FLOW FIELD SIMULATION AROUND ELONGATED RECTANGULAR CYLINDER: EFFECT OF NEAR WALL TREATMENT

Yokohama National University Student Member ○Md. Naimul HAQUE
Yokohama National University Regular Member Hiroshi KATSUCHI
Yokohama National University Fellow Member Hitoshi YAMADA
Yokohama National University Regular Member Mayuko NISHIO

1. INTRODUCTION

Computational Fluid Dynamics has now become a strong tool in wind engineering to simulate flow around engineering structure due to improvement of both turbulence theory and computer capacity. In different practical applications CFD are now being used successfully along with experiment. But to be an independent tools of research, still CFD need to pass through bottleneck, like handling turbulent flow. Flow field around bluff bodies are very complex at high Reynolds Number (R_e) and accurate prediction is required both for practical application and research purpose. Accurate prediction of response and flow behavior around bluff body is highly influenced by the presence of wall, where viscosity affected zone has large gradient in the solution variables. There should be sufficient grid to resolve the flow near wall or have to use wall function with moderate mesh. To deal with dilemma, non-dimensional wall distance (y^+) is a good strategy to select appropriate near-wall treatment (Gerasimov, 2006). Also there are no significant work has been found in the literature that tried to elucidate the near-wall treatment effect on elongated cylinder, where both flow separation and reattachment occurs. In this paper it is aimed to simulate the flow behavior around rectangular cylinder of side ratio R=3, under high Reynolds number R_e=4x10^4 with different values of y^+. Altering the mesh resolution near the object, wall y^+ value also was altered varying from 4 to 120. Three different turbulence model, k-e model, k-w model and k-ω-SST model are employed. Results are compared with the past experimental work in terms of drag force coefficient (C_D), lift force coefficient (C_L) and strouhal number (S_t). Also surface pressures around the body under all these conditions are plotted. It is tried to explain the near-wall treatment effect on response of bluff body and propose some guideline based on wall y+ value for the accurate prediction of response and the corresponding turbulence model.

2. NUMERICAL DETAILS

The flow around the object is modeled by Reynolds-Averaged Navier-Stokes (RANS) equation and OpenFOAM v2.2.0 is used as a solver. 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 density, μ is molecular viscosity, S_{ij} is strain rate tensor. Due to time averaging process, the new variable appears $\overline{\rho u'_j u'_i}$, is known as Reynolds stress and need modeling to close the equation. Turbulence modeling is attained by k-ω-SST model. In the present work only the result obtained from k-ω-SST model is presented. The computations are performed in a two dimensional domain with a dimension of 43DX25D, where D is the height of the domain. The object is placed 15D downstream from the inlet. At the outlet pressure boundary condition, at the top and bottom of the domain slip boundary condition and at the body non-slip boundary condition is implemented.

3. NEAR WALL TREATMENT APPROACH

Near the wall, the variables, momentum and other scalar transport changes vigorously and the viscosity affected region can be divided into three zones are, Viscous sub-layer (y^+<5); Buffer zone (5<y^+<30) and fully turbulent or log-law region (30<y^+). y^+ is a non-dimensional distance similar to local Reynolds number and used to present the quality of mesh depending on the flow. Accurate evaluation of flow around the wall mounted object depends on the accurate prediction of turbulent flow near the wall. y^+ value of larger than 30 with standard wall function or y+ value of 1-4 to resolve the turbulent flow (near-wall treatment) can be adopted for accurate prediction of flow (Gerasimov, 2006). In the present work wide range of y^+ value (2 to 120) with or without wall function is considered by varying the first cell mesh height away from the wall. In all the simulations, mesh is stretched with a factor of 1.05 in all the direction. For the initial assumption of y^+ value the equation of flat plate is used. The equation is given below,

Keywords: CFD, y^+, Reynolds number, Drag force coefficient, Strouhal number.
Contact address: Structural Eng. Lab., Dept. of Civil Eng., Yokohama National University, Yokohama 240-8501, Japan.

$$y = Ly^{+}\sqrt{74}R_{L}^{-13/14} \tag{3}$$

Where, y is the first cell height, L is characteristics length, R_L is Reynolds number and y^+ is non-dimensional number.

5. RESULTS AND DISCUSSION

In this paper, effects of near wall treatment is presented for a rectangular of R=3 under Reynolds number of 4×10^4. Table 1 shows the y^+ value obtained after running simulation with the mesh generated by approximating from eq. 3. The average value is pretty close to the approximated one. On the front side of the body experienced higher value, whereas the back surface lower. As the mesh becomes finer the computation becomes expensive, due to both in terms of cell number and finer time step to make the simulation stable. For y^+ 120, 60 and 35, standard wall function is utilized and for 15 and 4 no wall function is utilized as the flow will be resolved directly. As can be seen, near wall treatment doesn't affect drag force coefficients (C_d) seriously but influences the strouhal number directly. Figure 1 shows the pressure coefficient (C_p) plot around the object for various y^+ values. Under all values of y^+, it grasps the pressure coefficients perfectly at the front surface but there are some discrepancy at the side and back surface. But the first cell with y^+=4 predicts the most accurate pressure distribution around the object.

