ROTATIONAL EFFECTS ON VORTEX SHEDDING BEHAVIOUR OF A CYLINDER WITH SURFACE ROUGHNESS – AN EXPERIMENTAL PIV AND COMPUTATIONAL APPROACH

[1]M.T. ABU SEMAN, [2]F. ISMAIL, [3]H. YUSOFF

[1]Department of Mechanical Politeknik Seberang Perai, [2]School of Mechanical Engineering University Sains Malaysia,
[3]Faculty of Mechanical University Technology MARA (Pulau Pinang)
E-mail: mizie@psp.edu.my, aefarzad@eng.usm.my, hamidyusoff@ppinang.uitm.edu.my

Abstract- Mostly a wake flow can contribute to unwanted vortex shedding and vibration of different frequencies. If this phenomenon continues for a long time, a bluff body is likely to experience damage caused by inconsistent flow, which can lead to failure. This paper examines vortex shedding behaviour behind a rotating cylinder with surface roughness. Experimental and numerical two-dimensional approaches are taken. In a wind tunnel experiment, there is free-stream velocities of uniform incompressible flow around the circular cylinder (D=50mm). Surface roughness at 5.0μm<Ra<80μm is applied to the cylinder surface. The Reynolds number is in the range 2000<Re<42000, based on the cylinder diameter, and the rotation rates are 0<α<7.9 (ratio of the surface speed) and the free-stream speed is varied in the range 0.65–13.21m/s. A comparison is made with a stationary cylinder and three degrees of surface roughness on rotating circular cylinders. Sand paper is used to establish the rough profile. Particle Image Velocimetry (PIV) measures flow behind a rotating cylinder and captures images of vortices shedding during the experiment. Computational Fluid Dynamics (CFD) simulation is carried out to verify and analyse the vortex shedding formation. This can indicate the most suitable placement of the experimental device so the roughness suppresses vortex shedding. The PIV and CFD results are compared and confirm that the velocity behind a rotating cylinder at low and high speeds is affected by surface roughness. It is concluded that vortex shedding in the flow behaviour behind a rotating cylinder is generated in the far-wake of a cylinder with surface roughness.

Keywords- Circular cylinder, Surface roughness, Reynolds number, Lift and drag coefficient, Velocity profile.

I. INTRODUCTION

The effect on fluid dynamics when a circular cylinder is placed in flow has been studied, to understand the practical implications. Typical industrial applications which involve rotating cylinders include risers associated with marine engineering, offshore structures, buildings and bridges, pipelines, and heat exchanger tubes. Substantial areas in the flow can contain unsteady forces that develop from vortex shedding in the wake. These movements can potentially cause damage to the structure of bluff bodies. Research into these phenomena has attempted to identify methods to reduce the detrimental forces that develop when flow passes around objects. Sumer observes that at low Reynolds number (Re) there is symmetric flow past a circular cylinder.

However, increasing Reynolds number results in separation behind the cylinder, this interrupts the flow and vortex shedding can develop. At 40<Re<200, laminar vortex shedding takes place in the wake and at Re=200–300 there is a transition to turbulence. The peak of turbulent flow leads to laminar boundary layer separation at the subcritical region $300<Re<3x10^5$. Payne carried out some initial investigation into flow disturbance at Re=40–100. Both experimental and numerical analysis has been used to investigate flow at different Reynolds number, which is reported in the literature. The simulations that are carried out using Computational Fluid Dynamics (CFD) techniques are predominantly two-dimensional.

Research into flow patterns that occur behind a rotating circular cylinder is sparse, but contributions have been made through experiments carried out by Badr et al., and Kimura et al.. The experiments have concentrated on speeds of $\alpha \leq 2.5$ and visual inspection of flow behaviour and vortex shedding. The observations are at rotational speeds below a critical value, which is approximately $\alpha<2.0$ at $Re>1000$. It was found that flow in the wake became asymmetric and there was deflection of the cylinder to one side at $\alpha>0$. Until now the literature has not considered quantitative measurement of velocity fields and vorticity in the wake of a rotating cylinder, but concentrated on the vortex dynamics.

