COBEM2015

23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

VALIDATION OF EXPERIMENTAL AND NUMERICAL DATA IN A FLOW AROUND TWO CYLINDERS SIDE-BY-SIDE

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

Abstract. *The purpose of this paper is to present the comparison between the experimental and numerical studies for a turbulent flow around two smooth cylinders side by side with equal diameters and P/D (pitch/ diameter)=1.26. The experimental data was obtained using the hot wire anemometry technique in the aerodynamic channel, with the cylinders inside a flow with constant velocity. The collected data was statistically treated applying de power density spectrum for the analysis of the Strouhal Number in each cylinder. A boundary conditions study using the experimental velocity profile and the turbulent intensity were imposed to the equations solution, with polynomials that describe the profiles behavior inside the channel to verify the impact on the numerical simulations. The results showed better proximity using the homogeneous software profile than the polynomial approximation. For the numerical simulations, the mean values from the aerodynamic channel were applied as boundary conditions. The flow around cylinders numerical simulations were executed using Reynolds Average Navier-Stokes (RANS) with* $\kappa\omega$ *- SAS formulation and compared with experimental visualizations and LES results from the literature. The comparison shows coherent results in all the cases and the predominance of large scales in the phenomenon around the cylinders.*

Keywords: *Experimental, Numerical, Cylinder side by side.*

1. INTRODUCTION

Applying numeric simulations in the evaluation and solution of fluid mechanics problems became usual in the past years. The high performance computers number increased and allowed faster answers in this field. In the other hand, even with the big access to numerical solution models, it is necessary to validate the data obtained from the numeric simulations and that is why many of numeric papers use experimental publications to evaluate the results and be sure that they are trustable.

The boundary conditions application is one of the greatest doubts in comparisons between numerical and experimental data, because in the simulations this information is clear and well defined with the software. In the experimental publications the data is statistic, from time series. To understand the results behavior when applying the experimental conditions, different velocity profiles are created from the experimental velocity data and input on the software Fluent 13.0.

The turbulent flow characteristics around two side-by-side cylinders have been studied from the last decades, mostly stable behavior of the asymmetric structure is intriguing, because of the non-reason change in the wakes positions. The analysis made by Kim and Durbin (1988) showed some details that created the interest of many researchers over the years.

One interesting experimental study presented by Alam et al. (2003) shows the bi-stable characteristics in the lifting and drag forces in many P/D relations, showing the presence of different force magnitudes in cylinder with narrow and wide wakes. The presence of an undefinable mode, when the wakes get the same size was discussed as well. Besides that, they made visualizations with hydrogen bubbles to present the structure of the flow. Another experimental analysis of these phenomenon is presented by de Paula and Möller (2013) showing the bi-stable behavior and his characteristics using a probability density function in which the frequencies of axial and transversal velocity are compared and show the shape of distribution implementing the bi-stable data acquisition and presenting dye flow visualizations in water channel.

In the numerical field Afgan et al. (2011) shows the study of bi-stable flow with LES simulation, presenting the results for drag and lift forces behavior in each cylinder for the angle position, even comparing with Alam et al. (2003) results. The authors showed the Strouhal number for each P/D studied and the velocity/pressure fields in some frames that represent the bi-stable flow. The authors present the wake position change on time printing the behavior of the wakes in a time interval.

2. EXPERIMENTAL AND NUMERICAL METHODS

The experimental data used in the present study was obtained from the aerodynamic channel located in LMF- Fluid Mechanics Laboratory of UFRGS described in details on de Paula and Möller (2013), with the test section made of acrylic plates. The main velocity in the channel is 13.45 m/s with error of ±3 %. In the data collection to the free channel a section with 193 x 146 x 1020 mm was used and for the measures with tubes the test section had 193 x 146 x 900 mm with the cylinders positioned 690 mm from the inlet. The data acquisition of velocities over the section was made using the hot wire anemometry technique with the Streamline Equipment and for the pressure is used manometer Schöpe & Faeser connected to a digital multimeter Politerm (POL-79C) with the software Multimeters - RS232 for data record from the multimeter.

