Proceedings of the 4th International Conference on Civil Engineering for Sustainable Development (ICCESD 2018), 9~11 February 2018, KUET, Khulna, Bangladesh (ISBN-978-984-34-3502-6)

APPLICABILITY OF UNSTEADY RANS FOR PREDICTING FLOW FIELDS AROUND CYLINDERS

Md. N Haque

[1]*Assistant Professor ,Department of Civil Engineering, East West University, Bangladesh,*
e-mail: naimul@ewubd.edu

ABSTRACT

Accurate prediction of aerodynamic response mostly depends on the accurate modelling of turbulence. A number of turbulent modelling has been proposed over the years and each of them has its own merits and demerits. Unsteady RANS with k-ω-SST turbulence model is one of the popular simulation techniques which is computationally less expensive as compared to the others. In the present study, the performance of unsteady RANS is checked for other bluff bodies such as rectangular (R = width/depth =3) and circular cylinders at Reynolds number (R_e) of $1.2x10^4$. Simulations were conducted for a rectangular cylinder with a side ratio (R) of 3 and circular cylinders by using an open source code called OpenFOAM. The mean and rms values of steady state force coefficients were evaluated and compared with the previous experimental data. The mean pressure coefficients were also calculated at the bluff body surface. It was found that the unsteady RANS with k-ω-SST turbulence model can efficiently predict the aerodynamic responses around the selected bluff bodies.

Keywords: *Unsteady RANS; rectangular cylinder; circular cylinder, force coefficient and pressure coefficient etc.*

1. INTRODUCTION

Prediction of accurate wind load on structure is one of the most important engineering challenges. In conventional way, wind tunnel experiment is carried out to predict the wind load or aerodynamic responses of the structure. Wind tunnel is a reliable method of predicting wind loads on structure. However, wind tunnel is quite expensive and model setup procedure is also complicated. Moreover, through wind tunnel experiment one can obtain result in a limited location only. In contrary to wind tunnel experiment, Computation Fluid Dynamics (CFD) has overcome most of these limitations. Now-a-days, CFD has drawn attention of researchers of various fields and it is being used as a research tool. However, obtaining an accurate result through CFD simulation is not straightforward. To obtain an accurate result, users need to pay attention for a number of critical issues, such as turbulence model, numerical schemes, boundary condition, domain size, temporal and spatial discretization etc. Sufficient background studies required to be carried out for these issues to produce recommendations and proper guidelines regarding those issues.

Most of these issues were rigorously addressed in the literature (Murakami and Mochida 1989; Yu and kareem 1996, Rodi, Ferziger, Breuer, & Pourquie, 1997; Rodi 1997; Sohankar, Davidson and Norberg, 1998) and their influences on numerical results were investigated. However, still some issues need further analysis. For the selection of two-equation based turbulence model for two-dimensional simulation, it already known that *k-ω*-SST turbulence model has superiority than the other models. The applicability and performance of this model has also been checked in bluff body and bridge aerodynamics fields (Mirnada, Patruno, Ubertini & Vairo, 2014; Patruno, 2015 and Haque, Katsuchi, Yamada & Nishio, 2015). The flow behaviour around rectangular cylinders can be divided into three categories (Deniz and Staubli, 1997). The first category is the rectangular bluff section (Side ratio, R≤2) where the flow separates at the leading edge without any flow reattachment and vortices form at the leading edge (LEV). For the rectangular cylinder with side ratio (*R*) in between 2 to 5, the flow reattaches intermittently at the side face of the body and the impinging leading edge vortices form (ILEV). The third category is the elongated bluff section (*R*≥5) where the flow reattaches at the side surfaces and trailing edge vortices form (TEV). In previous studies the performance and applicability of *k-ω*-SST was mainly checked for predicting aerodynamic responses of rectangular bluff sections of first and third categories. Therefore, it is also important to check the performance and applicability of *k-ω*-SST turbulence model for the remaining (second category) category of semi-bluff sections (ILEV). Moreover, the effect of other issues such as boundary condition, domain size, temporal and spatial discretization etc. on results depends on the selection of turbulence model. If the turbulence model changes, their effects on numerical results also changes. In past studies the influence of those issues were mainly checked for the other turbulence models, especially for the LES. Later, Haque et al. (2015) investigated the effects of spatial discretization on results and proposed guidelines for selecting appropriate grid resolution for *k-ω*-SST turbulence model. No research was dedicated to investigate the effect of domain size or to provide some guidelines to select the domain size using the *k-ω*-SST turbulence model.