Studies on flow have examined rotating oscillating circular cylinders. The oscillation has either been in-line, lateral, or rotational. Lock-in is observed when the lateral oscillation frequency synchronises with the natural vortex shedding frequency. During lock-in, regularity and two-dimensionality predominate in vortex shedding and the patterns of vortices. In contrast, lock-in associated with in-line oscillation of rotating cylinders is initiated when the frequency of the fluctuating drag force is approached, due to natural vortex shedding. Moreover, when the cylinder oscillation and natural vortex shedding frequencies begin to coincide, synchronisation is introduced between the vortex shedding and cylinder movement in rotationally oscillating cylinders. These attributes facilitate studies of the shedding mechanism and vortex street dynamics and development. Suppression of vortex shedding is possible by controlling the

interface between cylinder oscillation and natural vortex shedding frequencies, particularly at very high speeds.

The association between flow and cylinder movement can begin to form when a circular cylinder is rotating at a steady speed. An example is potential flow, in which the rotation results in acceleration of flow on one side and deceleration on the other. Flow is normally a consequence of the Reynolds number and the non-dimensional rotation speed. This is expressed as the ratio of circumferential speed at the cylinder surface to free-stream velocity of the uniform cross-flow ($\alpha=\omega D/2U_o$) where, D is the diameter, ω is the angular rotation speed of the cylinder, and U_o is the free-stream velocity.

To predict flow, there is a range of aspects that require clarification and both experimental and computational approaches are beneficial. In this research the focus is on observation and quantification of two-dimensional (2D) flow behind a counter-clockwise rotating cylinder. The cylinder is modified by different degrees of surface roughness. Jose Luis Duarte Ribeiro studies drag forces and pressure associated with roughness in the range $50000 \leq Re \leq 400000$ and unevenness heights $0.0018 \leq k/d \leq 0.0123$. His experiments involve surface modification using sand paper, wire mesh screen, and ribs, in which the application of ribs produces the smallest relative difference. Shih et al. demonstrates the flow on smooth and rough cylinders in steady and unsteady flows over a maximum achievable range of Reynolds number. However, both experimental approaches involve a stationary circular cylinder and results are validated against similar studies reported in the literature.

The aim of our research is to analyse data originating from experimental and numerical investigation of vortex shedding behaviour. The research is concerned with four grades of surface roughness and an atomiser is used for seeding in the towing tank of a wind tunnel. The effect of Reynolds number in the range $2000<Re<42000$ and varied rotation rates are measured. The focus is on identifying changes in vortex shedding behaviour associated with different rate ratios in respect of rotation and roughness. In addition, the results are reviewed by a comparison of Particle Image Velocimetry (PIV) and CFD findings.

II. METHODOLOGY EXPERIMENTAL SET-UP

Analysing the scaling used in the construction of the cylinder is an important consideration, to ensure the experimental results are reliable and suitable for analysis. Cylinders with degrees of surface roughness are prepared (Table 1). Lightweight materials are used to fabricate the cylinders. The body is made of hollow (D = 50mm) aluminium tubing and the cylinder length is 300mm. The roughness is created by interspersing the surface of a smooth cylinder with non-uniform pieces of sandpaper. Profile analysis of the resulting surface roughness is carried out (Alicona IF-Profiler).

Table 1: Average roughness profiles of the circular cylinders

Type	Roughness
	Ra (μm)
Cylinder A - (Smooth)	5.078
Cylinder B - (SiliconCarbide#80)	39.88
Cylinder C - (Silicon Carbide #60)	60.15
Cylinder D - (Silicon Carbide #36)	76.99

An open-circuit wind tunnel is used to measure the forces produced by rotating cylinders with different surface roughness. The rotation is controlled bya lightweight motor attached to the cylinder which is placed in a 300mm x 300mm test section of a wind tunnel. The maximum velocity is 36 m/s, moderated by an electronic control. A force measurement system, data acquisition unit, and a computer, form the measurement set-up (Fig. 1).

