NUMERICAL COMPARISON OF TWO-DIMENSIONAL AND THREE-DIMENSIONAL ANALYSIS OF A CROSSFLOW OVER TWO SIDE BY SIDE CYLINDERS

Roberta Fátima Neumeister, roberta.neumeister@ufrgs.br
Adriane Prisco Petry, adrianep@mecanica.ufrgs.br
Sérgio Viçosa Möller, svmoller@ufrgs.br
Universidade Federal do Rio Grande do Sul
Rua Sarmento Leite, 425 – Porto Alegre

Abstract. *The purpose of this paper is to present a comparative numerical analysis of transversal flow over two side by side cylinders with same sizes and space ratio of 1.26. On the study, a turbulent flow with Re = 2,2 x 10^4 is simulated using the Unsteady Reynolds Averaged Navier-Stokes equations with the turbulence model k ω – SAS (Scale Adaptative Simulation) and laminar flow with Re=2250. The transient turbulent analysis is applied with three-dimensional and two-dimensional domains to compare the impact of excluding the third spatial component in a flow interacting with two cylinders, due the small space ratio. The comparisons are executed with help of the streamlines on planes during the transient simulations and with temporal data series collected from the numerical analysis and experimental results from the equivalent aerodynamic channel. On the analysis, the different behaviors in each approach were observed, with periodic wake modes change in the two-dimensional analysis, while in the three-dimensional results the change modes are random.*

1. INTRODUCTION

Flow over circular cylinders are observed on various engineering areas, in some of then, as heat exchangers, bridges supports and nuclear reactors, the flow behaviors are physically and geometrically equivalents to the applications and the flow effects are directly related. In other areas, the use of flow over two cylinders can be approximated as two circular cylinders side by side to facilitate the analysis. In many of these engineering applications, the numerical solution method is even more simplified to reduce the time of simulation and these simplifications are made when the influences generated by the additional considerations are small, comparing with the spent time on the solution. In other situations, the use of simplifications, as in this study, are to understand some mechanisms that are presented on the flow and evaluate the losses on the results with the third spatial component disregarded.

The flow over two side by side cylinders with small pitch show asymmetric configuration on the wakes, generating difference on the mean velocity on the wakes, and also impacting on the drag and lift forces on the cylinders. The asymmetric configuration changes over time, in a random behavior, and this change of mode is called bistability, because the wakes remain in two stable levels (narrow and large wake). These is a phenomenon that continue unknown in many ways and has motivated studies of flows over two side by side cylinders to obtain some answers.

The study presented by Alam et al. (2003), was an experimental analysis of aerodynamic characteristics of a flow over two side by side cylinder, showing the main singularity with one large wake and one narrow wake. The authors also showed the distinct behavior on the drag and lift coefficients on the large and narrow wake configurations analyzing distinct *P/D*, with *P* = pitch and *D* = diameter.

Afgan et al. (2011) executed a numerical study using LES (Large Eddy Simulation) on a turbulent flow with Re=3000 over two side by side, with various *P/D* and the authors found similarity with the modes behavior, lift and drag coefficients to the ones described by Alam et al. (2003).

Sarvghad-Moghaddam et al. (2011) performed a laminar and turbulent flow analysis on two-dimensional domain with two cylinders with spacing ratio, *P/D* = 1.5 to 4. The authors observed the asymmetric wake region for laminar flows with *Re* = 100 and 200 and turbulent flow with *Re* = 10^4. They concluded that there was variation of the position of wakes in time. The authors made an analysis of drag and forces and observed a flip flopping behavior on the turbulent analysis.

de Paula and Möller (2013), studied the flow over two side by side cylinders using hot anemometry technique and flow visualization in a water channel. The results showed similarity with the literature and the authors observed on the water visualization that the bistability is preferentially two-dimensional, even with a subcritical Reynolds number and turbulent flow.

The turbulent study by definition is three-dimensional and many studies, as the one executed by Akwa (2014), with flow over one cylinder, presented the imprecision on the results collected from a two-dimensional analysis comparing with experimental and three-dimensional numerical results, where the author studied the drop pressure on the channel wall and the results showed similarity on the tendency, but magnitudes differences.

The use of two-dimensional analysis in this study has the objective to compare the behavior of the vortices interaction with a three-dimensional analysis. Furthermore, compare the velocity behavior on a two-dimensional turbulent simulation with a two-dimensional laminar, a three-dimensional turbulent simulation and an experimental result. This comparisons are proposed considering the observation done by de Paula and Möller (2013), where the visualizations characteristics pointed to the predominantly two-dimensional behavior.