In the present study detail domain sensitivity analysis is carried out for a rectangular cylinder using *k-ω*-SST turbulence model. The influence of upstream, downstream and height of the domain is varied and their influences on aerodynamic force coefficients are presented. The performance and applicability of unsteady RANS with *k-ω*-SST turbulence model is also checked for the rectangular cylinder with side ratio (*R*) of 3 where partial or intermittent flow reattachment occurs by comparing the results with past experimental data. Along with this the performance and applicability of *k-ω*-SST turbulence model is also checked for a circular cylinder with smooth surface.

2. NUMERICAL MODEL AND SETUP

Flow is assumed to be two dimensional, unsteady and incompressible in nature. The flow around the object is modelled by Reynolds-Averaged Navier-Stokes (RANS) equation. The governing equations are shown as follows;

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

$$\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_j}{\partial x_j} = -\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j}(2\mu S_{ij} - \overline{\rho u'_j u'_i}) \tag{2}$$

The vectors U_i and x_i are velocity and position respectively, *t* is time, *P* is the pressure, *ρ* is the density, *μ* is the molecular viscosity and S_{ij} is the strain rate of tensor. Due to time averaging process, the new variable $\rho u'_j u'_i$ appears, which is known as Reynolds stress. It needs modelling to close the equation. Turbulence modelling is attained by *k-ω*-SST model (Menter, 1994). An open source Finite Volume code OpenFOAM (V2.2.0) is used to evaluate the flow filed numerically. The flow is discretized spatially by a structured non-uniform grid arrangement based on the recommendation given by Haque et al. (2015). Figure 1 shows the grid system. To confirm the stability during simulation, time step (Δ*t*) is selected such that courant number (C_o) doesn't exceed 0.8. All the simulations are conducted for 600 non-dimensional time unit and during response analysis first 200 non-dimensional times unit data are ignored from the stability point of view. A uniform flow (Along the flow, *u*=1, across the flow, *w*=0) is prescribed at the inlet as a boundary condition. At the outlet, Neumann boundary condition is applied. At the top and bottom wall of the domain, slip boundary condition is implemented. No-slip and no-penetration boundary conditions are prescribed on the solid walls around the object.

3. RESUTLS AND DISCUSSION

3.1 Domain Sensitivity Analysis

At the present work, a detail domain sensitivity analysis is conducted at high Reynolds number of 1.2×10^4 for side ratio (*R*) of 3 (B/D) with the *k*-*ω*-SST turbulence model. In domain sensitivity analysis, upstream length (X_u), downstream length (X_d) and height (*H*) of the domain are changed and global response of the body is observed. All the dimensions are normalized with the height (*D*) of the object. All three chosen lengths are varied from 5 to 40 when one parameter is being changed; other two parameters are kept constant at the basic value as shown in Figure 2. Figure 3 shows the mean and root mean square (rms) values of steady state force coefficients for normalized distance of interest. As can be seen from the figure that all the response parameters become stable when the upstream distance (X_u) exceeds a value of 15. But a value less than 10 affects the response significantly. Sohankar and his associates (Sohankar, Davidson & Norberg, 1995 and Sohankar, Norberg & Davidson, 1996) recommended a normalized upstream distance of 10 to get independent result, which is a bit liberal according to the present investigation. Similarly, for the case of height of the domain (*H*), when the value exceeds 25 units all the responses become stable and no correction for blockage is required. No significant effect of outlet location is found on the mean results except rms. A normalized downstream length of more than 25 units would be a recommended value to keep outlet disturbance away from after body when Neumann type outlet boundary condition is used.