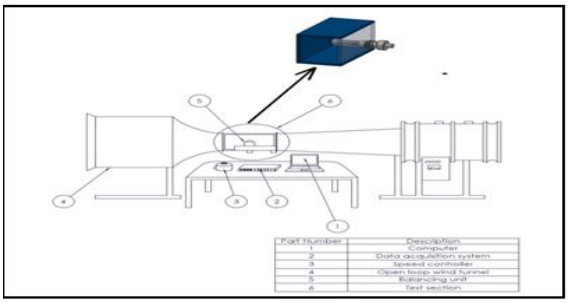


Fig. 1: Schematic of the experimental set-up

West and Apelt report a slight pressure distribution around a circular cylinder dependent on the blockage ratio. Where the blockage represents less than 6% of the wind tunnel, the effects are negligible. In a study by Park et al., there is a 4.2% blockage ratio for a 2D cylinder and their conclusions confirm the negligible blockage effect. Based on these finding, the blockage effect is disregarded for the purposes of the current study.

An AC motor with a maximum rotation of 10000 rpm generates the motion. Slight modification to the casing and shaft is carried out, the total weight of the lightweight smart motor (LWSM) is under 1kg, and the assembly method is simple. The motor is mounted horizontally inside the wind tunnel without any support. The rotation of the motor causes minimal vibrational effects on measurements of the aerodynamic forces. The overall simplicity, rigidity, and precision of joints between the motor and cylinder, result in an efficient and robust experimental set-up (Fig. 2). A two-step calibration

of the motor takes place, the first carried out before mounting in the wind tunnel. A magnetic gauge is used to contain displacement due to vibration at points on the cylinder to around ±0.15mm. The subsequent calibration in the wind tunnel first takes readings without free-stream velocity then with incoming free-stream velocity as is repeated before each test.

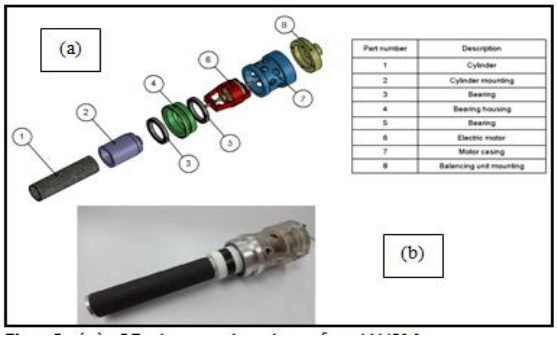


Fig. 2 (a) 3D Isometric view for LWSM components (b) Complete LWSM set

PIV is used to measure flow behind the counter-clockwise rotating circular cylinder. It captures images of the vortex shedding during each experiment. The PIV consists of a CCD camera (Digital monochrome progressive scan camera: Flow Sense M2 10bits) with a 1600 x 1186 pixels spatial resolution image. A continuous-wave Nd:YAG laser captures images which are then stored in the computer. The flow area is illuminated by long exposure to a laser-generated light sheet and a mirror reflects it 90^o. A convex and cylindrical lens produces a narrow sheet of light. In the test section the laser sheet is 1.0 mm and the thickness is controlled. Seeding using cornoil in the atomiser is used during visual inspection. The oilis released at high pressure into the test section of the wind tunnel by a central air unit. The average corn oil particle diameter range is 1–2μm.

Images are captured at intervals of 100 ms to record the evolution of flow structure behind the rotating cylinder. The regularity corresponds to about 1% in the characteristic formation period of a large-scale vortex. Using the cross-correlation 2-frame method, 800 simultaneous velocity fields are obtained.

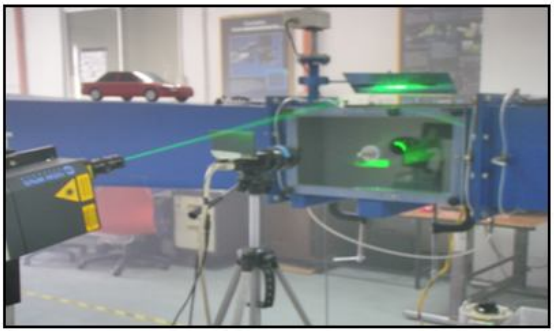


Fig. 3 PIV set-up

A comparison is made of the wake behind a cylinder with Reynolds number $2000 \leq Re \leq 42500$. The field of view is 120 x 120 mm^2 (Fig. 3).

