6).

$$\Omega_i = \varepsilon_{ijk} \cdot \frac{\partial u_k}{\partial x_j} \tag{6}$$

Moreover, by applying the rotation operator to equation (6), is results in Eq. (7). In equation (7), its left side represents the terms of local variation and convective transport of vorticity, already the right arm describes, by $\vec{\Omega} \cdot \nabla \vec{u}$, the vorticity variation by torsion or extension of a line of vortices, by $\nu \nabla^2 \vec{\Omega}$, the variation tax of $\vec{\Omega}$ due the vorticity's viscous diffusion.

$$\frac{D\vec{\Omega}}{Dt} = \vec{\Omega} \cdot \nabla \vec{u} + \nu \nabla^2 \vec{\Omega} \tag{7}$$

3.2 Numerical Analysis

The numerical analysis of the flow's dynamic was conducted to a qualitative comparison of the generated images both experimentally and computationally to generate a better comprehension around the processes in study inside the diffuser. The numerical simulations were obtained by the CFX commercial package present in Ansys 18.1 software.

The analysis system used was the CFX logic tree inside Ansys Workbench, where the intent is to project a geometry to be simulated on a flow, this geometry was modelled inside the Spaceclaim 18.1. The geometries designed for this research were the two utilized in the experimental setup, so then to have data to stablish a link between both numerical and experimental tests, being the conical diffuser a cone trunk like structure with both inlet, 20 mm, and outlet, 32 mm, having a circular format, and the other being a diffuser with both inlet and outlet having a square format with a hydraulic diameters of respectively 20 mm and 32 mm; the length of these is equal to 250 mm, although a structure starting in the outlet of both structures with a length of 190 mm, dimension gathered from the setup, was designed so the flow is able to behave more smoothly. The mesh details for the tests can be seen on Tab.01, it was refined utilizing the refinement mesh component system in the fluid flow logic tree, where the definition of refinement was 3 for the inlets and outlets, and 1 for the diffuser body, this way the mesh acquires a more dense and detailed in the areas of most interest of this analysis, which are the inlets and outlets.

Tab 01. Mesh details: number of nods and elements

Geometry	Nods	Elements
Cone trunk like diffuser	74951	386522
Pyramid trunk like diffuser	60633	314687

The numerical simulation was based on the data obtained by the experimental setup, these velocities were applicated to the analysis in the CFX-pre component system. The conducted simulations utilized the following velocities, 0.212 m/s, 0.530 m/s, 0.818 m/s and 1.167 m/s for the inlet of the major flow, for the secondary flow responsible of causing the turbulence the velocity used was 3.464 m/s.

On the analysis part, conducted on the CFX-post component system, the use of the countour function was applied to demonstrate the development of both velocity and pressure inside the diffuser with the intent of verifying the primary characteristics of this kind of structure which is the conversion of energy from kinetic to potential. Furthermore to the visualization of the lines of current and the turbulent structures the streamline function was used, with a number of 20 points.

4. RESULTS AND DISCUSSION

The outcome of the experimental tests conducted on the workbench was a series of images, see Figs.3,4,5,6, gathered from the visualization of the vortices and the turbulent structures generated on the flow by the addition of a faster secondary flow.

Figure 3. Visualization of the turbulent flow in the circular inlet diffuser for a Re = $4.224x10^3$

Figure 4. Visualization of the turbulent flow in the square inlet diffuser for a Re = $4.224x10^3$

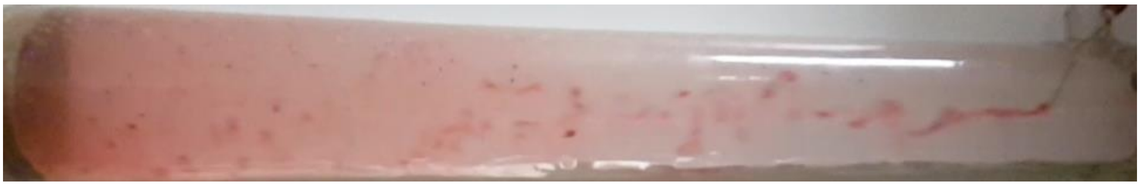

Figure 5. Visualization of the turbulent flow in the circular inlet diffuser for a Re = $23.251x10^3$

Figure 6. Visualization of the turbulent flow in the square inlet diffuser for a Re = $23.251x10^3$