2. METHODOLOGY

The numerical analysis solves Navier-Stokes equations using the approach Unsteady Reynolds Averaged Navier Stokes, URANS, described on Wilcox (1994), with the commercial software ANSYS FLUENT 13 applying the interpolation scheme power-law and the coupling pressure-velocity SIMPLEC, described on Ansys (2010). To solve the Reynolds Tensor, the model kω – SAS, described on Menter and Egorov (2010), was used with time-step 0.001s, the continuity and momentum equations with the Reynolds Tensor are presented in Equation 1 and Equation 2. In the first analysis, Case 1, the equations are used in the full form, with i and j ranging from 1 to 3. On the Case 2, with the simplification of the third spatial component i and j range from 1 to 2, becoming a two-dimensional analysis and the Case 3 is two-dimensional laminar, without the Reynolds Tensor on the equation.

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \overline{u}_i}{\partial t} + \overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \nu \frac{\partial^2 \overline{u}_i}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j}\overline{u_i' u_j'} \tag{2}$$

with $\overline{u}$ representing the mean velocity in m/s, $\overline{p}$ the mean pressure in Pa and $\overline{u_i' u_j'}$ the Reynolds Tensor obtained from the Reynolds approximation.

The domain top view is presented on Figure 1, with the dimensions used on all the simulations, two and three-dimensional, the additional information for the three-dimensional domain is the high of 146 mm on the direction *z*. The boundary conditions information was obtained experimentally in the aerodynamic channel and the positions of inlet, outlet and walls are show on Figure 2.

For the Case 1, turbulent flow with three-dimensional domain, and Case 2, with turbulent two-dimensional analysis the inlet condition is prescribed as mean velocity of 13.45 m/s, generating $Re = 2.2 \times 10^4$, with uniform velocity profile, turbulent intensity of 0.6% and length scale of 0.001 m. At the outlet condition the prescribed pressure is applied with atmospheric pressure, turbulent intensity of 0.83% and scale length 0.01 m. On the walls the condition of no slip was used in both cases. The initial condition was equivalent to the inlet characteristics.

For the Case 3, laminar flow with two-dimensional domain, the inlet condition was the mean velocity of 1.345 m/s, generating $Re = 2250$. On the outlet the pressure of 1 atm was implied and the walls were considered no slip.

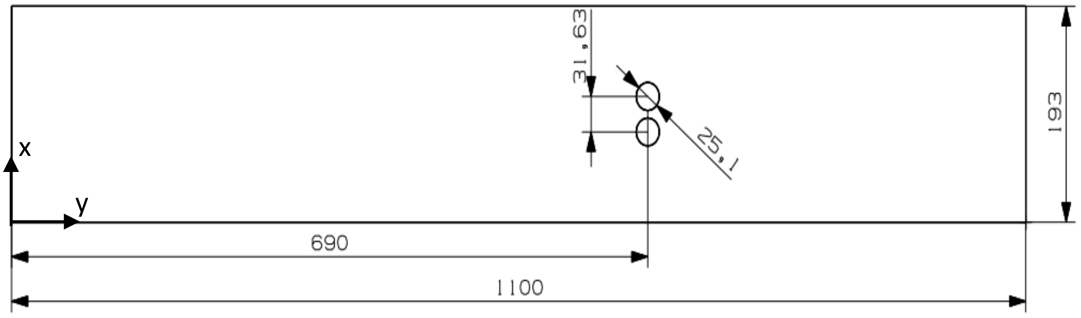

Figure 1 – Computational domain with dimensions in mm.

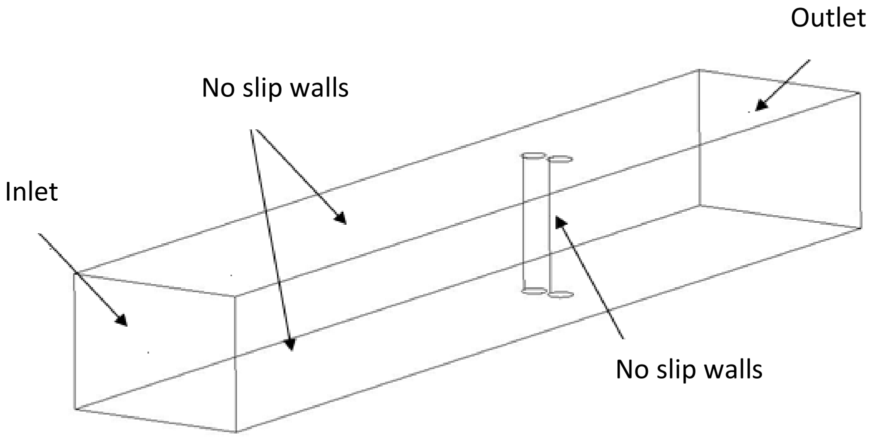

Figure 2 – Boundary conditions applied on the numerical analysis.