III. NUMERICAL SET-UP

ANSYS FLUENT® commercial CFD software simulates the flows involving rotation and surface roughness. The geometry of the cylinder is modeled with the R_a value to simulate the surface roughness. The surface roughness is interpreted as a collection of minute step function profiles along the surface, based on approximation of the actual cylinder surface profile (Fig. 4). There are, nevertheless, limitations inherent in this method, as finer irregularities cannot be detected. To improve the method, it is possible to approximate the actual rough surface onto the geometry using the average heights of the irregular surface. The irregular roughness is measured in micron per meter (μm)**.**

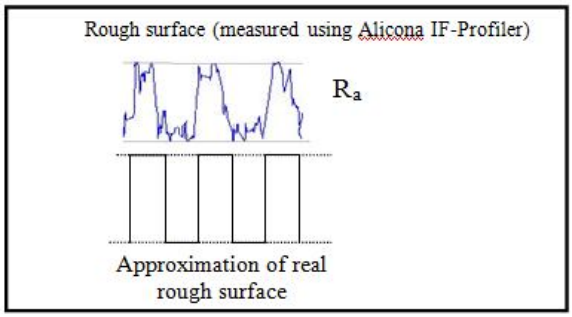


Fig. 4 Principal schemes of roughness

Simulations at various Reynolds number based on the same cylinder configuration in the experimental set-up are used to predict the forces around the rotating cylinder ($2000 \leq Re \leq 42000$). With the cylinder centre at x=y=0, the simulation confirms $-0.7 \leq x/D \leq 1.7$ and $-0.3 \leq y/D \leq 0.3$, where x and y denote respectively the stream wise and transverse directions. At the velocity inlet on the left, uniform inflow (Ui) enters and then exists via the pressure outlet at the right (Fig. 5). The rotational velocity is prescribed and there is no-slip conditions at the cylinder wall, similar to testing carried out by Stojkovic et al., Karabelas et al., Srinivas et al. , Fujisawa et al. , Rajani et al., and Mahbubar et al.. In the computational study, steady incompressible RANS equations are solved using implicit and segregated methods. The convective terms and viscous terms are discretised by second order upwind and central discretisation techniques respectively. The SIMPLE algorithm is used for coupling of the pressure and velocity.

Several blocks comprise the flow domain with multiple mesh densities to facilitate capture of local flow resolution at large velocity gradients. Micron-sized spacing and sizing is used in the design of the mesh points and surface roughness profile respectively, interspersed along the cylinder designed

in ABAQUS software then converted to gambit files. Commercial mesh generation package GAMBIT is used in the mesh generation. The simulation is based on a typical structured mesh (Fig. 5). The y+ value near the cylinder surface is close to 1 to capture the effect of surface roughness.

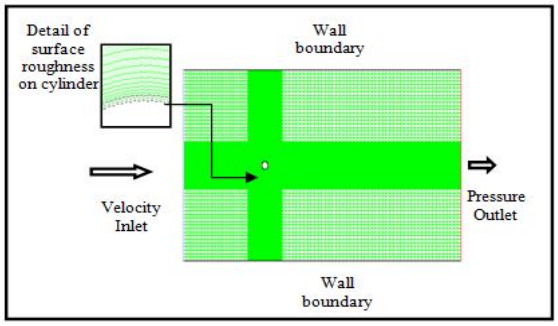


Fig. 5 Domain and grid mesh

The turbulent flow is modelled using Spalart-Allmaras (S-A) as it is cost-effective and highly accurate in simulations involving boundary layers exposed to adverse pressure gradients, as discussed by Willis et al., Krishnan et al., Martin et al., Takahashi et al..

IV. RESULTS AND DISCUSSION FLOW PATTERN

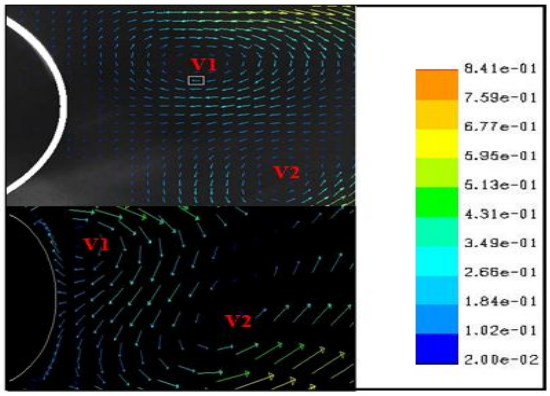


Fig. 6 Experimental versus simulation - Vector flow pattern at Re=2083 and zero rotational rate for Cylinder A: Label V1 and V2 are field-generated vorticities behind the cylinder