The study used two numerical approaches for the analysis. The first one was three-dimensional steady flow in the free channel using RANS, with the model $\kappa\omega$ - Standard that presents one equation for the kinetic turbulent energy and one for the viscous dissipation rate, all the model constants were the default ones, the equations are described by Wilcox (1994). The domain has the same dimensions as the experimental channel, 193 x 146 x 1020 mm, and the interpolation method was second order upwind for all the equations. The second analysis was three-dimensional transient using URANS (Unsteady Reynolds Average Navier-Stokes) and applying the $\kappa\omega$-SAS (Scale Adaptive Simulation) that was developed by Menter and Egorov (2005) to solve the turbulent spectrum in unstable flow conditions, as cylinders wakes. The method demands a second order interpolation method and for the case with large time step was applied second order upwind and for the case with small time step the power law method. The domain in this case is 200 mm longer than the experimental, to see the wake behavior behind the cylinders, it dimensions are presented in Fig.1 (a) in a top view showing the cylinders with P/D=1.26. The height of the channel is 146 mm.

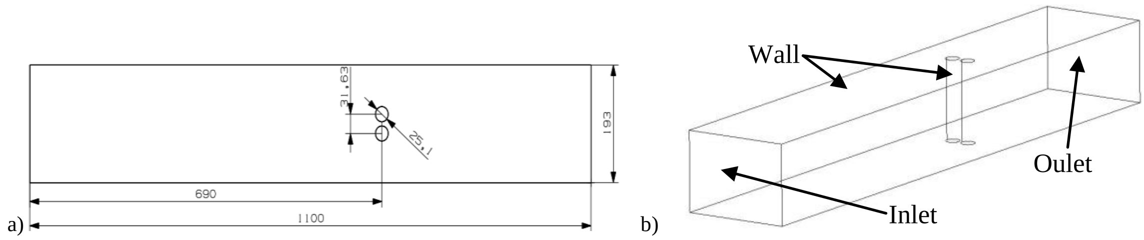


Figure 1. Computational Domain a) Top view with dimensions in mm and b) Three-dimensional geometry with boundary conditions.

The applications of the boundary conditions are presented in Fig. 1 (b). For the inlet, in the case with free channel, some polynomials profiles created from the experimental data were input with a User Defined Function (UDF), which is an external routine that input the equation for the velocity profile in the software Fluent (Fluent, 2010). The length scale of 0.001 m and intensity turbulence of 0.6 % were applied on the inlet. On the channel sides the wall condition with no slip was used. The outlet is considered as atmospheric pressure, 0 Pa, with length scale of 0.01 m and turbulence intensity 0.83 %.

For the case with cylinders was applied, the homogeneous velocity profile with 13.45m/s mean value, length scale of 0.001 m and intensity turbulence of 0.6 %. In the side plates and cylinders the wall condition with no slip was used. The outlet surface is considered with atmospheric pressure, 0 Pa, length scale of 0.01 m and turbulence intensity 0.83 %. The conditions are based on the experimental test section.

3. ANALYSIS OF DIFFERENT BOUNDARY CONDITIONS

The experimental data was obtained with the free section and with the unidirectional probe positioned 285 mm from the homogenizing grid that presents a spacing of 0.001 mm. The objective in this analysis is evaluate the difference presented using the homogeneous profile generated with the software Fluent and with a polynomial that represent the experimental profile. After the data treatment using Matlab 2012, the velocity profiles are plotted from the experimental data and from the velocity information the turbulent kinetic energy and viscous dissipation rate are calculated, the three profiles are presented in Fig. 2. The outlet mean turbulent intensity and the length characteristic are collected at 1019 mm from the inlet and results a turbulent intensity of 0.83 % and length scale of 0.01 m. The channel walls are smooth.

The wall drop pressure over the channel is used to compare the numerical and experimental data. In the right lateral plate was made nine pressure taps with spacing of 100 mm between then, in each position 1024 acquisitions are recorded and after the statistical treat resulting in the drop pressure presented in Fig. 3. The experimental error in the pressure collect is 1.913 %.

The experimental profiles were approximated by polynomials of third, fourth a sixth degrees, each one was input in the software separately. For the study initially just the velocity profiles are used, keeping the mean intensity turbulent and length scale. The curves from the numerical simulations are created with the results of pressure in the same points as the experimental data and the curves are plotted in Fig. 3. The position error in the experimental data is ±0.402 mm.