The two-dimensional mesh, showed on Figure 3-a), was constructed with hexagonal elements and layers with the growing direction from the cylinders, to detail the flow around the bodies. The three-dimensional mesh, showed on Figure 3-b), was generated using tetrahedral volumes except for the wall regions, where layers of prismatic volumes were inserted in the region around the cylinders. On the wake area, a density is applied with growth ratio of 1.6.

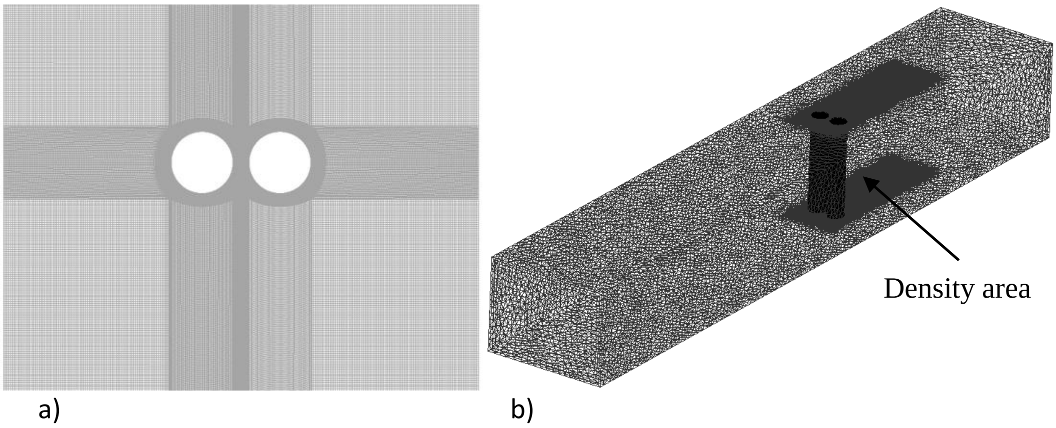

Figure 3 – Mesh details a) Two-dimensional domain and b) Three-dimensional domain.

The mesh quality evaluation was executed using the GCI method (Grid Convergence Index) proposed by *Roache (1994),* and detailed on *NASA (2015*). The GCI indicates an error band on how far the solution is from the asymptotic value. It indicates how much the solution would change with a further refinement of the grid. A small value of GCI indicates that the computation is within the asymptotic range. The GCI analysis to the three-dimensional mesh is shown in Table 1, where can be seen the refinement ratios of 1.39 and 1.44. The speed values on the position $y = 0.51\ m$ were monitored in each mesh, ranging between 20.61 and 19.38 m/s. The convergence of the iterative solution is in the order of 2.79 and generating an error band of 1.6%. Taking into account the increased computing time required for the solution with the refined mesh, the option is the use of the median mesh.

For the two-dimensional analysis the same evaluation, with GCI method, is executed and presented on Table 2, the ratios between the meshes are 1,35 and 1,19, the results of velocity in the monitored position are from 12,43 to 10,49 m/s, the final error band is of 6,4% and that leads to the use of the refined mesh on the simulations executed on the present analysis.

Table 1– Mesh Quality using GCI method for three-dimensional domain.

Mesh 1 - Refined	9.955.670 volumes
Mesh 2 - Medium	3.714.222 volumes
Mesh 3 - Coarse	1.237.784 volumes
r_{12}	1,39
r_{23}	1,44
f_1	19,38 m/s
f_2	19,69 m/s
f_3	20,61 m/s
$f_{exato,\ 12}$	19,17 m/s

$f_{exato,\ 23}$	19,17 m/s
ε_{12}	1,6 E-02
ε_{23}	4,7 E-02
h	2,79
GCI $_{12}$	0,0133
GCI $_{23}$	0,0328
a	0,9843
Band of error	1,59%

Table 2– Mesh Quality using GCI method for two-dimensional domain.