The experiment and simulation results (Fig. 6) show unsteady flow for Re = 2083 at zero-rotating (Cylinder A) in the near-wake directly behind the cylinder. Using phases for the same Reynolds number, the velocity vector directions and corresponding vorticity contours are visible, confirming the alternating formation, convection, and diffusion of vortices. Both images show velocity vectors are present in a high velocity near-wake region. A saddle point (solid circle) indicates the unsteady boundary layer separation at the surface of the cylinder (V1). The PIV shows an enlarged vortex core with an almost similar pattern to the numerical representation. The magnitudes of velocity also have similar values, especially adjacent to the cylinder. The disparity is in the delay in vortex formation behind the cylinder at location V1. The cause is the turbulence model used in the simulation.

After making a comparison of the methods, the S-A model is deemed incompatible with the flow type generated by a rough cylinder. The incidences of vortex shedding show small flow reversals within the shedding circle, particularly evident in the simulation. Over a long time period the phenomena would result in damage and potentially failure of bluff bodies.

The velocity vectors all have low magnitude values adjacent to the cylinder because the free-stream velocity is only 0.65 m/s. A small saddle point is triggered in the near-wake area behind the cylinder (V2). There is severe distortion as the vortices are still forming in relation to the vortex point. This indicates a reversal of flow vector direction is taking place after the saddle point. A vortex core with a small configuration is generated, but will not affect a large bluff body. The movement of shed vortices in the far-wake region, is at a nearly constant convective velocity. The near-wake flow structure is similar in both the experimental and simulation approaches and concurs with findings observed by Kuo et al. [9].

V. VELOCITY PROFILE

Figure7describes the dimensionless velocity profile at the symmetry line (y=0)in the range $10000 \leq Re \leq 20000$. It shows the differences between the angles at location $y/D<0.008$. At ±90°there is recirculation at the lower surface of the rough cylinder, with maximum velocity at $\theta \geq 60°$. The maximum velocity for Re>20000 is larger than for Re<10000. Kumarasamy and Barlow report vortex suppression in a stationary cylinder may result from the stability mechanism, as the kinematic conditions at the wall minimally affect the critical gap ratio. In a stationary cylinder a strong correlation between the periodic vortex mechanism and the position of the maximum mean velocity in the gap, is reported by Kim et al .They also note that regular vortex formation takes place in similar regions to those identified in our study. As the maximum velocity is at the lower surface of the cylinder, it suggests the surface roughness has affected the separation process. Therefore, an investigation of separation is essential to understand mechanisms to reduce drag.

Velocity profiles for variations in Reynolds number show at Re>20000 the velocity magnitude after separation is higher than at Re<10000, in each case the maximum value is at the lower surface of the cylinder. A fully-developed recirculation region occurs on the cylinder surface at 90°. The area of recirculation for Re<10000 is larger than for Re>20000, probably because of more dominant

vertical effects from rotation in lower Re. However, dimensionless velocity (V/U_∞) values indicate high velocity for Re>20000 relative to Re<10000, evidencing larger inertial effects.

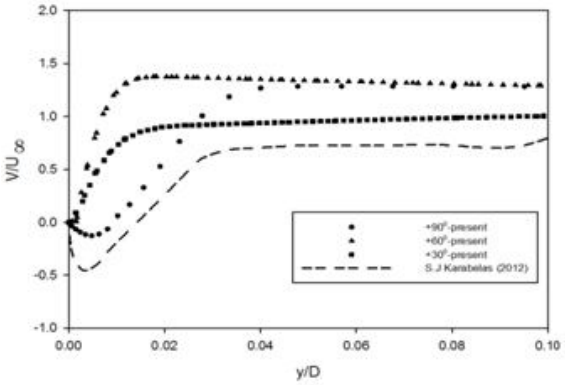


Fig. 7Velocity profile against y/D at azimuth angle +30°, +60° and +90° for non-rotating cylinder in range 10000≤Re≤20000 (negative angles are not included due to symmetry) [24]

CONCLUSION

The effect on flow of a rotating circular cylinder with surface roughness has been investigated. Velocity and the intrinsic nature of flow patterns are investigated for a range of Reynolds number and speed of rotation. The lightweight motor configuration facilitates the rotation with minimal vibration or impact on the accuracy of the wind tunnel measurements. From the results of the current study the following conclusions are reached.