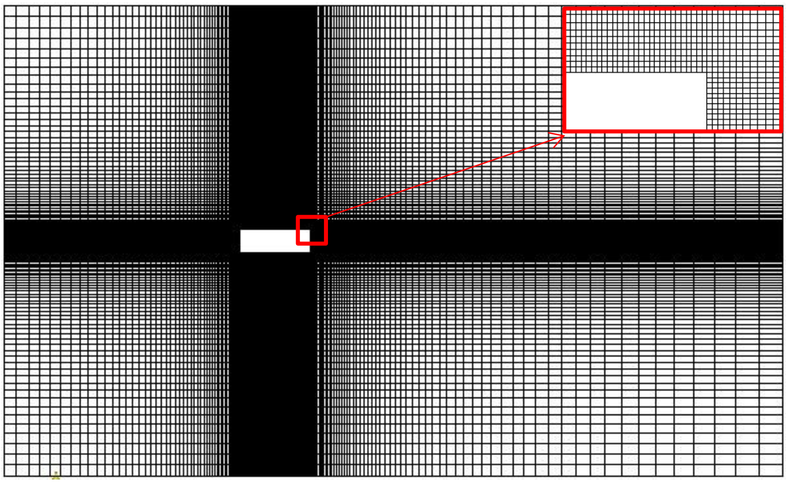


Figure 1: Non-uniform spatial discretization of the doamin

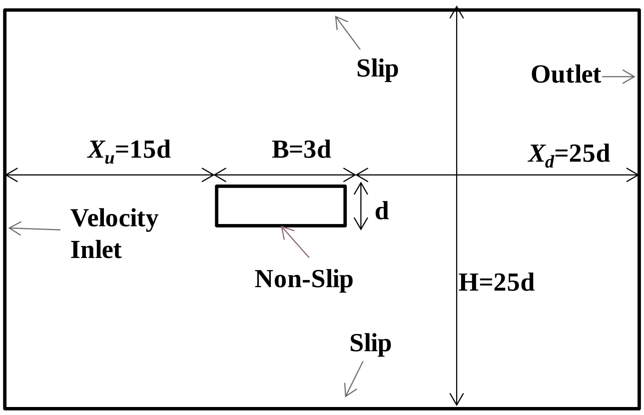
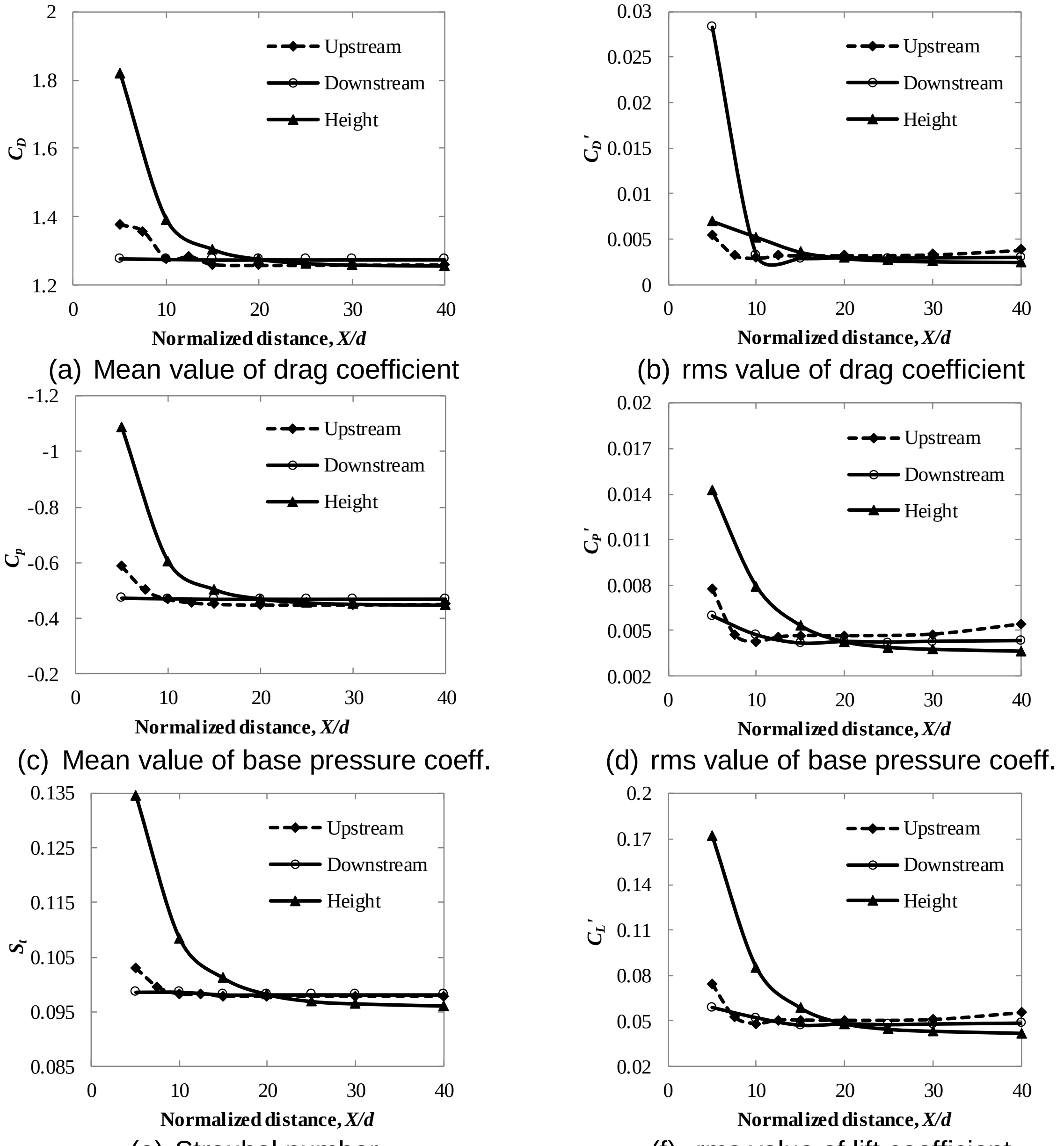