Mesh 1 - Refined	547.496 volumes
Mesh 2 - Medium	299.776 volumes
Mesh 3 - Coarse	209.783 volumes
r_{12}	1,35
r_{23}	1,19
f_1	10,79 m/s
f_2	11,48 m/s
f_3	12, 43 m/s
$f_{exato,\ 12}$	10,44 m/s

$f_{exato,\ 23}$	10,44 m/s
ε_{12}	0,0639
ε_{23}	0,0828
h	3,64
GCI $_{12}$	0,0399
GCI $_{23}$	0,1127
a	0,9399
Band of error	6,39%

The numerical data is compared with experimental results obtained with hot anemometry technique in the LMF's aerodynamic channel, described by de Paula and Möller (2013). The test section dimensions are the same as presented on Figure 1, but with total length of 900 mm instead of 1100 mm, this additional length was necessary to reduce the outlet condition impact on the numerical simulation. The boundary conditions used on the turbulent numerical simulations were also collected in the aerodynamic channel. The experimental data was acquired with acquisition frequency of 1000Hz and low pass filter of 300Hz.

2.1. Results and discussion

For the analysis in the turbulent three-dimensional flow, Case 1, a transient simulation with the *Re* = $2{,}2 \times 10^4$ and 0.001s timestep was executed during 8,2s. In this total time, one change of wake modes was observed. The wake modes change can be observed on the streamlines presented on Figure 4, the planes were positioned on 0.073 m from the base. The change of modes happened between the instants 6.362 s and 6.462 s and in all the frames the asymmetry on the wakes can be observed. The wake change in the period does not present an indicative of changing and has a random behavior on the velocity sign presented on the Figure 6.

Based on the observations from the flow visualizations by de Paula e Möller (2013), that the behavior of bistability is preferentially two-dimensional, even in subcritical Reynolds, a two-dimensional simulation was executed, keeping the same boundary conditions as the three-dimensional analysis (Case 1) and eliminating the third spatial component. This evaluation was executed in order to understand the behavior of the wakes and the losses caused by this simplification, since there is no vertical component to the interaction and dissipation of vortices.