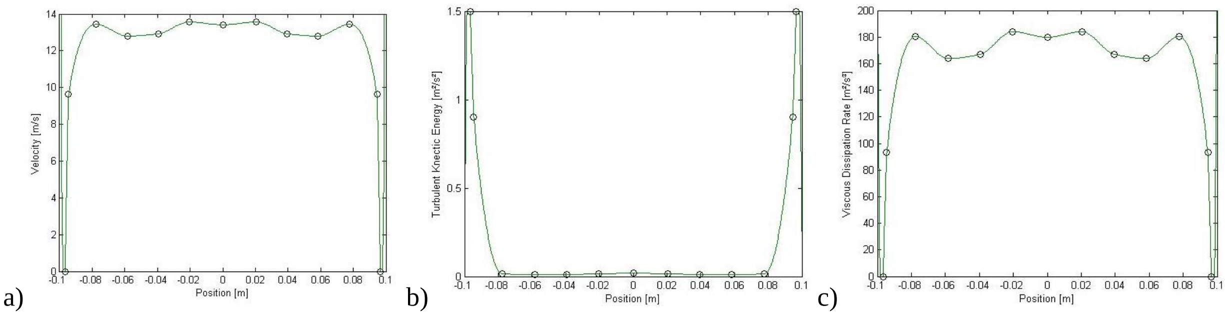


Figure 2. Profiles from the experimental data. a) Velocity, b) Turbulent kinetic energy and c) Viscous dissipation rate.

To assure the physics coherence of the results is executed the mesh quality analysis applying the Grid Convergence Index (GCI). Roache (1994) presented this method and it is based on the Richardson extrapolation and indicates one asymptotic approach to quantify the mesh uncertainty. The method demands the use of three mesh sizes with refinement ratios above 1.3, the interactive method varies the P value until the value obtained for GCI_{23} is equal or with a small difference in relation to r^P*GCI_{12}. This method allows the prediction of a result with the mesh with zero spacing (analytical) and allows calculating one error value for the numerical simulation.

For this case the mean velocity in the position y = 0.51 mm was used as the comparable variable in the steady simulations. The ratio between the values of GCI_{23} and r^P*GCI_{12} is 1.0018 and means that the results are very close even with a large difference between the mesh size. From this relation it is possible calculate the difference for the asymptotic convergence that is 0.1845 %. Another indicative of good quality can be verified in the estimated values f_{12} and f_{23}, that are very close in both calculations even with different initial values. GCI details for the case can be seen in Table 1 and the formulation used is in Roache (1994).

Table 1. Mesh quality in free channel.

GCI - Free Channel		
Mesh 1 - R	4817214	elements
Mesh 2 - M	826200	elements
Mesh 3 - C	54864	elements
Refinment ratio between mesh 1 and 2	1,7999	
Refinment ratio between mesh 2 and 3	2,4694	
Mean velocity on y=0.51 - Mesh 1	10,8400	m/s
Mean velocity on y=0.51 - Mesh 2	10,8200	m/s
Mean velocity on y=0.51 - Mesh 3	10,7200	m/s
P - Estimated	1,5209	-
P - Calculated	1,5209	-
GCI_{12}	0,0016	-
GCI_{23}	0,0039	-
Estimated value - f_{12}	10,8538	-
Estimated value - f_{23}	10,8538	-
r^P*GCI_{12}	0,0039	-
Ratio between GCI_{23} and r^P*GCI_{12}	1,0018	-
Value of asymptotic convergence	0,1845	%

Figure 3 presents the results from each numeric simulation and the experimental results. Comparing the curves of third and fourth degrees it is easy to see the difference between the numerical results and the experimental data, with 40 % of difference in the initial position and getting closer in the lasts points. This behavior gives the impression that if the channel gets longer the results will follow the experimental curve because it is adapting in the domain, but for the case these profiles do not represent the experimental behavior. For the case with sixth degree it is possible to observe more proximity than the other profiles with the experimental drop pressure, but keep with a large difference in relation to the experimental curve. The simulation with homogeneous profile showed closer results with the experimental points, without look in the first position all the experimental points are close to the numerical curve.

Based on the results it is possible affirm that the homogeneous profile made by Fluent is enough to represent the experimental profile and it is not necessary the creation of UDFs. The mean experimental values can be input directly in the software. Having these results for the velocity profiles it was not made the same analysis for the turbulent kinetic energy and viscous dissipation rate, because the mean values already proved to be very good in the application.