Table 1. Comparison of results obtained from mesh with different y+ values and its expense for running 20 sec.

Sl	y^+	y^+ Obtained			Expense		Accuracy		
No	Calculated	Min	Max	Average	Time (Days)	Processors	Est. C_d	Est. S_t	Exp.
1	120	35	349	127	0.15	1	1.24	0.094	C_d=1.25[1]
2	60	5	32	22	0.4	1	1.23	0.087	
3	35	38	2.5	14.5	1	1	1.24	0.037	S_t=0.15[2]
4	15	0.49	6.5	3.5	13	2	1.23	0.1	
5	4	0.02	2.5	1.25	20	8	1.28	0.125	

[1]Nakaguchi et al., 1968; [2]Norberg 1993.

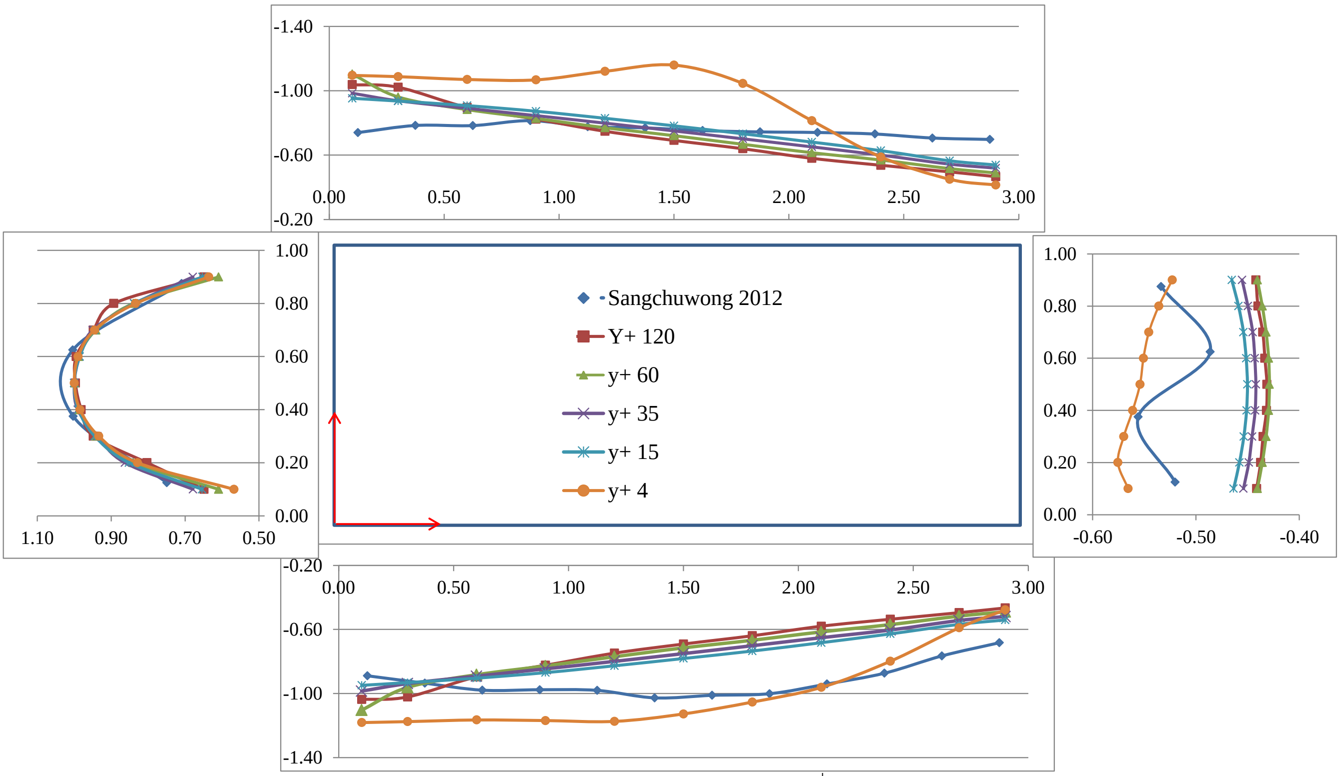

Figure 1. Pressure Coefficient (C_p) plot around the cylinder for various y^+ value.

REFERENCES

Gerasimov, A., 2006. Modelling Turbulent flows with fluent. Europe, ANSYS Inc.

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).

Norberg, C., 1993. Flow around rectangular cylinders: pressure forces and wake frequencies. *J. Wind Eng. Ind. Aerodyn.* 49, 187-196.

Sangchuwong, P., Katsuchi, H. and Yamada, H., 2012. Study on turbulence effects on flow patterns around rectangular cylinders. The 7th International Colloquium on Bluff Body Aerodynamics and Applications (BBAA7) Shanghai, China, September 2-6, 2012.