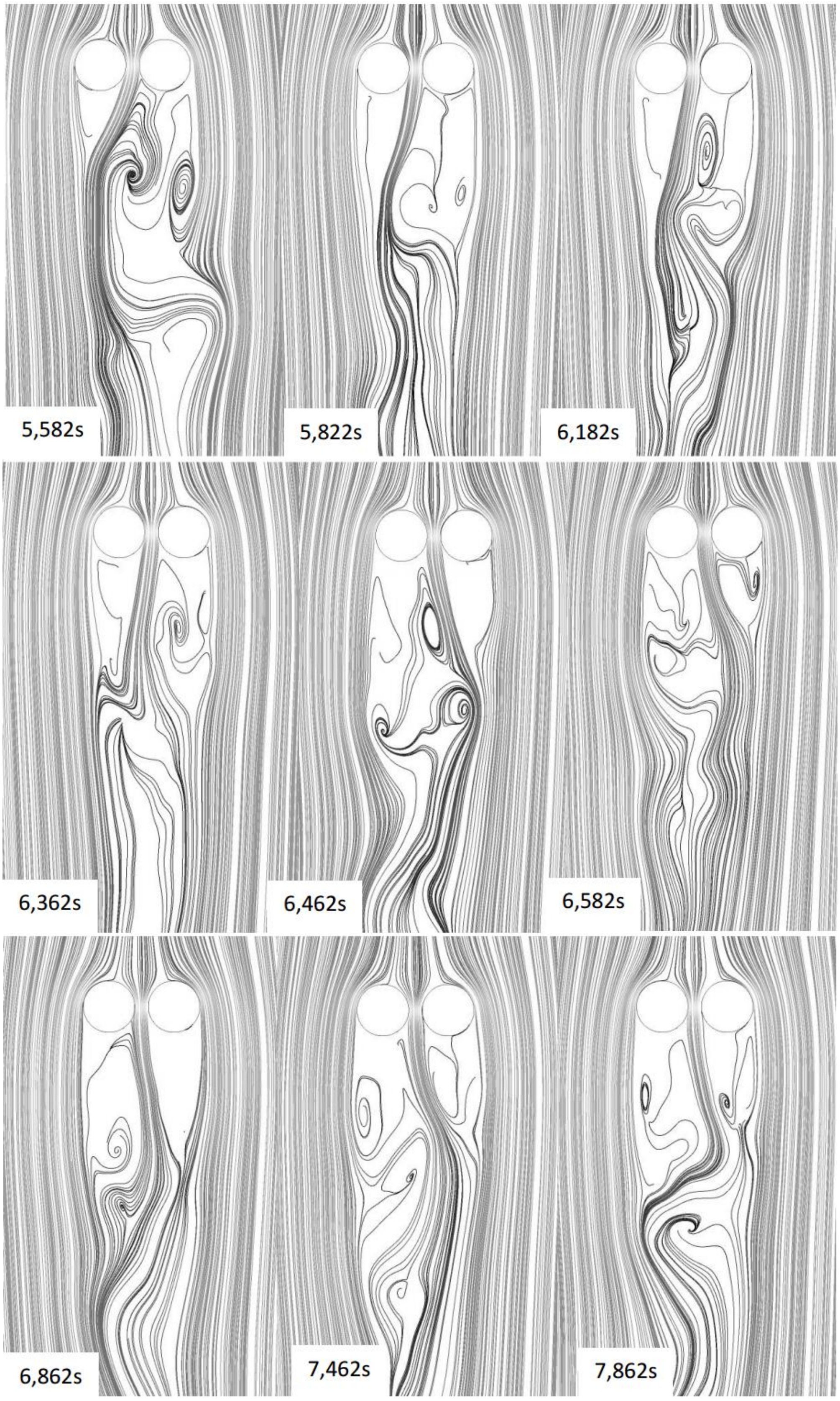

Figure 4 – Streamlines on planes positioned 0,073 m from the base with the wake change on the three-dimensional simulations (Case 1).

The two-dimensional simulation, Case 2, was executed during 5,5s and the comparison planes with streamlines are shown in Figure 5, the data series was also obtained during this period and is presented on Figure 7. On Figure 5, two wakes with different sizes are noticed, one larger than the other one, as in Figure 4, but a small size of the wake downstream are identified comparing both results. Without the possibility of change in the z component, the vortices have higher definition. Another observation on Figure 5 is the relation between vortices on the times 3,808s and 3,823s, when structures with the same signal join and make the size of wake change the position. The vortices with same signals approximate in a periodic pattern and the wake change happens, in the same pattern.