1. The flow characteristics show comparable behaviours in the experimental and simulation approaches. A two-dimensional method is very predictive of turbulent flow, due to faster convergence. However, the potential of the method has not been fully investigated, as this study is restricted to velocity magnitude and flow characteristics. More expansive studies, to include drag force, would be beneficial in verifying the applicability of this method.

2. Surface roughness causes a delay in flow separation in the near-wake of a rotating cylinder, which decreases pressure difference across the areas immediately in front and behind the cylinder. From this the conclusion reached is that drag coefficients are lower when there is surface roughness.

ACKNOWLEDGMENT

Extracts contained in this paper originate from studies that took place at University Sains Malaysia, sponsored by the Ministry of Higher Education, with additional support from the Department of Mechanical Engineering, Polytechnic Seberang Perai, Penang, Malaysia. Particular gratitude is extended to Mr M. Najib, for invaluable assistance in conducting the simulations.

REFERENCES

[1] Zdravkovich M M. Flow around circular cylinders,volume 1: fundamentals, Oxford University Press Inc., New York, 1997.

[2] Sumer, B.M, Hydrodynamics Around Cylindrical Structures, World Scientific, Singapore (1997).

[3] Payne, R.B. Calculations of Unsteady Viscous Flow Past a Circular Cylinder, J. Fluid Mech. 4, (1995), 81

[4] Pietro Catalano, Meng Wang, Gianluca Iaccarino, Parviz Moin Numerical simulation of the flow around a circular cylinder at high Reynolds numbers, Int. Journal of Heat and Fluid Flow. 24 (2003) 463-469

[5] M.E Youngs and A.Oii, Comparative Assessment of LES and URANS for flow over a cylinder at a Reynolds number of 3900, 16th Australasian Fluid Mechanics Conference Crown Plaza, Gold Coast, Australia 2-7 December 2007.

[6] Arthur G. Kravchenko and Parviz Moin, Numerical studies of flow over a circular cylinder at Re=3900, Phys. Fluids, Vol.12 (2000).

[7] Reiehl, P. Hourigan, K. and Thompson, M.C., Flow Past a Circular Cylinder Close to a Free Surface, J. Fluid Mech. 533 (2005) 269-296.

[8] S. Mittal and A. Raghuvanshi, Control of vortex shedding behind circular cylinder for flows at low Reynolds numbers, Int. J. Numer. Meth. Fluids. 35 (2001) 421-447

[9] C.H. Kuo, L.C. Chiou, C.C. Chen, Wake flow pattern modified by small control cylinders at low Reynolds number, Journal of Fluids and Structures, 23 (2007) 938-956.

[10] Badr,H.M., Coutanceau, M., Dennis, S.C.R., Menard,C. Unsteady flow past a rotating circular cylinder at Reynolds numbers 103 and 104. Journal of Fluid Mechanics, 220 (1990), 459–484.

[11] Kimura, T., Tsutahara, M., Wang, Z.Y. Wake of a rotating circular cylinder. A.I.A.A. Journal 30 (2) (1992) , 555–556.

[12] K. Srinivas, N. Fujisawa Effect of rotational oscillating upon fluids forces about a circular cylinder, J. of Wind Eng. and Industrial Aerodynamics 91 (2003) 637-652.

[13] Ongoren A., Rockwell D. Flow structure from oscillating cylinder. Part 1. Mechanisms of phase shift and recovery in the near wake. Journal of Fluid Mechanics (1988) 191, 197-223.

[14] Griffin, O.M., Ramberg, S.E Vortex shedding from a cylinder vibrating in line with an incident uniform flow. Journal of Fluid Mechanics (1976) 75, 257–271.

[15] Okajima, A., Takata, H., Asanuma, T. Viscous flow around a rotationally oscillating circular cylinder. Report of the Institute of Space and Aeronautical Science, University of Tokyo (1975), No. 532, pp. 311–338.

[16] Lam, K.M. Phase-locked education of vortex shedding in flow past an inclined flat plate. Physics of Fluids (1996), 8 (5), 1159–1168.

[17] Jose Luis Duarte Ribeiro, Effect of surface roughness on the two-dimensional flow past circular cylinders: mean forces and pressures, Journal of wind engineering and Industrial Aerodynamics, 37(1991) 299-309.

