NUMERICAL AND EXPERIMENTAL FLOW VISUALIZATION OF SQUARE CYLINDER WAKES

Odenir de Almeida
Cláudio Lindquist
Edson Del Rio Vieira
Sérgio Said Mansur
UNESP – Faculdade de Engenharia de Ilha Solteira, Departamento de Engenharia Mecânica
15385-000, Ilha Solteira, SP, Brasil. E-mail: odenir@dem.feis.unesp.br
Aristeu Silveira Neto
UFU – Faculdade de Engenharia Mecânica, Departamento de Engenharia Mecânica
38400-902, Uberlândia, MG, Brasil.

Abstract

Flow visualization is an important tool to understand complex flows around bluff bodies. In this work, using numerical and experimental hydrodynamic flow visualization, different wake configurations produced by a square-section cylinder have been analyzed in moderate Reynolds numbers. Two-dimensional numerical calculations have been performed using Fluent®5.0 software, employing a finite volume method with different schemes of convective transport. The experiments have been carried out in a vertical water tunnel driven by gravitational action and operated in continuous mode. Experimental visualization testes have been done, employing the dye wash and hydrogen bubble techniques. The images of flow patterns have been captured on photographic film, and the results obtained when compared with numerical ones showed good agreement.

Keywords: square cylinder, vortex street, numerical simulation, flow visualization, dye wash and hydrogen bubble techniques.

1. INTRODUCTION

For its wide application in several engineering problems, the flow around square-sectioned cylindrical bodies has been the subject of several researches in the last decades. In fact, many structures used in engineering applications have square or rectangular cross section and are exposed to continuous action of several flow types. These structures can be bridge pillars, energy transmission towers, heat exchangers fins, and others fluidmechanical components.

From the fluid dynamics point of view, this problem is characterized by a high complexity degree, due to the simultaneous interaction of different kind of flows: the boundary layer, the separation zone and the vortex wake. Since the pioneer works of Strouhal (in 1878) and von Kármán (in 1912), the phenomenon of the alternate vortex shedding has been intensively studied. In spite of that, several aspects related to the vortex generation and shedding in the cylinder wake remain obscure.

The intense activity of experimental research verified in this field has been based frequently on results obtained by different flow visualization techniques, which have contributed substantially to the understanding of the phenomena related to vortex dynamics. According to Merzkirch (1987), an additional advantage of flow visualization is that one can obtain qualitative and quantitative data from the flow images, without introducing physical disturbance to the flow field. Several visualization techniques have been employed in order to make evident the flow patterns in cylinder wakes, as can be seen in the works of Hammache & Gharib (1991) and Williamson (1996) that used the smoke technique in wind tunnels, and Williamson (1989) that used dye wash technique for hydrodynamic flow visualization, among many others authors.

Concerning the use of numeric tools in the treatment of external flows, the enormous increase of computers processing capacity and the improvement of numeric methodologies have contributed extensively in the identification and analysis of several types of phenomenons that occurs in flows. A pioneer numerical flow visualizations in the study of the flow around bluff bodies was presented by Fromm & Harlow (1963). Since that, a great variety of numeric works have been conduced. Davis & Moore (1982) presented a study on vortex shedding past rectangular cylinders in infinite domain. In this work, the computational visualization was also used as a tool for results interpretation, obtained good agreement between the numerical and experimental results. Okajima (1982) compares the experimental visualization of the cylinder wakes, for Reynolds number varying from 150 to 600, with numeric visualizations, showing some differences in development of the vortex-shedding phenomenon.

The purpose of the present work is to study some characteristics of the square-sectioned cylinder wake through numerical and experimental visualization, for a low Reynolds number (Re = 150). For the numerical calculations, the computational fluid dynamics package Fluent®5.0, has been used, using three different convective transport schemes. In order to visualize the vortex street, the passive marker particles are injected upstream of the body. The experimental part has been carried out in a vertical water tunnel, where two flow visualization techniques have been used – the dye wash and hydrogen bubble techniques. The numerical results yield good pictures from the cylinder wake which, when compared to the experimental ones, have shown good agreement.