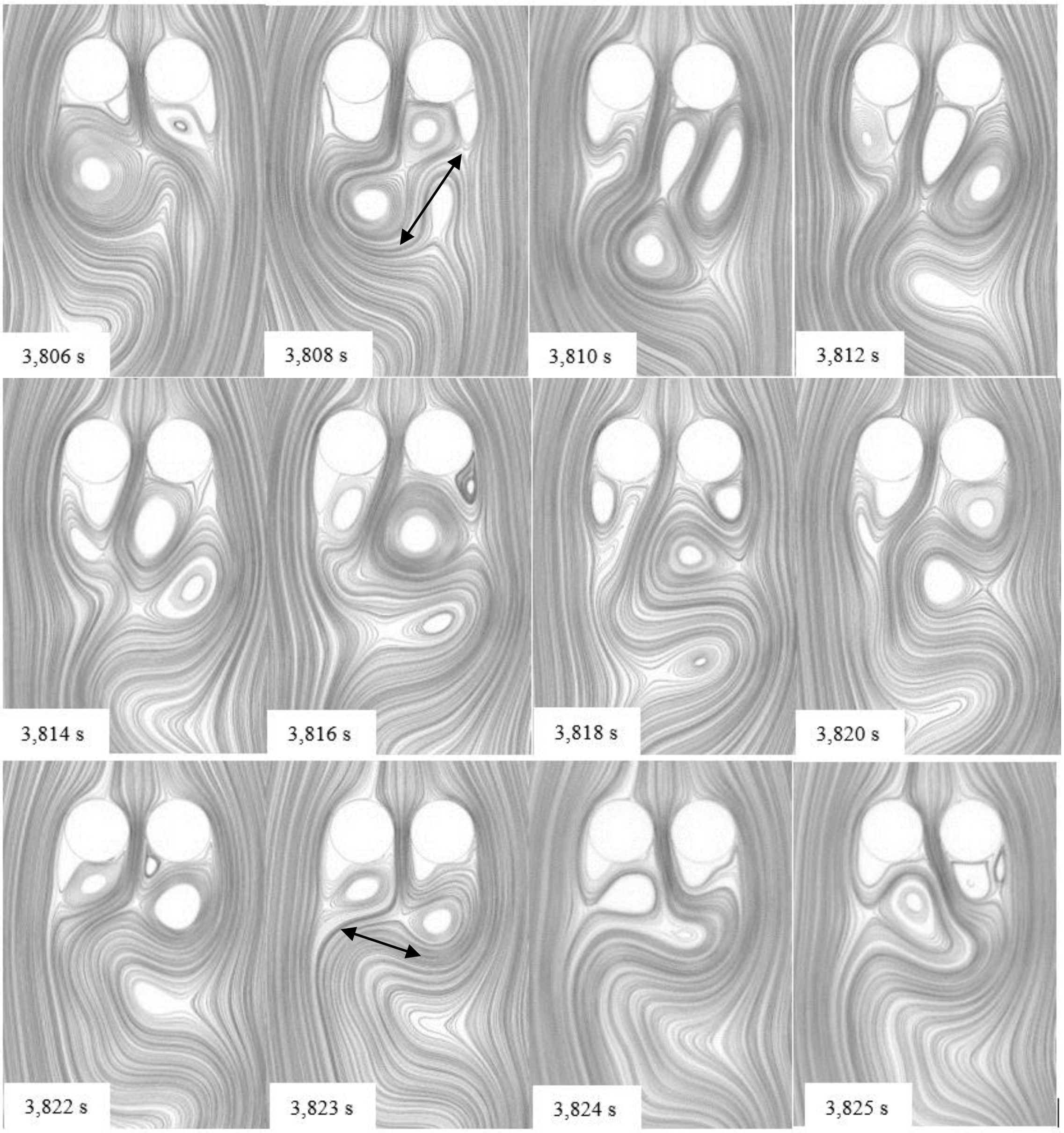

Figure 5 – Streamlines from the turbulent two-dimensional simulations (Case 2).

Comparing the results from Figure 4 and Figure 5, some characteristics show wide variation as the vorticities definition, on Case 1 the vortices structure is visible in some instants, as 5,582s and 7,462s, while in the other times the vorticities are dispersed, or in a different plane, due to the migration of the structures on the *z* component. On the Case 2 the velocity on the wake is higher, visible on streamline proximity, and the vortices are well defined in all times, even creating a periodic change on the wake modes due to the interaction of the same signal vorticities, and all these differences can be linked to the two-dimensional consideration.

The data series obtained behind the right cylinders, on the position x = 0.12 m and y = 0.71 m, during the simulations of Case 1, Case 2 and Case 3 are presented on Figures 6, 7 and 8, respectively, showing 1s of the results for each case. On the Figure 9 the experimental data obtained on the aerodynamic channel, in the same position with uncertainty of ±0,05 mm, is presented for comparison. Observing the results for each situation it is possible to identify the behaviors differences between then, on Figure 6 a random behavior on the velocities with regions of higher and lower values is observed, with mean value tending to 3m/s, showing similarity with the experimental results.

On the experimental case presented on Figure 9, where the values have an associated uncertainty of ± 5%, the sign pics are more visible because it is the physical representation while on the Figure 6 the pics values are attenuated due the modelling used in the solution with URANS. On the results from the turbulent two-dimensional analysis, Figure 7, a quasi-periodic behavior is observed, presenting a repetition on the sign in approximately 0,015 s, this timing is equivalent with the change modes observed on Figure 5, when the union of vortices happens. Another divergence between the results is the time series velocity is the mean value around 10 m/s, that represents more than three times the experimental results value.

The same periodic characteristic observed on the turbulent two dimensional analyses, but with higher repeatability is observed on the results from the two-dimensional laminar solution. On Case 3, where two pics, one with velocity up to 2.5 m/s and other with velocity up to 1.5 m/s, are connected with the narrow and wide wake, respectively.

On the literature, a periodic relation was observed on the drag and lift forces presented on the two-dimensional analysis for *P/D* = 2 and with *Re* = 200, executed by Sarvghad-Moghaddam et al. (2011), but not with the period so defined as the one observed in Figure 8. On the present case, the velocity behavior has a cycle that repeats in each 0,1 s, this higher organization can be related to the small pitch used in the study, because with the bodies proximity the interaction between wakes increase.

In general, laminar simulations can be executed on two-dimensional domains due the organized flow characteristic, but in the case of flow over cylinders with small pitch the periodic wake change does not represent the experimental response. Observing experimental behaviors from the study made by Sumner (2010), where the author analyzes flows with low Reynolds, the wakes do not flip-flop and do not have a periodic characteristic, but a random behavior as the one observed in turbulent flows.