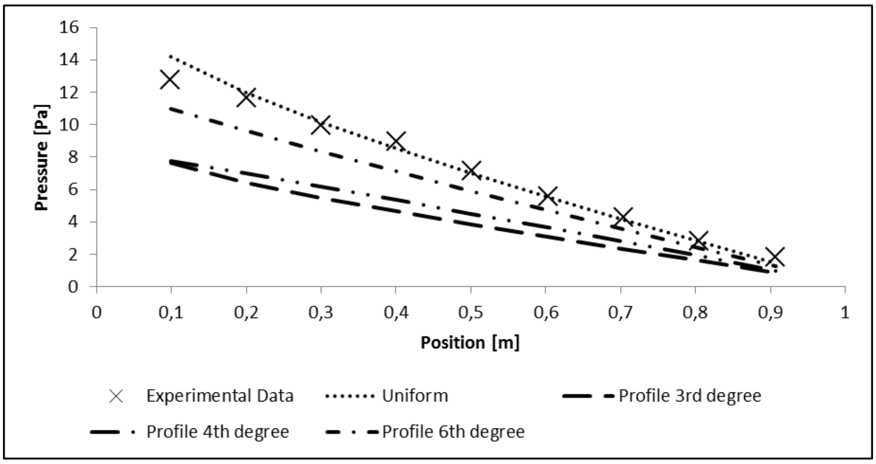


Figure 3. Pressure drop over the free aerodynamic channel wall.

4. FLOW AROUND CYLINDERS

For the initial numerical studies of flow around two side-by-side smooth cylinders were applied a tetrahedral mesh with 1,600,000 volumes, transient analysis, time step of 0.5 s and in a URANS solution using $\kappa\omega$-SAS model for the solution as described on the Section 2.

The boundary conditions applied were 13 m/s velocity-inlet with turbulent intensity of 0.6 % and length scale 0.001 m. In the outlet surface the condition of outflow was imposed and no slip wall was the condition in the additional surfaces. The numerical total time was 49s and in the Fig. 4 is possible to observe frames in 11s of simulation. Reynolds's number based in the stream velocity and one cylinder diameter is of 2.15×10^4.

The frames show the velocity plotted in streamlines, the wide wake and narrow wake cited in many studies are visible and in some moments it presents large structures. Until the time 30s the wide wake is behind the left cylinder and in the next second it is in the right one, the same phenomenon happens from the frame 35s to 36s. This event looks like the bi-stable characteristics presented by Kim and Durbin (1988), Alam et al (2003) and de Paula and Möller (2013). It is not possible to assure because of the large time step the simulation generates small quantity of velocity data behind the cylinders for a profound analysis, but the behavior changing the mode without a warning and in non-periodic time are some of the bi-stable flow main characteristics.

Observing the evolution of structures behind the cylinder it is visible that they have vortices of large scales and that is why even with a coarse mesh and large time-step the traditional asymmetric configuration and possible bi-stable behavior are visible. The large scales are the ones formed from the cylinders inside the flow. Möller and Silvestrini (2004) say that in high Reynolds's numbers flow, the viscous force can be neglected comparing with inertia forces, with exception near the wall. For the length scale it is applied the frontal area, or the diameter of cylinder and for the time scale D/U. In this way, it is possible to say that with the small scales occurring in the flow, they get the energy from the large scales that command the flow and even with a simulation that uses large time step it represents the phenomenon.

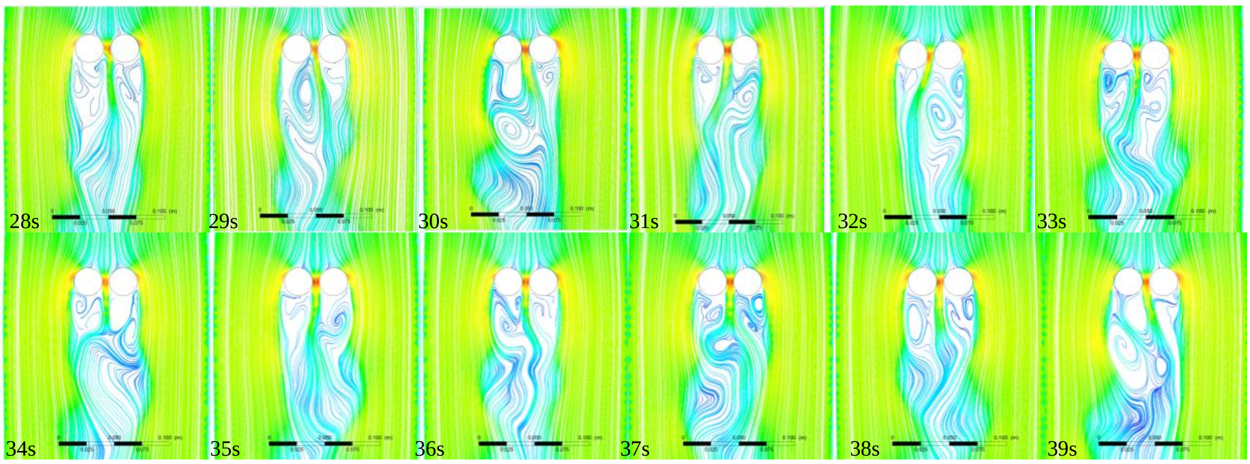
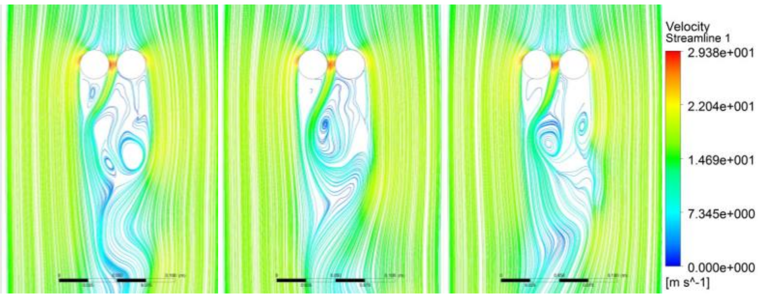