2. OUTLINE OF NUMERICAL MODELING

The momentum and continuity equations for unsteady state, viscous and incompressible flow have been numerically solved using the computational code Fluent®5.0. In this study, the fluid properties are assumed to be constant and the governing equations may be written as follows:

$$\frac{\partial u_j}{\partial t} + u_i \frac{\partial u_j}{\partial x_i} = -\frac{1}{\rho}\frac{\partial p}{\partial x_j} + g_j + \frac{\partial}{\partial x_i}\left(\nu \frac{\partial u_j}{\partial x_i}\right) \tag{1}$$

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{2}$$

The numerical simulations have been computed without any turbulence model, due to the considered low Reynolds number (Re = 150). In the finite volume method, the governing equations have been discretized over elementary control volumes on a cartesian coordinate system. The velocity-pressure coupling has been obtained following the algorithm SIMPLEC (Patankar & Spalding, 1972). The convective terms have been discretized with the third-order QUICK scheme (Leonard, 1979). In the diffusive terms, it has been employed a second-order central difference scheme. For time marching, the first order fully implicit scheme was used, which is unconditionally stable with respect to the time step size. The solution has advanced until the fully developed vortex street has been obtained. The streaklines have been calculated using a Lagrangian formulation, with new particles being injected at a time interval of 0.02s.

The computational domain is rectangular, and has been discretized in the cartesian coordinate system, with 170×90 grid points non-uniformly distributed, as one can see schematically in Figure 1. Most of the grid points have been placed close to the cylinder and in the wake region, in order to improve the simulation accuracy. The closest grid line to the wall body was fixed at distance of approximately $\delta y/B$ = 0,09 in the vertical and $\delta x/B$ = 0,16 in the horizontal direction.

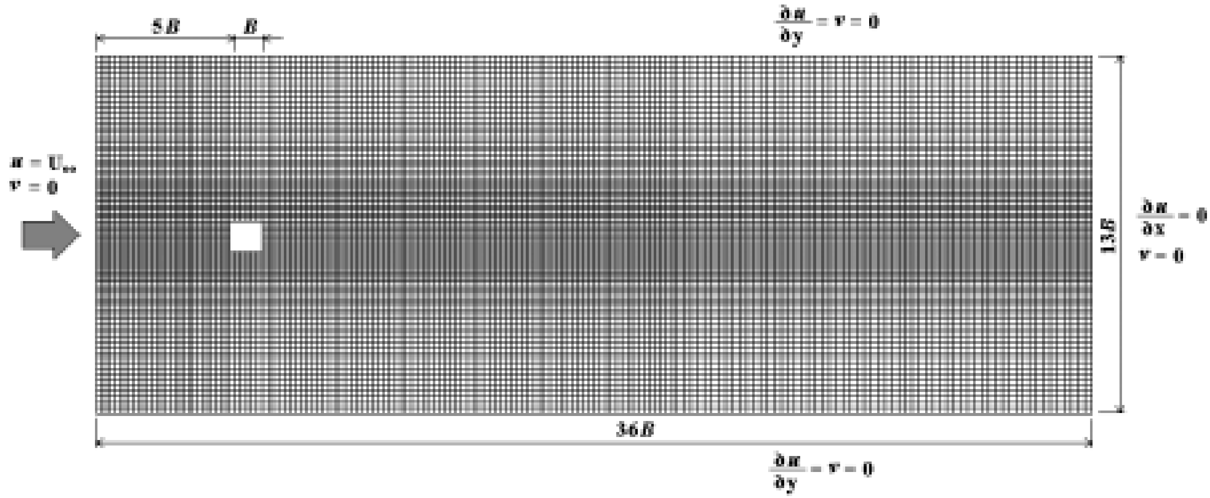


Figure 1. Computational grid and boundary conditions

The streamwise length of the computational domain has been set to 36*B*, where *B* is the dimension of the square cross-section side, while the transversal length has been set to 13*B*. The cylinder has been placed at 5*B* downstream the grid inlet (left side in Figure 1).

The boundary conditions have been set as follows. At the upstream boundary, the inlet flow has been assumed to be uniform. At the outlet, a zero gradient boundary condition for both u and v has been used. It is important to emphasize that if the grid outlet is placed sufficiently far from the body, this Neumann-type boundary condition usually works well, Sohankar (1998). A no-slip condition has been prescribed at the body surfaces ($u = v = 0$). At the upper and lower boundaries a symmetry conditions was used ($v = \partial u / \partial y = 0$).