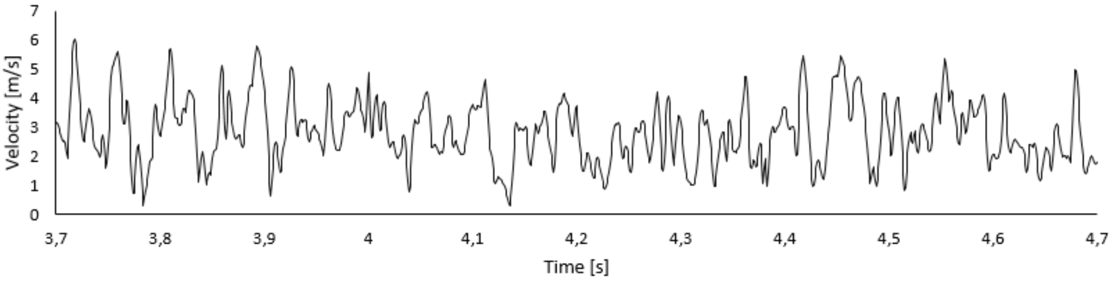

Figure 6- Numerical data series behind the right cylinder on the three-dimensional analysis with *Re* = 2,2 x 10^4.

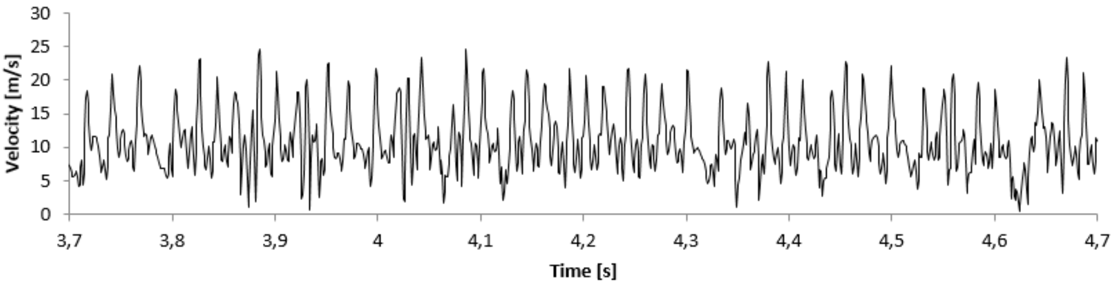

Figure 7 - Numerical data series behind the right cylinder on the turbulent two-dimensional analysis *Re* = 2,2 x 10^4.

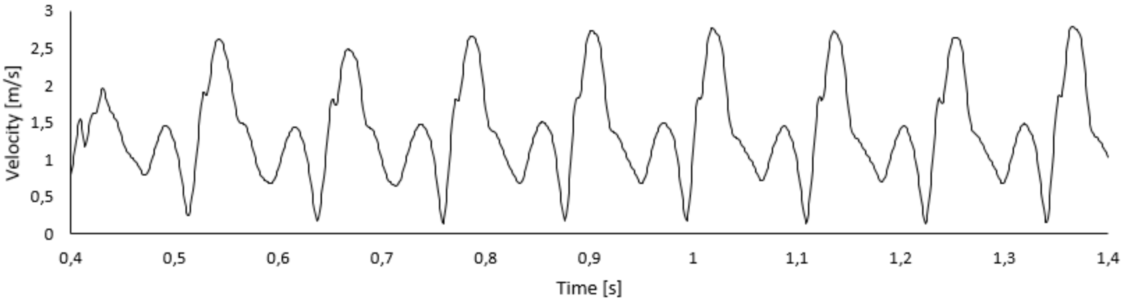

Figure 8- Numerical data series behind the right cylinder on the two-dimensional analysis with Re = 2250.

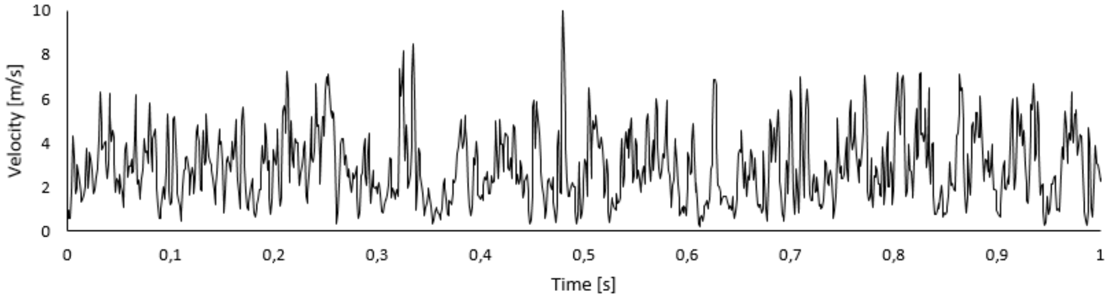

Figure 9- Experimental data series behind the right cylinder from a flow with *Re* = 2,2 x 10^4.