Figure 4. Streamlines frames showing the wakes behavior from 28s up to 39s in a numerical simulation with coarse mesh and time step.

To be sure about that affirmation the analysis of mesh quality is executed applying the GCI method, described on Section 3, and for the best representation of the flow it is recommended to use a mesh with 3,700,000 volumes and that gives a GCI value of asymptotic convergence of 1.59 %.

The simulation with the refined mesh is executed with 0.001s time step, 13.45 m/s inlet velocity, 0.6 % inlet intensity turbulence and 0,001 m length scale, 1 atm pressure outlet with 0.83 % turbulence intensity and 0.01 m length scale. The numerical simulation is executed using $\kappa\omega$ - SAS with the standard coefficients from the software Fluent (2010), during 3.1s for stabilization and after that the following results are taken, as shown in Fig. 5. Observing the Fig. 5 and comparing with Fig. 4, verifies that the structure in the frames are basically the same, with asymmetric behavior and the large vorticities as the presented in the simulation with coarse mesh and large time step.



3.187s 3.192s 3.197s

Figure 5. Streamlines frames showing the wakes behavior from 3.187 s up to 3.197 s in a numerical simulation with 3,700,000 elements mesh and 0.001s time step.

The numerical simulations need a high computational time to give series with the same frequency and large enough to compare with the experimental data, and the present simulation has 0.8 s of results in a frequency of 1000Hz. With this data can be seen the mean velocity in each wake to compare with the experimental values. The points of monitoring are 1D from the cylinder and 3.5D from each side wall and the data series obtained are presented on Fig. 6. The mean velocities are: in the narrow wake 8.15 m/s and for the wide wake 2.57 m/s. The large fluctuation on the narrow wake happens because the gap movement during the flow and changes the velocities more than inside the wide wake.