3. EXPERIMENTAL SETUP

Qualitative experimental results have been obtained from flow visualization tests. These have been conducted in a vertical water tunnel, operated by gravitational action in continuous mode. The test section size is 146×146×500 mm and the free stream turbulent intensity is less than 1%. More details about this water tunnel − illustrated in the Figure 2 − facility and its operation are available in Vieira *et al.* (1997).

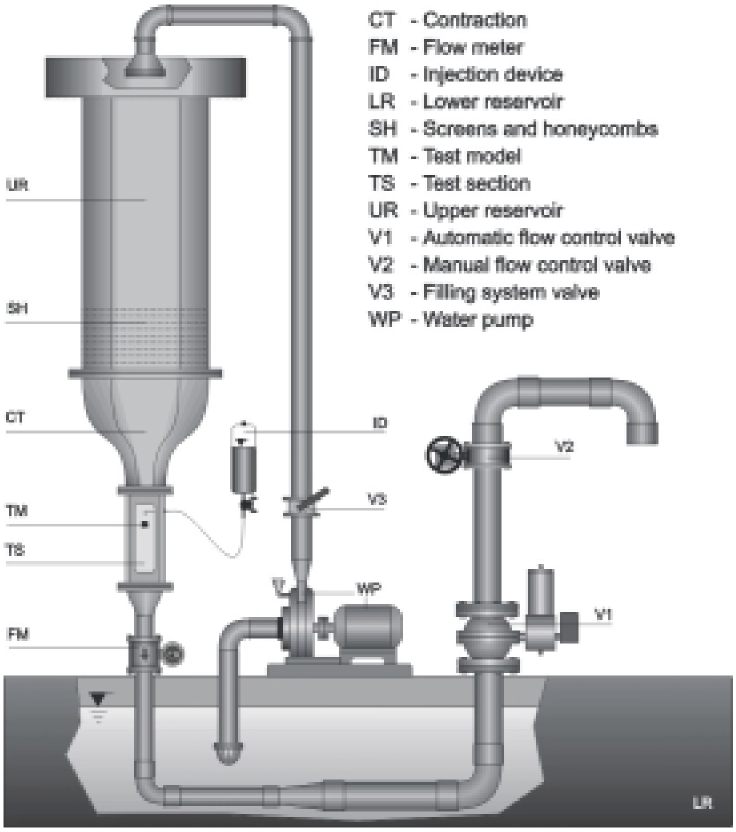


Figure 2. Water tunnel operated by gravity action.

Two flow visualization techniques have been employed in this work. The first, called dye wash technique, consists on adequate injection of opaque liquid dye into the non-disturbed flow field upstream to the solid body by means of a needle. Then, dye injection is suddenly stopped, the needle is removed, and the clean water stream washes the flow field, except the regions where the flow speed is relatively small, as in the cylinder boundary layer and wake. This procedure enables to visualize, for some seconds, the vortex street downstream the cylinder. In the present work, the section test was illuminated in back-light by eight flooding lamps of 150 W, providing an uniformly diffuse bright background against which the dye patterns have been photographed. The dye employed is strong aqueous solution of black PVA pigment.

The second visualization technique used is the classical hydrogen bubble technique. A thin tungsten wire ($d = 0{,}075$mm) has been stretched through the test section side walls upstream the cylinder, working as the negative electrode (cathode). The anode (positive electrode) simply consists in a metallic object in electrical contact with the flow, placed near the cathode. If the applied electric tension to the electrodes is large enough, the water hydrolysis takes place, with the formation of hydrogen bubbles on the cathode and oxygen on the anode. Parallel bubble lines may be obtained, if one vary the electric tension in the form of square wave pulses. For this visualization technique, the illuminating apparatus consists in two flash units, placed near the test section side windows and forming with them an angle of approximately 30°, yielding a good contrast between the bubbles and the test section back window. The flow images have been captured using a Nikon F4s camera equipped with a Nikkor 60 mm micro lens and an ISO 100 photographic film.

The free stream velocity inside the test section has been measured with an Yokogawa ADMAG AE208MG electromagnetic flow meter mounted downstream the test section. The overall uncertainty estimate in the Reynolds number has been calculated to be less than ±5%.