Figure 2: Boundary condition and domain size



(a) Mean value of drag coefficient

(b) rms value of drag coefficient

(c) Mean value of base pressure coeff.

(d) rms value of base pressure coeff.

(e) Strouhal number

(f) rms value of lift coefficient

Figure 3: Influence of domain size on steady state aerodynamic force coefficient

3.2 Aerodynamic Response of Rectangular Cylinder (*R* of 3)

To check the performance and applicability of unsteady RANS with *k*-*ω*-SST turbulence model for predicting aerodynamic response of semi-bluff section with ILEV, simulation is conducted for a rectangular cylinder of side ratio, *R*=3. Table 1 compares the calculated steady state force coefficients with the experimental data. As can be seen the calculated drag force coefficient is very close to the experimental one. Similar to the drag force, the present turbulence model could predict the Strouhal number (S_t) very close to the experimental one. For better comparison of the obtain results, Figure 4 compares the surface pressure coefficients with the experimental data. The *k*-*ω*-SST turbulence model grasps the surface pressure distribution quantitatively. Small discrepancy can be noticed at the top surface leading edge corner. However, except that the present turbulence could predict the aerodynamic responses quite efficiently.

Table 1: Aerodynamic characteristics of rectangular cylinder with a side ratio (*R*) of 3 at R_e=1.2x10^4

	C_d	C_L'	S_t
Current CFD	1.28	0.349	0.125
Exp. of Nakaguchi, Hashimoto & Muto(1968) at R_e=2-6x10^4	1.25		
Exp. of Norberg (1993) at R_e=4x10^4	1.24		0.155

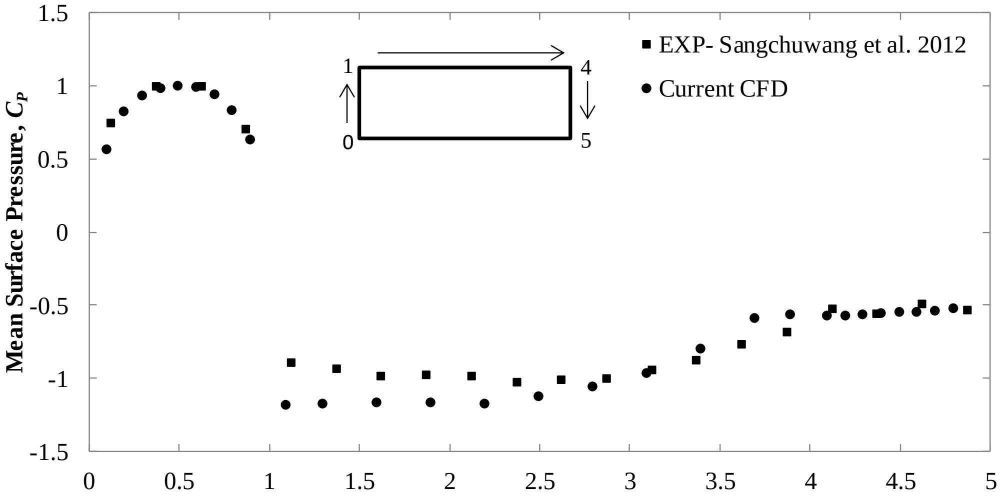


Figure 4: Surface pressure distribution around rectangular cylinder (*R* of 3) at R_e 1.2x10^4. Experimental work: Sangchuwang, Yamada & Katsuchi, 2012.