It is seen by the exhibited images, Figs.3-6, that the high Reynolds number have such influence on how the flow behaves. The test with Reynolds of $4.224x10^3$, which is almost the double needed to turn a flow into a turbulent one, as stated by Brunetti (2008), shows the structures generated in the process, although it is seen that from the middle of the diffuser the flow is barely visible, this is caused by the dilution of the used dye, by the velocity of the flow rate utilized which was of 0,212 m/s, and also by the quality of the images captured. As the Reynolds increases is visible that the flow becomes more turbulent, as seen on the tests with Reynolds of $23.251x10^3$, where the camera could not focus on the lines of flow. It is also important to note that the lines of current are not exactly moving continuously on the diffusers wall due to the dye's density; with time, the accumulation of dye also harmed the visualization, as seen on Figs.4,6.

When comparing both diffusers it is clear that the square inlet one slows the flow with more intensity, this is due its geometry which does not let the flow run downstream smoothly, differently from the circular one. This comparison is also represented in Figs.7,8, generated in Ansys utilizing the countour function in the CFX-post with 100 points. This observation leads to a perception that the lines of current in the second diffuser tend to be more similar to laminar than the conical one, the idea that this diffuser has less influence on the generation of turbulence movement is also conclusive.

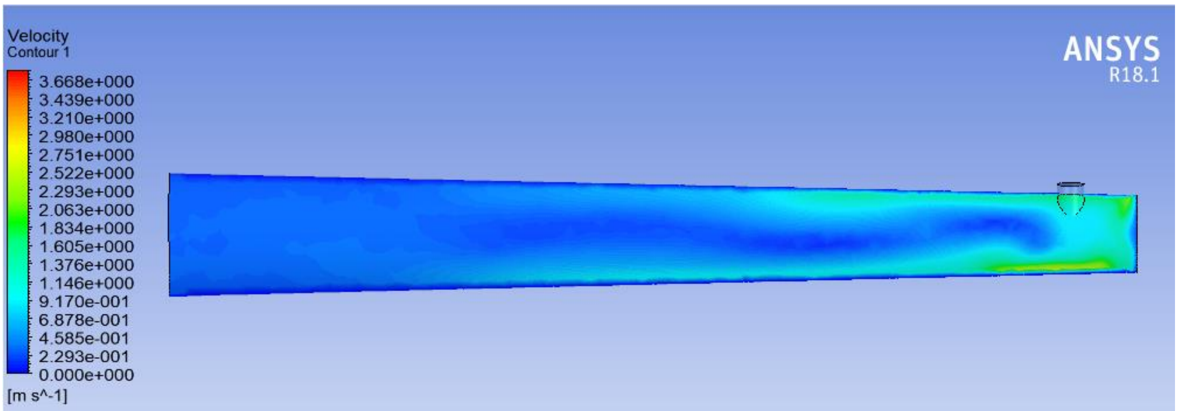

Figure 7. Development of the downstream velocity for the circular inlet diffuser

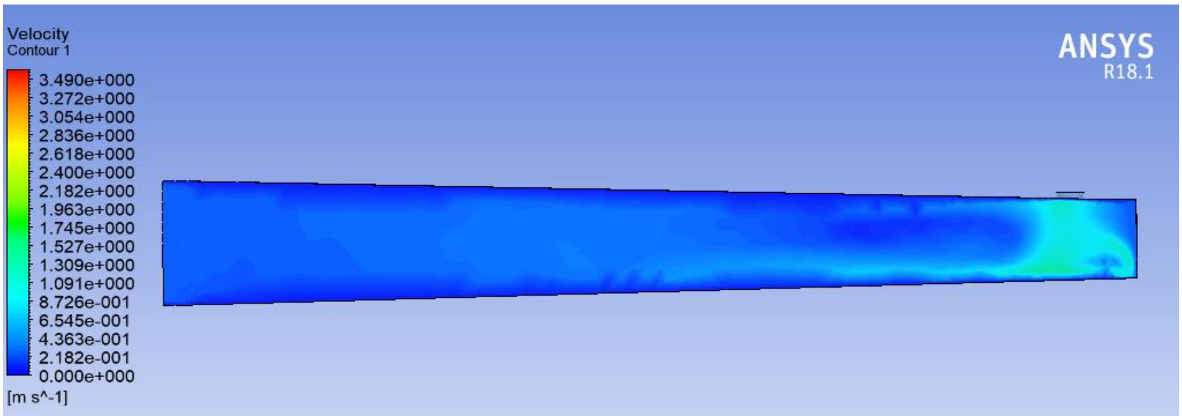

Figure 8. Development of the downstream velocity for the square inlet diffuser

Aside the velocity contour analysis, a study on the visualization of the lines of flow was also conducted by making the use of the 3D streamline function present in the CFX-post. Differently as done for the velocity case in contour, the amount of points used for the streamline was reduced to 20 to ease the visualization, otherwise only the external face of the flow would be visible. The conical diffuser with the circular inlet is shown in Fig.9, for a Reynolds of 4.224x10^3 with an inlet velocity of 0.212 m/s, note that the second flow utilized in the experimental tests was fixed at 3.464 m/s; with the same conditions of simulation, the diffuser with the square inlet is represented in Fig.10.