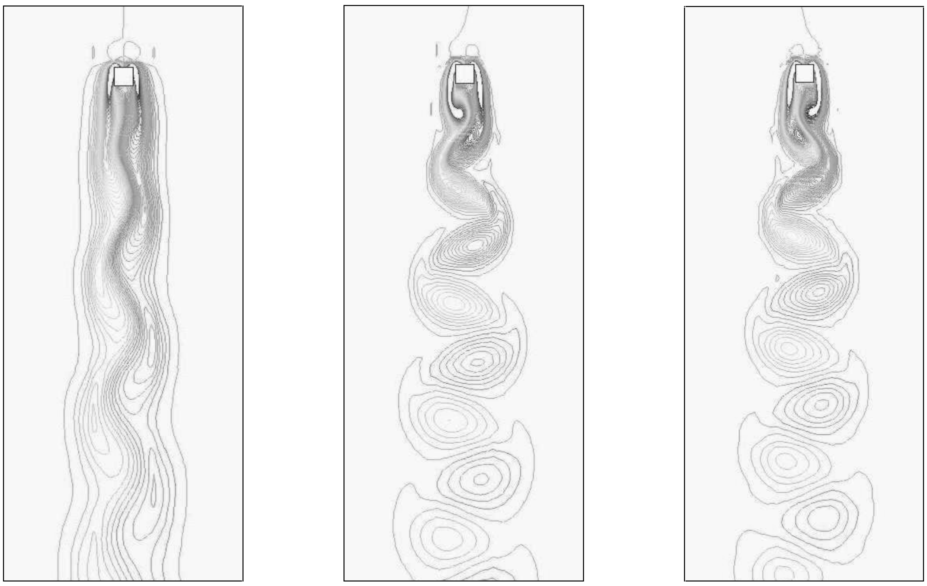
4. RESULTS AND DISCUSSIONS

It is known that the use of different convective transport schemes for the numeric solution of the same problem of fluid flows, can supply completely different results. In fact, the numerical diffusion or false diffusion, associated to the use of non-exact interpolation functions can harm the obtaining of a result physically correct. To verify this influence in the numerical results, three simulations test with different convective transport schemes available in the computational code were developed: a) First-order UPWIND scheme; (b) Second-order UPWIND scheme; (c) QUICK scheme.

The Figure 3 presents the flow field visualization, through isovorticity plots, considering a dimensionless time $t = U_{\infty} t' / B$ = 225. It has been verified that the use of low order schemes as First-order UPWIND in this type of flows, generates high levels of numeric diffusion, what introduce an artificial viscosity in the fluid. That is verified in Figure 3(a) by the strong flattening of the vortex wake behind the cylinder. On the other hand, the use of higher order scheme facilitates the obtaining of quite representative results for the shedding vortex process. However, it is observed that for the same instant of computational time, different configurations of the cylinder wake are obtained. In a certain way, that difference can be attributed to the presence of numeric oscillations in the schemes of this type.

In Table 1, results of useful physical quantities were computed comparing the different convective transport schemes. The quantities presented are respectively, Strouhal number (St), Mean drag coefficient (C_D), RMS drag coefficient (C_D'), RMS lift coefficient (C_L'), Stagnation pressure coefficient at centerline (C_{ps}) and base pressure coefficient at centerline (C_{pb}). These values are compared with numerical data obtained by Sohankar (1995) using the QUICK scheme. The values obtained by QUICK scheme, in this work, shows good agreement with the numerical data registered by Sohankar (1995). It is also verified that the values obtained with the use of the First-order UPWIND scheme are smaller than others.

Figure 4(a) present the experimental flow patterns for one vortex shedding cycle, visualized with the dye wash technique. Numerically computed isovorticity plots are shown for comparison, Figure 4(b). The numerical results presented in the Figure 4(b) have been obtained with the SIMPLEC algorithm and QUICK scheme.