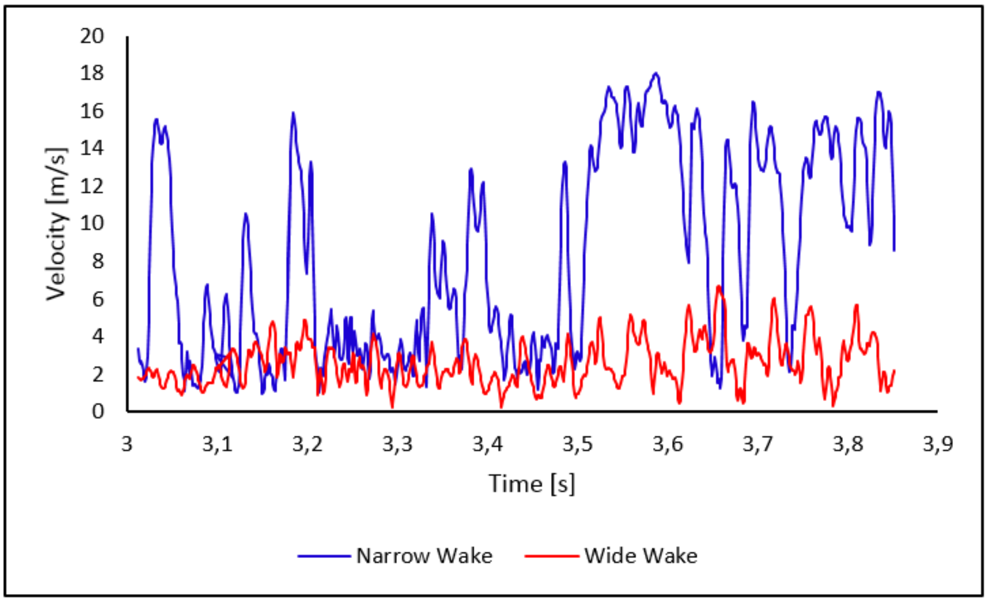


Figure 6. Numerical results for the velocity behind the cylinder 1 and 2

Comparing the results from the numeric simulations with the hydrogen bubble flow visualizations presented by Alam et al. (2003) it is visible that the configuration of the flow is similar with the asymmetric wakes and the large structures. In the Fig. 7, with the pictures side by side, can be seen the big vortices behind the upper cylinder with the wide wake, while in the cylinder below, the narrow wake has smaller vortices that basically recirculate the fluid behind the body. The picture presents a similar shedding vortices position for the large wake with approximately 4D for the experimental visualization and 2D for the numerical result. This difference can be relative to the diameter of the cylinder because in the experimental case the diameter cylinder is 49 mm while in the numerical analysis it is 25.1 mm.

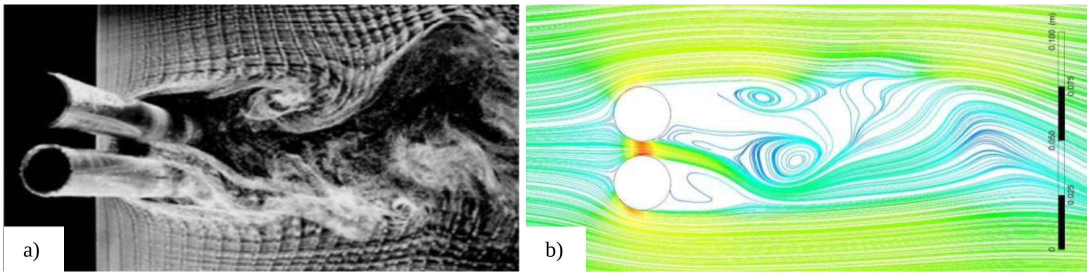


Figure 7. a) Hydrogen Bubble flow visualization – Alam et al. (2003) and b) Present numerical result on 3.197s.

In the Fig. 8 the comparison is made between the water channel dye visualization executed by de Paula and Möller (2013) and in this case the large structures are not visible. The visualization presents the narrow and wide wake that are similar to the structures obtained with the numerical simulation, but in the visualization the upper wake is wider than the presented in the simulation frame. The authors affirm in the paper that the wakes presents predominant two-dimensional behavior, but in the numerical simulation it is visible that the perturbations are in three dimensions because the vortices change direction.

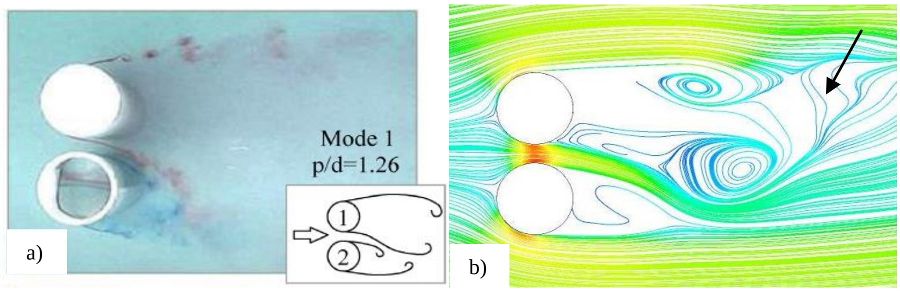


Figure 8. a) Dye flow visualization – de Paula and Möller (2013) and b) Present numerical result on 3.197 s.

As a last comparison it is used the numeric velocity and pressure fields obtained with LES numerical simulation by Afgan et al. (2011), as can be seen in Fig. 9. The first observation is the best detailing showed in the frames (a) and (b), this quality is related with the simulation method adopted, because in URANS ($\kappa\omega$ - SAS) just the mean field is solved in each time step. Despite that, the velocities fields are very similar in structure and qualitatively the magnitudes are equivalent, because the dark blue represents small velocities and yellow/red the large ones. For the pressure fields the definition is higher and it is visible the regions of vortices centers and in the recirculation in the narrow wake with very low pressure (blue) while in the RANS simulation they are smaller and not so visible, once again that is relative to the formulation.