3.3 Aerodynamic Response of Circular Cylinder

The performance and applicability of is also checked for the circular at Reynolds number (R_e) of 1.2x10^4. Table 2 shows the calculated steady state force coefficients. The current simulation also could predict the aerodynamic responses with reasonable accuracy. Figure 5 compares the mean surface pressure around circular cylinder with previous experimental work. The *k-ω*-SST maps the mean pressure accurately but a slight discrepancy at the flow separation point can be found. One important reason could be limitation of the turbulence model itself and another reason could be the variation of Reynolds number between the present and experimental one.

Table 2: Aerodynamic characteristics of circular cylinder at R_e=1.0x10^4. Previous works: Cantwell and Coles (1983) at R_e=1.5x10^4, Wieselsberger (1921) at R_e=1.5x10^4, Ribner and Etkin (1958) at R_e=1.5x10^4, Mustto and Bodstein (2011) at R_e=4x10^4.

	C_D	C_D'	C_L	C_L'	S_t
Current CFD	1.48	0.078	0.00377	1.128	0.191
Exp. (Cantwell & Coles 1983)	1.18				
Exp. (Wieselsberger 1921)	1.16				
Exp. (Ribner & Etkin 1958)					0.191
Numerical (Mustto & Bodstein 2011)					0.177

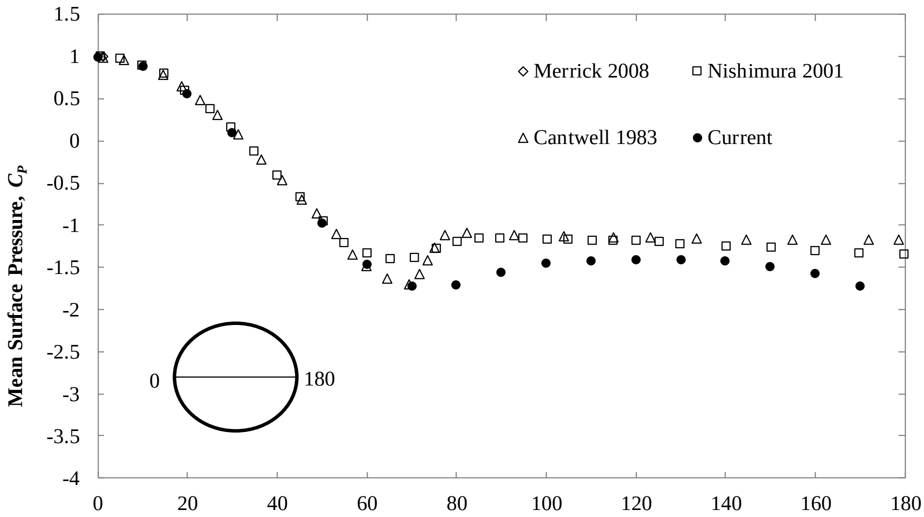


Figure 5: Surface pressure distribution around circular cylinder at R_e=1.2x10^4. Experimental results are: Merrick & Bitsuamlak (2008) at R_e=2.8x10^5; Nishimura & Taniike (2001) at R_e=6.1x10^4; Cantwell & Coles (1983) at R_e= 1.4x10^5

4. CONCLUSIONS

In the present paper, the performance and applicability of *k-ω*-SST turbulence is checked for simulating aerodynamic responses of semi-bluff section. Domain sensitivity analysis is carried out and simulations are performed for circular cylinders as well. Steady state force coefficients and surface pressure distribution are calculated and compared with the experimental results. Based on the observation and discussion, the following important conclusions are drawn,

i) For 2D RANS simulation with *k-ω*-SST turbulence model, inlet of the domain should be located at least 15D upstream of the object, outlet should be more than 20d downstream of the object and height of the domain should be at least 25D. A value less than 10D for upstream, 15D for downstream and 20D for height of the domain would alter the steady state responses significantly.

ii) It can be concluded that the Unsteady RANS with *k-ω*-SST turbulence model can predict the mean aerodynamic responses around the semi-bluff bodies having ILEVs both qualitatively and quantitatively. The Unsteady RANS failed to predict the responses accurately at the location of flow separation. Therefore, when this method is adopted to obtain the aerodynamic responses of bluff bodies with more complex shapes, especial care should be taken during the time of explaining the aerodynamic responses.