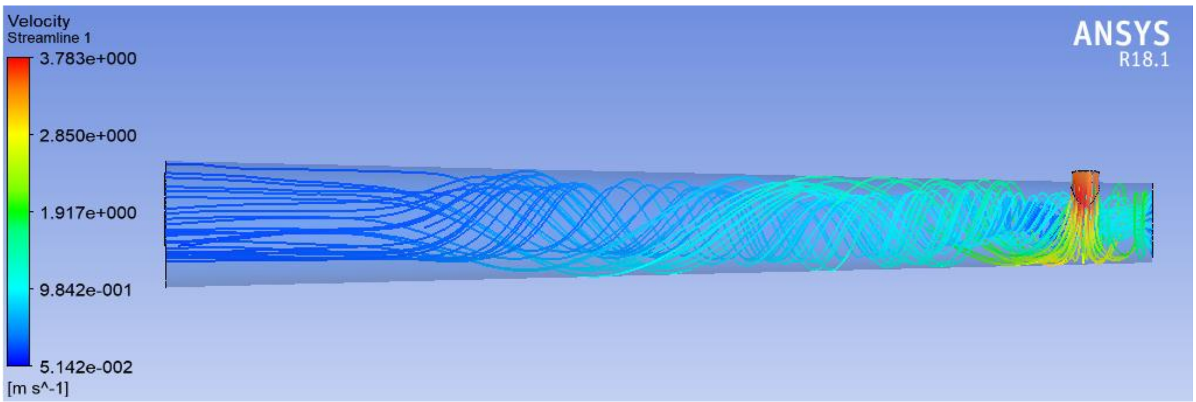

Figure 9. Streamline of the conical circular inlet diffuser with a velocity of 0.212 m/s

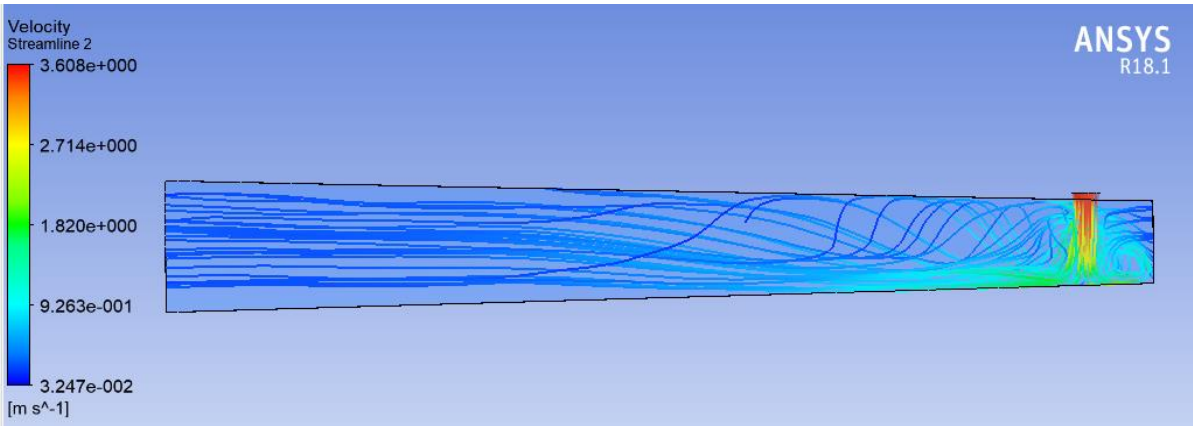

Figure 10. Streamline of the conical square inlet diffuser with a velocity of 0.212 m/s

By the streamline analysis, it is possible to visualize the lines of current and the presence of the turbulent structures. It is also notable, a start of the turbulence inside the diffuser with the presence of swirls, although due the applied the conditions it would be more evident near the diffuser's inlet.