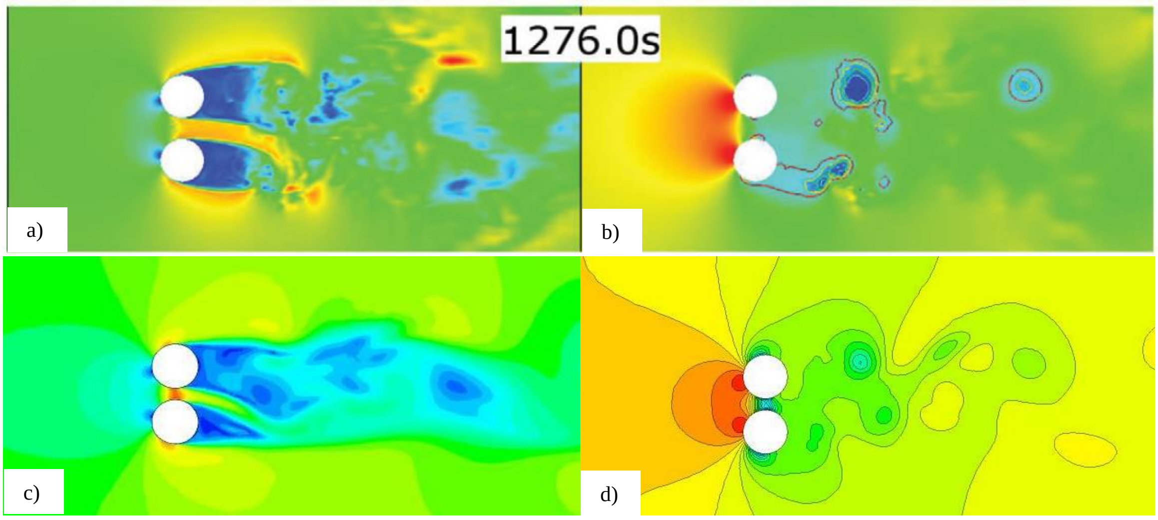


Figure 9. a) Velocity field LES – Afgan et al. (2011), b) Pressure field LES – Afgan et al. (2011), c) Velocity field present numeric result on 3.197s and d) Pressure field present numerical result on 3.197s.

5. EXPERIMENTAL DATA

The experimental data was collected in the LMF aerodynamic channel with the tubes on 690 mm from the entrance using the hot wire anemometry technique. The probe position was 1D behind the cylinder and 3.4D from the right wall. The mean velocity in the flow was 13.45 m/s, the acquisition frequency 3000 Hz and it was applied a low pass filter of 1000 Hz to avoid the aliasing. The raw data obtained in the acquisition is presented on the Fig. 10 where can be seen velocity modes, created by the bi-stable flow, one with high velocity that shows the results inside the narrow wake and other one with low velocity and it is inside the wide wake.

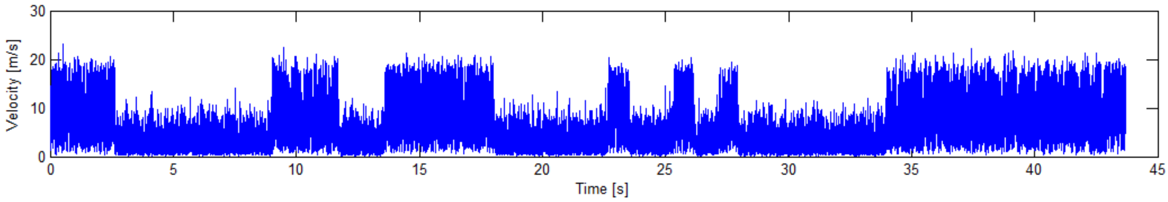


Figure 10. Raw velocity signal behind the right cylinder.

The mean velocity of each mode is, for the narrow wake 8.96 m/s and for the wide wake 3.20m/s. Comparing these values with the numerical ones presented on the Fig. 6, the difference is of 19 % and 9 % respectively, and they are probably caused by the small numerical series.

To evaluate the Strouhal number (St), sections of the signal are made, getting the data from 3 s to 8.5 s for the narrow wake and 38 s to 43 s to the wide wake. This data was adapted to a dyadic series and the FFT power spectra density applied in Matlab. From this analysis the spectra showed in Fig. 11 (c) shows a peak on 70 Hz resulting in a St=0.13 in the narrow wake based on the mean velocity of the flow. For the wide wake, Fig. 11 (b), a small peak in the same frequency and equivalent St is presented. The same analysis, but using the total velocity series is presented on Fig. 11 (a), where just one peak can be seen at 70Hz that gives the same St Number as the obtained on the separated wakes.

The result with just one frequency in both wakes agrees with the results obtained by Xu et al (2003), they shown that for the P/D=1.25 just one frequency was obtained with a Re = 14000 and the St = 0.09. The flow behavior is equivalent to the literature, but the Strouhal value shows difference and that could be caused by the P/D difference and the higher Re in this experimental acquisition.