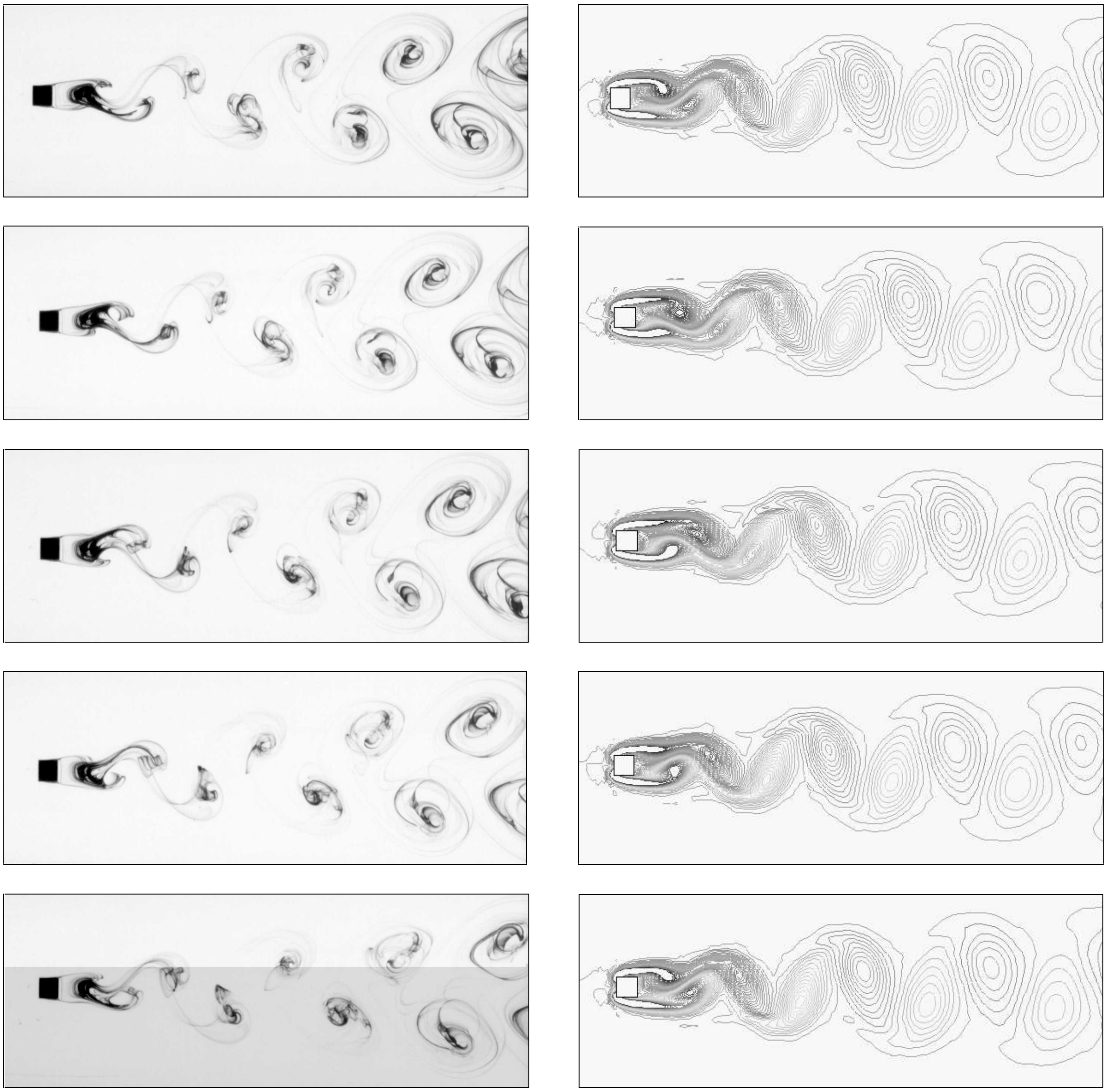
(a) 1st order UPWIND (b) 2nd order UPWIND (c) QUICK

Figure 3. Influence of the transport convective schemes on flow pattern calculation around a square cylinder.

Table 1. Computed parameters from the flow field around a square cylinder (Re = 150).

Reference	Scheme	St	C_D	$C_{D'}$	$C_{L'}$	C_{ps}	$-C_{pb}$
Present work	$1^{st.}$ UPWIND	0.120	1.5609	0.0003	0.0468	1.10	0.60
	2^{nd} UPWIND	0.154	1.558	0.0057	0.1896	1.09	0.757
	QUICK	0.161	1.544	0.0045	0.1667	1.11	0.742
Sohankar et al.(1995)	QUICK	0.161	1.408	0.0061	0.177	1.03	0.730

The results show a good agreement between the real and the numerical prediction of flow field. However, some discrepancies are seen over the prediction of formation zone size located in the cylinder near wake. The numeric results show a smaller formation zone than in the real flow field. It can be seen that the Von Kármán vortex street in Figure 4(b) are more regularly shaped and less spaced than those in Figure 4(a).