REFERENCES

Cantwell, B. and Coles, D. (1983). An experimental study of entrainment and transport in the turbulent near wake of a circular cylinder. Journal of Fluid Mechanics, 136, 321–374.

Deniz, S. and Staubli, T. (1997). Oscillating rectangular and octagonal profiles: interaction of leading- and trailing-edge vortex formation. Journal of Fluids and Structures, 11, 3-31.

Haque, M.N., Katsuchi, H. Yamada, H. and Nishio, M. (2015). Strategy to develop efficient grid system for flow analysis around two-dimensional bluff bodies. *KSCE Journal of Civil Engineering*, 20(05), 1-12.

Merrick, R. and Bitsuamlak, G.T. (2008). "Control of flow around a circular cylinder by the use of surface roughness." 4th International Conference on Advances on Wind and Structures (AWAS08), 29-31 May 2008, Jeju, Korea.

Miranda, S.D., Patruno, L., Ubertini, F. and Vairo, G. 2014. On the identifications of the flutter derivatives of bridge deck via RANS turbulence models: Benchmarking on rectangular prisms. *Engineering Structures*, 76, 359-370.

Murakami, S. and Mochida, A. (1995). On turbulent vortex shedding flow past 2D square cylinder predicted by CFD. *Journal of Wind Engineering and Industrial Aerodynamics,* 54-55,191-211.

Mustto, A.A. and Bodstein, G.C.R. (2011). Sub-grid scale modeling of turbulent flow around circular by mesh-free vortex method. *Engineering Application of Computational Fluid Mechanics*, 5(2), 259-275.

Nakaguchi, H., Hashimoto, K., Muto, S., (1968). An experimental study on aerodynamic drag of rectangular cylinders. *J. Japan Soc. Aeronaut. Space Sci*. 16,1-5 (in Japanese).

Nishimura, H. and Taniike, Y. (2001). Aerodynamic characteristics of fluctuating forces on a circular cylinder. *Journal of Wind Engineering Industrial Aerodynamics*, 89,713-701.

Norberg, C. (1993). Flow around rectangular cylinders: pressure forces and wake frequencies. Journal of Wind Engineering and Industrial Aerodynamics,. 49,187-196.

Patruno, L. (2015). Accuracy of numerically evaluated flutter derivatives of bridge deck sections using RANS: Effects on the flutter onset velocity, *Engineering Structures*, 89, 49-65.

Ribner, H.S. and Etkin, B. (1958). "Noise research in Canada." Proc. 1st International Congr. Aero. Sci., Madrid, Pagamon press, London,1958.

Rodi, W., Ferziger, J., Breuer, M., and Pourquie´, M. (1997). Status of Large- Eddy Simulations: Results of a Workshop, *ASME Journal of Fluids Engineering*, 119, 248–262.

Sangchuwong, P., Yamada, H. and Katsuchi, H. (2012). Study on turbulence effects on flow fields around sharp-edged bluff bodies. *Journal Strucural Engineering (JSCE),* 59A, 627-636.

Sohankar, A., Davidson, L., Norberg, C. (1995). "Numerical simulation of unsteady flow around a square two-dimensional cylinder." Proc. Twelfth Australasian Fluid Mechanics Conf. 10-15 December, 1995, Sydney, Australia, 517-520.

Sohankar, A., Norberg, C., Davidson, L. (1996). "Numerical simulation of unsteady low-Reynolds number flow around a rectangular cylinder at incidence." Proc. 3rd Int. Colloq. on Bluff Body Aerodynamics and Applications, July 28- 1 August 1996, Virginia, USA.

Sohankar, A., Davidson, L., and Norberg, C. (1998). Low- Reynolds- number flow around a square cylinder at incidence: study of blockage, onset of vortex shedding and outlet boundary condition. *International Journal for Numerical Methods in Fluids*, 26, 39-56.

Wieselsberger, C. (1921). Neuere feststellungen .uber die gesetze des fl. ussigkeits und luftwiderstands, *Phys. Z*. 22, 321–328.

Yu, D., Kareem, A., (1996). Two-dimensional simulation of flow around rectangular prisms. *Journal of Wind Engineering and Industrial Aerodynamics,* 62, 131-161.