[18] W.C.H Shih, C. Wang, D. Coles, A. Roshko. Experiments on flow past rough circular cylinders at large Reynolds Numbers,Journal of wind engineering and Industrial Aerodynamics, 49(1993) 351-368.

[19] G.S. West, C.J. Apelt, The effects of tunnel blockage and aspect ratio on the mean flow past a circular cylinder with Reynolds numbers between 104 and 105, J. Fluid Mech. 114 (1982) 361±377.

[20] Cheol-Woo Park, Sang-Joon Lee, Free end effects on the near wake flow structure behind a finite circular cylinder, Journal of Wind Engineering and Industrial Aerodynamics 88 (2000) 231±246.

[21] S. Mittal and A. Raghuvanshi, Control of vortex shedding behind circular cylinder for flows at low Reynolds numbers, Int. J. Numer. Meth. Fluids. 35 (2001) 421-447

[22] Andrew M.R. Hopkins, 2010. Fluid Dynamics and Surface Pressure Fluctuations of Two-Dimensional Turbulent Boundary Layers Over Densely Distributed Surface Roughness. Doctor of Philosophy, Virginia Polytechnic Institute and State University.

[23] Stojkovic D, Breuer M and Durst F. Effect of high rotation rates on the laminar flow around a circular cylinder. Physics of Fluids, Vol. 14, No. 9 (2002), pp. 3160-3178.

[24] S.J. Karabelas*, B.C. Koumroglou, C.D. Argyropoulos, N.C. Markatos, High Reynolds number turbulent flow past a rotating cylinder, Applied Mathematical Modelling 36 (2012), 379-398.

[25] K. Srinivas, N. Fujisawa, Effect of rotational oscillation upon fluid forces about a circular cylinder, Journal of Wind Engineering and Industrial Aerodynamics 91 (2003) 637–652.

[26] N. Fujisawa, Y. Asano, C. Arakawa, T. Hashimoto, Computational and experimental study on flow around a rotationally oscillating circular cylinder in a uniform flow, Journal of Wind Engineering and Industrial Aerodynamics 93 (2005) 137–153.

[27] B.N. Rajani, A. Kandasamy, Sekhar Majumdar, (2009), Numerical simulation of laminar flow past a circular cylinder, Applied Mathematical Modelling 33 (2009) 1228-1247.

[28] Md. Mahbubar, Md. Mashud Karim and Md. Abdul Alim, (2007), Numerical Investigation of unsteady flow past a circular cylinder using 2D Finite Volume Method, J. of Naval Architecture and Marine Engineering (2007) 1813-8535.

[29] FLUENT 2006. FLUENT 6.3 Documentation.

[30] Willis D.,Israeli E., Persson P., Drela M., Peraire J., Swartz S.M. & Breuer K.S. A computational framework for fluid structure interaction in biologically inspired flapping flight. In 25th AIAA Applied Aerodynamics Conference, Miami Florida, June 2007. AIAA paper number 2007-3803.

[31] Vivek Krishnan, Kyle D. Squires, James R. Forsythe, Prediction of the flow around a circular cylinder at high Reynolds number, 44th AIAA Aerospace Science Meeting and Exhibit, 9-12 January 2006, Reno, Nevada.

[32] Martin, J. & Nail, K.Y. (2012) The effect of a Gust on the Flapping Wing performance. 50th AIAA Aerospace Science Meeting including the New Horizons Forum and Aerospace Exposition 09-12 January, Nashville, Tennessee, AIAA 2012-1080.

[33] Shun, Takahashi, Ichie Monjugawa, Kazuhiro Nakahashi (2008), Unsteady flow computation around moving multiple bodies using overset unstructured grids, Transactions of The Japan Society for Aeronautical and Space Science, Vol. 51, pp.78-85.

[34] Kumarasamy, S., Barlow, J.B., 1997. Computation of unsteady flow over a half-cylinder close to a moving wall. Journal of Wind Engineering and Industrial Aerodynamics 69-71, 239–248

[35] Kim, T.-Y., Lee, B.-S., Lee, D.-H., 2005. Study on the unsteady wakes past a square cylinder near a wall. Journal of Mechanical Science and Technology (KSME International Journal) 19 (5), 1169–1181.

★★★