(a) Experimental (b) Numerical

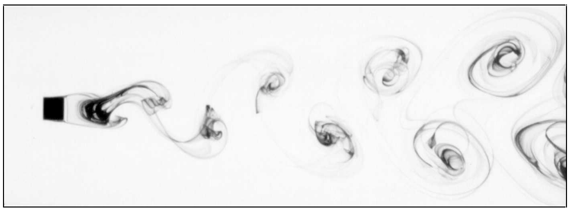
Figure 4 - Comparison between experimental (dye wash) and numerical (isovorticity) visualization of the flow around a square cylinder for one vortex shedding cycle (Re = 150).

In this regime (Re = 150) the flow state is completely laminar exhibiting a periodic and characteristic wake defined by well shaped vortices. In agreement with Zdravkovich (1990), the formation of Karman vortex street in this regime produces a rise in drag coefficient due pressure (C_{Dp}) and a fluctuating lift coefficient (C_L').

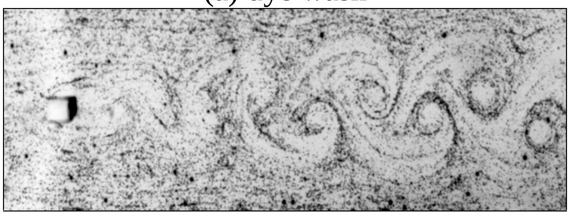
From this numerical analysis, it was identified that the regions of larger vorticity (magnitude) are positioned in near frontal corners of cylinder and that the relative major values of vorticity occur inside the vortices presents in the wake. Another important observation, in this study, is regarding to location of the passive marker particles in flow field, particularly, inside the vortices. It is observed that the particles introduced ahead of the obstacle were transported for regions of higher vorticity located inside these structures. The results found in this work are in agreement with similar data of Davis & Moore (1982).

It will be seen later that this is also in agreement with experimental visualization dye wash technique. This phenomenon was not verified in the hydrogen bubble technique.

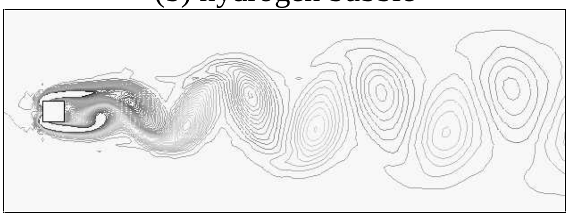
The Figure 5 shows the comparison between different experimental and numerical visualization techniques of cylinder wake employed in this work. The photos 5(a) and (b) illustrates the dye wash and hydrogen bubble techniques for flow field visualization. It is verified a contrast among the two wakes observed.



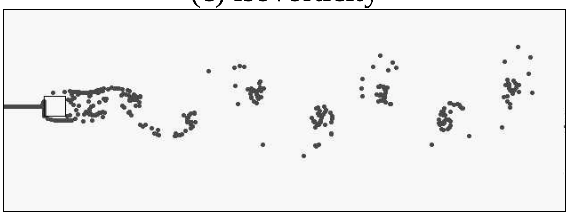
(a) dye wash



(b) hydrogen bubble



(c) isovorticity



(d) streakline

Figure 5. Comparison between the experimental and numerical flow visualization techniques employed in the preset work (Re = 150).

The Figure 5(b) shows a wake more regularly defined with vortices less spaced than the Figure 5(a), however, it is not possible to have a good visualization of the formation region behind the cylinder using the hydrogen bubble. In this case, is visible that the use of different technique for visualization can supply distinct qualitative results. This is because, mainly, the nature of employed experimental technique achieving results influenced by the varying laboratory conditions, such as temperature, for example. It is interesting to emphasize that the flow visualizers as hydrogen bubbles are not passive in flow field due the flotability effects.