Comparing the experimental and the numerical simulations it is observable that the conditions of Reynolds and velocity utilized in the studied flow are responsible for the low quality visualization in the experimental setup, and not only the dye wash fault. For this matter, exploring this possibility even further, simulations with two times the original velocity, 6.928 m/s, see Figs.11,12, were run with the intent of analyzing the influence of the secondary faster flow being even faster; note that the main flow had its velocity maintained at 0.212 m/s.

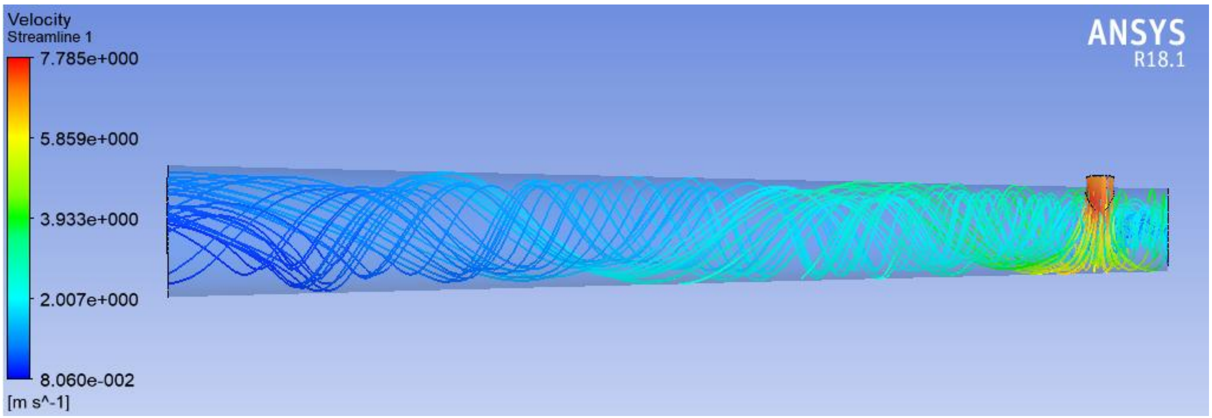

Figure 11. Streamline of the conical circular inlet diffuser with a secondary flow velocity of 6.928 m/s

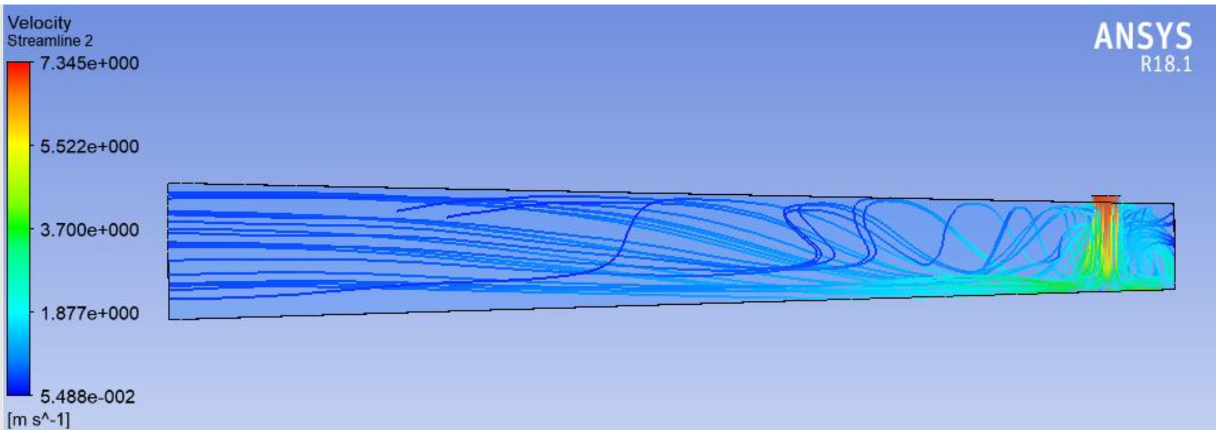

Figure 12. Streamline of the conical square inlet diffuser with a secondary flow velocity of 6.928 m/s

By this simulation with faster secondary flows, the flow visualized on the experimental setup gets more visible and understandable, as the areas where the concentration of velocity is greater is coincident with the areas where the turbulence were stronger on the experimental tests. Here the different behavior of the flow inside each geometry gets more evident, as the square inlet one tends to break the lines of current and the circular one tends to let the flow go downstream more smoothly.