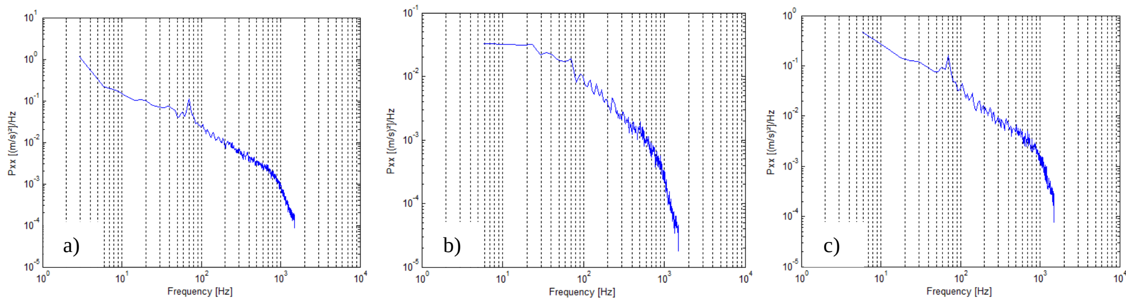


Figure 11. Power spectra density a) Complete series, b) Wide Wake and c) Narrow wake.

6. CONCLUSIONS

The study presents a numerical and experimental analysis of the boundary condition impact in a free channel and the behavior of flow around two side-by-side cylinders. For the study in the free aerodynamic channel, a pressure comparison is executed between the data from pressure taps in the aerodynamic channel wall and the RANS $\kappa\omega$-Standard steady flow. In the flow around the cylinders the RANS with $\kappa\omega$-SAS model numerical simulation is executed and the experimental data collected in the aerodynamic channel with hot wire anemometry technique. Comparisons with hydrogen bubble visualization, dye on water channels visualizations and the numeric results with LES from the literature were used to qualitative validate the results from the numerical simulations.

The study of the velocity profile impact used an approximation of the experimental data in polynomials of third, fourth and sixth degrees. A mesh quality analysis was executed with GCI to show that the one in use represents the phenomenon. The results showed that all the profiles tested, were not representative because they keep adapting on the domain from the beginning of the channel. The case with homogeneous profile, using uniform velocity showed the best behavior, with the initial point diverging but in the other positions presented close values, that keep inside the error margin.

The results presented initially from the numerical simulation for the flow around two side-by-side cylinders used a coarse mesh and large time step showing the presence of the wide and narrow wakes with change the modes that seems to be bi-stable flow. To be sure about the results and if they represent the phenomenon a mesh quality evaluation was executed and simulations with a refined mesh and small time-step were executed, and they showed the asymmetric behavior of the wakes but do not present the change of modes. The reason for that is the small total time, because in the initial simulation the total time is 49s and in the second one 0.8s were executed. Even with the time distinction, the comparison showed that the large scales govern the behavior of the flow. With a large time step and coarse mesh the presented structure is very similar to another one, presenting big vortices. The qualitative comparison between literature flow visualizations and the numeric results were made, and showed similarity in the shape and distribution of vortices as in the comparison between a LES research and the author results.

The experimental data behind the cylinders showed the bi-stable phenomenon and agreed with the presence of just one frequency on the wake. The analysis of each mode showed a mean velocity difference between the numerical and experimental results, showing a difference of 9% in the wide wake and 19% in the narrow. The difference in the narrow wake happens because the large variation in the velocity, visible in the numerical and experimental results. Another reason for the divergence in both wakes is the short numerical series.

7. ACKNOWLEDGMENTS

Authors are gratefully indebted to the CNPq National Council for Scientific and Technological Development, for the financial support. Roberta Neumeister thanks the CNPq for granting her fellowship.

8. 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.

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.

Kim, H. J.; Durbin, P. A.,1988. "Investigation of the flow between a pair of circular cylinders in the flopping regime". Journal Fluid Mechanics, Vol. 196, p. 431-448.

Menter, F.R. and Egorov, Y., 2005. "A Scale-Adaptive Simulation Model using Two-Equation Models". 43rd AIAA Aerospace Science Meeting and Exhibiting 2005-1095.

Möller, S. V. and Silvestrini.J. H., 2004. *Turbulência.* ABCM, Rio de Janeiro,Vol. 4, p.26.

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

Xu, S. J., Zhou Y., and So R.M.C., 2003. "Reynolds number effects on the flow structure behind two side-by-side cylinders" Physics of Fluids, Vol. 15, p. 1214-1219.

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

9. RESPONSIBILITY NOTICE