3. CONCLUSIONS

This study presents an analysis of the differences observed between two and three-dimensional analysis of transversal flow over two side by side cylinders. Three cases are studied, the Case 1 was a turbulent flow with three-dimensional domain, the Case 2 a turbulent flow with two-dimensional domain and Case 3 was a two-dimensional laminar flow. The results obtained from the three cases were compared between each other and with experimental results obtained from an aerodynamical channel. The comparisons were executed using streamlines and time data series.

The results showed that even with visualizations from the literature indicating the predominantly two-dimensional behavior, the third spatial component is the main cause of the random bistable phenomenon. From the turbulent two-dimensional approach, a quasi-periodic behavior on the velocity values is observed caused by the direct interaction between wake vortices of same signals, that create the wake mode changes. The laminar evaluation presented a periodic behavior on the velocity signs, caused also by the vortices interaction, due the small space ratio, *P/D* = 1,26, that creates a large interaction in the wakes regions. In both two-dimensional cases, the results do not represent the experimental behavior, even in the case with laminar consideration the small pitch creates a relation between the vortices that cannot be analyzed without the third component.

In conclusion the present study shows that the use of two-dimensional numerical evaluations does not represent the bistable phenomenon, showing quasi-periodic or periodic responses that are not representative of the physical phenomenon. In the other hand, the analysis showed clearly the interaction between vortices on the wake and the mode changes happening because of this union. This can be the characteristic of the three-dimensional analysis that generates the mode changes, but it happens randomly because the third spatial component allow the vortices dispersion and prevents the vortices alignment on the wake.

4. ACKNOWLEDGEMENTS

Authors are gratefully indebted to the CNPq National Council for Scientific and Technological Development, for the financial support.

Roberta Neumeister thanks also the CNPq for granting her a fellowship.

Thanks are also due to the CESUP (National Center for Supercomputation) for the computational resources provided.

5. REFERENCES

Afgan, I.; Kahil, Y.; Benhamadouche, S.; Sagaut, P., 2011.” Large eddy simulation of the flow around single and two side-by-side cylinders at subcritical Reynolds number”. *Physics of Fluids*, Vol 23, p. 075101.

Akwa, João Vicente. 2014. “Estudo numérico e experimental do escoamento sobre um rotor eólico Savonius em canal aerodinâmico com alta razão de bloqueio.” – Doctor Thesys – Federal University of Rio Grande do Sul.

Alam, M. M.; Moriya, M.; sakamoto, H., 2003. “ Aerodynamic characteristics of two side-by-side circular cylinders and application of wavelet analysis on the switching phenomenon”. Journal of Fluids and Structures, Vol.. 18, p. 325–346.

Ansys Inc., 2010 Ansys Fluent 13.0 – Theory Guide.

de Paula, A. V.; Möller, S. V. 2013, “Finite mixture model applied in the analysis of a turbulent bistable flow on two parallel circular cylinders”. Nuclear Engineering and Design, Vol. 264, p. 203–213. 2013.

Menter, F. R., and Y. Egorov. 2010. “The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 1: Theory and Model Description.”. Turbulence and Combustion, Vol. 85, p. 139 – 165.

NASA, 2015. “Examining Spatial (Grid) Convergence.” May 27. http://www.grc.nasa.gov/WWW/wind/valid/tutorial/spatconv.html.

Roache, P. J., 1994. “Perspective: A Method for Uniform Reporting of Grid Refinement Studies”. Journal of Fluids Engineering, Vol. 116, p. 405-413.

Sarvghad-Moghaddam, Hesam, Navid Nooredin, and Behzad Ghadiri-Dehkordi. 2011. “Numerical Simulation of Flow over Two Side-by-Side Circular Cylinders.” Journal of Hydrodynamics, Vol. 23, p. 792–805.

Sumner, D. 2010. “Two circular cylinders in cross-flow: A review.” Journal of Fluids ans Structures, Vol. 26, p. 849 – 899.

Wilcox, D. C., 1994. Turbulence Modeling for CFD. DCW Industries, Lã Cañada, California

6. RESPONSIBILITY NOTICE

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