Aiming to analyze the influence of the area of the secondary inlet, simulations with half the original area were conducted, see Figs.13,14, with a diameter of 3.5 mm; the condition of the main flow velocity was 0.212 m/s and the inlet was 3.464 m/s.

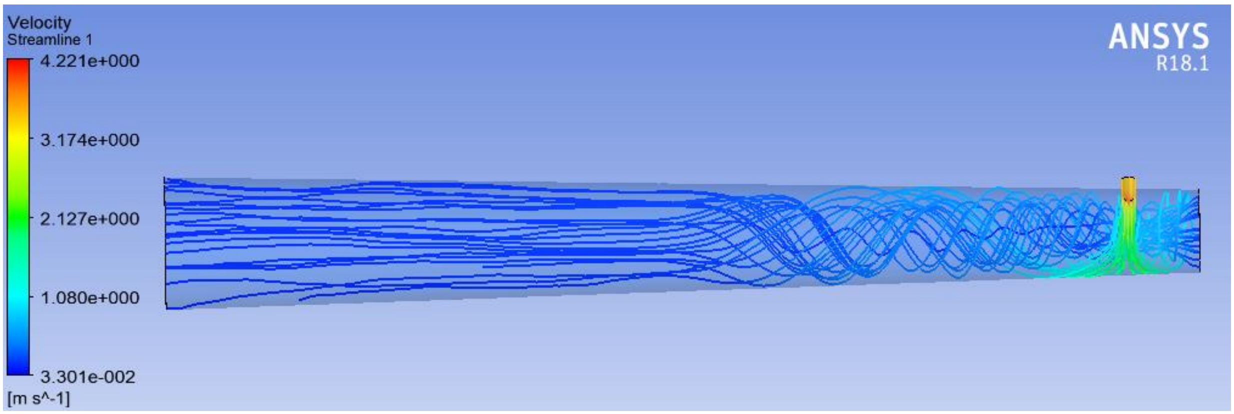

Figure 13. Streamline of the conical circular inlet diffuser with a reduced inlet diameter of 3.5 mm

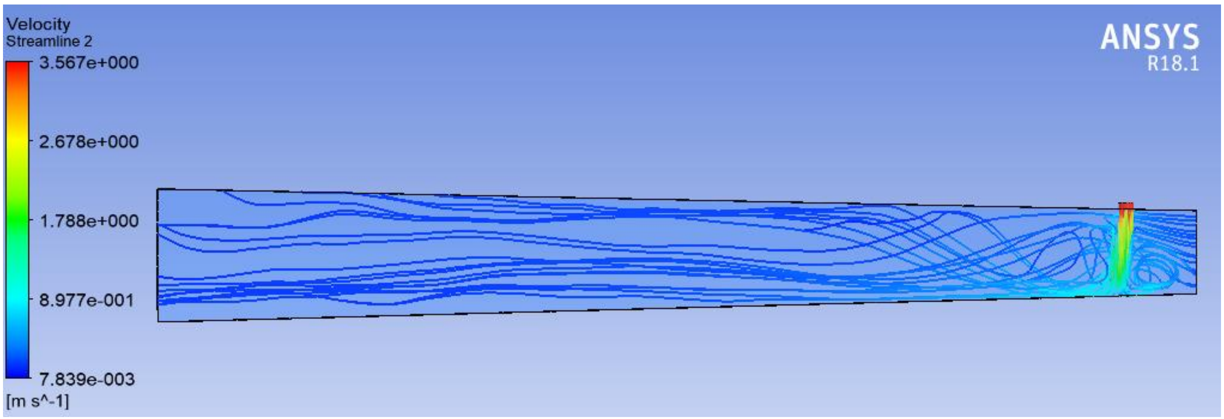

Figure 14. Streamline of the conical square inlet diffuser with a reduced inlet diameter of 3.5 mm

It is possible to observe that the reduction of the inlet area implies in more turbulence, increasing the amount of turbulent structures present in the flow.

5. CONCLUSION

The initial objective of this research was to visualize the formation of the turbulent structures inside a conical diffuser in a macroscopic scale, for these tests both experimental setups and numerical simulations were run, so these could serve of basis for comparison for each other to generate a better comprehension around the flows behavior. Visualizing the flow and its peculiarities caused by the conditions it is taken, led to a better understanding on how a flow behaves, and this can lead to better designs and more efficient equipments; it also opens space for more focused researches around the turbulent structures like swirl, vorticity and head loss dedicated works. The utilized velocities and Reynolds used in the workbench were satisfactory for the visualization of the turbulence, although numerical analysis with wider variables were executed with the intent of observing the influence of these and the conclusion obtained was that a diffuser with a pyramid trunk like geometry is a better converter of kinetic energy considering it slows the flow faster, being a better kinetic to potential energy converter, it was also concluded that the secondary flow velocity, which is responsible for causing the turbulence, is directly and proportionally responsibly for the amount of turbulent structures generated. The tests with different areas for the secondary flow led to the observation that as the thinner the flow is, the more it generates turbulent structures, once it kind of cuts the major flow.