The Figures 5(c) and (d) shows the numerical prediction of cylinder wake with visualization isovorticity plots and passive marker particles, respectively. The results in Figure 5(d) were obtained for to validate the discrete phase modeling available in computational code. Although being a test case, the results provided an excellent mean for visualizing the motion of vortices. The numerical flow visualization shows that the particles are swept into vortices and consequently are shed with them composing the Von Karman vortex street. It is identified that the position of vortices are corresponding those obtained in Figure 5(c).

It can be seen that the regions of high dye concentration in Figure 5(a) are numerically represented by regions of high concentration of particles, as observed in the Figure 5(d). This tendency is important to localize the center of vortex. On other hand, this is not verified in the Figure 5(b) with hydrogen bubble technique once the bubbles were positioned in the contours of vortices.

5. CONCLUDING REMARKS

The flow around a square cylinder at Reynolds number 150 were numerically and experimental conducted. The numerical results were visualized by vorticity and streakline plots. For the numerical simulation procedure utilized have been used the SIMPLEC algorithm and QUICK scheme for convective transport terms. Two different techniques of experimental flow visualization were employed – dye wash and hydrogen bubble. The qualitative results shown good agreement between numerical and experimental approach. Some discrepancies were identified as a largest difusivity of vortices and a smaller length of the formation region for the numerical case. The comparative results shown a better agreement between the numerical and the hydrogen bubble visualization techniques. Still, qualitative and quantitative numerical results were presented for comparison among different schemes of convective transport. Finally, the use of techniques of experimental and numerical visualization are constituted in useful tools that aid in the understanding of several types of phenomena in flows, contributing each one to the improvement of the technology.

6. ACKNOWLEDGMENTS

This study has been developed with financial support provided by FAPESP and FUNDUNESP.

7. REFERENCES

Davis, R. W. & Moore, E. F., 1982, "A numerical study of vortex shedding from rectangles", *J. Fluid Mech.*, vol.116, pp.475-506.

Fromm, J. E. & Harlow, F. H., 1963, "Numerical solution of the problem of vortex street development", *Physics of Fluids*, vol.6, n.7., pp.975-982.

Hammache, M. & Gharib, M., 1991, "An experimental study of the parallel and oblique vortex shedding from circular cylinders", *J. Fluid Mech.,* vol.232, pp.567-590.

Leonard, B. P., 1979, "A stable and accurate convective modelling procedure based on quadratic upstream interpolation", *Comp. Meth. Appl. Mech. & Engng,* pp.59-98.

Merzkirch, W., 1987, "Flow Visualization", ed.2, Academic Press, Orlando.

Okajima, A., 1982, "Strouhal numbers of rectangular cylinders", *J. Fluid Mech.*, vol.123, pp.379-398.

Patankar, S. V. & Spalding, D. B., 1972, "A calculation procedure for heat and mass transfer in three-dimensional parabolic flows", *Journal of Heat and Mass Transfer*, vol.15, pp.1787-1806.

Sohankar, A. E., 1998, Numerical study of laminar transitional and turbulent flow past rectangular cylinders, Tese de Doutorado – Gothenburg.

Sohankar, A., Davidson, L. & Norberg, C., 1995, "Numerical simulation of unsteady flow around a square two-dimensional cylinder", *Proceeding 12th Australasian Fluid Mechanics,* Conference, R. W. Bilger (ed.), pp.517-520.

Vieira, E.D.R., Lindquist, C.; Woiski, E.R. & Mansur, S.S., 1997, Vortex Velocity Measurement by Hydrodynamic Flow Visualization. Proceedings (in CD-ROM) of COBEM / 97 XIV Brazilian Congress of Mechanical Engineering, paper code 059, 8pp, Bauru.

Williamson, C.H.K., 1989, Oblique and parallel modes of vortex shedding in the wake of a circular cylinder at low Reynolds numbers, Journal of Fluid Mechanics, vol.206, pp.579-627.

Williamson, C. H. K., 1996, "Three-dimensional vortex dynamics in bluff body wakes", *Experimental Thermal and Fluid Science,* vol.12, pp.150-168.

Zdravkovich, M. M., 1990, "Conceptual overview of laminar and turbulent flows past smooth and rough circular cylinders", *Journal of Wind Engineering and Industrial Aerodynamics*, vol.33, pp.53-62.