6. ACKNOWLEDGMENTS

The first author would like to thank to Centro Universitário do Distrito Federal - UDF, for supporting him during this research.

7. REFERENCES

Angulo, T. M A., 2016. *Obtenção das Características de Desempenho de Turbinas Francis e Parametrização do Tubo de Sucção Utilizando Técnicas de Dinâmica dos Fluidos Computacional*. Msc. thesis, Universidade Federal de Itajubá, Minas Gerais, Brasil.

Brunetti, F., 2008. *Mecânica dos Fluidos*. Prentice Hall Brasil, São Paulo, Brasil, 2nd edition.

Cant, R. S., and Mastorakos, E., 2008. *An introduction to turbulence reacting flows*. Imperial College Press, Cambridge, United Kingdom.

Chen, B., Ho, K., Kin, F. G. F., Jiang, R., Abakr, Y. A., and Chan, A., 2015. "Validation and visualization of decaying vortex flow in an annulus", In *The 7th international conference of applied energy – ICAE2015*. Energy Procedia 75, Abu Dhabi, United Arab Emirates, pp.3098-3104.

Coelho, J. G., Junior, A. C. P. B., and Noleto, L., 2006. "Escoamento turbulento em difusores cônicos – simulações transientes". In *Anais da 5ª Escola de Primavera de Transição de Turbulência EPTT – 2006*. ABCM, Rio de Janeiro, Brasil, ett-06-18.

Chand, D., Tavoularis, S., 2012. "Numerical simulations of developing flow and vortex street in a rectangular Channel with cylindrical core". *Nuclear Engineering and Design* 243, 176-179.

Choueri, G., Tavoularis, S., 2014. "Experimental investigation of flow development and gap vortex street in an eccentric annular channel". *Part 1. Overview of the flow structure*. J. Fluid Mech, vol. 752, 521-542.

Johnson, P. A., and Patel, V. C., 1999. "Flow past a sphere up to a Reynolds number of 300". In *Journal of Fluid Mechanics*, 378(1). Cambridge, United Kingdom, pp.19-70.

Moller, S. V., and Silvestrini, J. H., 2004. "Turbulência: Fundamentos". In *Anais da 4ª Escola de Primavera de Transição de Turbulência EPTT – 2004.* ABCM, Porto Alegre, Brasil.

Neiva, R. Q., Sousa, A. J., Coelho, J. G., Junior, A. C. P. B., 2007. "Experimental and Numerical Study of the Swirling Flow in Conical Diffusers". In *19th International Congress of Mechanical Engineering – COBEM 2007*. ABCM, Brasília, Brasil.

Panofsky, H. A., and Dutton, J. A., 1984. *Atmospheric Turbulence: Models and Methods for Engineering Applications.* Wiley-interscience, New York, United States, Part.1.

Price, S. J., Sumner, D., Smith, J. G., Leong, K., and Paidousiss, 2002. "Flow visualization around a circular cylinder near to a plane wall". In *Journal of fluids and structures,* 16(2). Quebec, Canada, pp.175-191.

Guellouz, S., Tavoularis, S., 2000. *The structure of the turbulent flow in a rectangular channel contening a single rod – Part 1: Reynolds-Average measurements*. Exp. Thermal and Fluid Sci., 23, 59-73.

Schlichting, H., 1979. *Boundary-layer theory*. McGraw-Hill Book Company, New York, United States.

Tennekes, H., and Lumley, J. L., 1972. *A first course in turbulence*. MIT Press, Cambridge, United Kingdom, pp.149-164.

Vila, J. L., Dias, L. H., Mattos, P. H., Almeida, P. P., Melo, T., 2017. "Confecção de uma Bancada Experimental para Visualização de Escoamento Turbulento". In *24th Congresso de Estudantes de Engenharia Mecânica – CREEM 2017*. ABCM, Rio Grande, Brasil.

White, F. M., 2000. *Viscous fluid flow*. DCW Industries, Inc.,New York, United States, Chap.6.

8. RESPONSIBILITY NOTICE

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