NUMERICAL SIMULATION ON AERODYNAMICS OF 2D RECTANGULAR CYLINDER UNDER DIFFERENT ANISOTROPIC TURBULENCES

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

Turbulence of natural wind is highly anisotropic at the atmospheric boundary layer and continuously interacting with our structure. It is reported by different researchers that the level of anisotropy or the variation of properties along different directions, is also highly variable depending on the type of the flow (Xiao et al, 2009). On the other hand, bluff body of rectangular shape is a very common configuration found in tall building, bridge and towers. Being very important its flow behavior has been widely investigated under both smooth and turbulent flow by different past researchers. Mainly wind tunnel experiment was used to investigate the turbulent flow effect on bluff body. However, in the wind tunnel the level of anisotropy is almost unique. At the present time, CFD has become a very strong tool to investigate flow behavior, where one can control the inlet data at a desire level. The main objective of the present study is to investigate the effect of different level of anisotropy of turbulences on the response of 2D rectangular cylinder. Along wind turbulence is kept constant for all cases and vertical wind turbulence is changed. Four different anisotropic turbulent flows are considered in the present work. Side ratio, R-0.1875, 0.375, 0.625, 0.8 and 1 of rectangular cylinder are taken into consideration to observe vertical wind turbulence effect on flow pattern. Large eddy simulation with smagornisky subgrid scale model is performed for all the cases. Results were mainly discussed in terms of drag and base pressure coefficient.

2. MODELING, MESH AND BOUNDARY CONDITION

Commercial software starccm+ (version- 5.04) is used to conduct the simulation. Large eddy simulation with smagornisky subgrid scale model is employed for all the computations. The governing equations used in LES model are obtained by filtering the time-dependent Navier-stokes equations as follows;

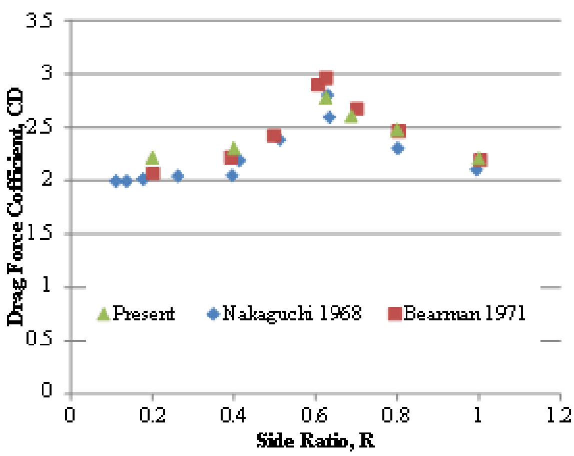
$$\frac{\partial \rho \bar{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \bar{u}_i) + \frac{\partial}{\partial x_j}(\rho \bar{u}_i \bar{u}_j) = \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \bar{u}_i}{\partial x_j}\right) - \frac{\partial \bar{p}}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

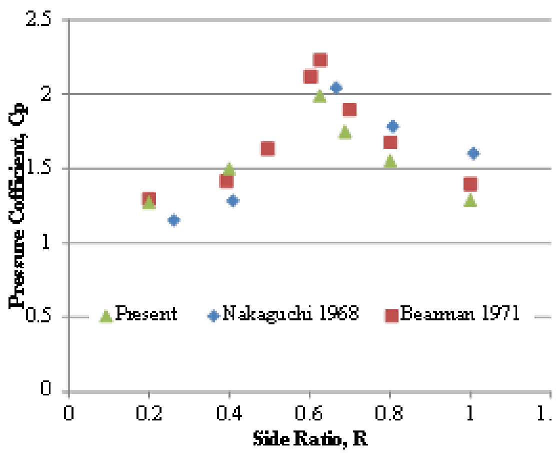
Three dimensional flow domain is considered with dimension 20Dx14DX2D, where D is the height of the object. Three different types of mesh size are used to discretize the flow domain with the smallest one around the object where 16 grid points per D is maintained. For the top and bottom wall slip boundary condition and in the span wise direction periodic boundary is incurred. Convective boundary condition is chosen for the outlet.

3. VERIFICATION OF THE MODEL

To verify the model, first simulation is conducted against smooth flow with Reynolds number, Re=2.2x10^4. Figure 1 (a)



(a)Drag Force Coefficient, CD



(b)Base Pressure Coefficient, Cp

Fig. 1 Drag and base pressure coefficient for smooth flow.

Keywords: Anisotropic turbulence, Side ratio, Large eddy simulation, Drag force coefficient, Base pressure coefficient.
Contact address: Structural Eng. Lab., Dept. of Civil Eng., Yokohama National University, Yokohama 240-8501, Japan.

and (b) shows the plot of drag and base pressure coefficient with varying side ratio. It can be seen that there is very good compatibility with the past research and can be used to simulate turbulent flow.

4. INLET TURBULENCE

Wind tunnel time domain turbulent data is produced at our wind tunnel laboratory. Four set of turbulent data with different level of anisotropy is produced by combining previously produced turbulent flow. Table 1 shows the properties of the anisotropic turbulence. To define turbulent flow, reduced turbulence intensity, I_r (Katsuchi and Yamada 2010) is used and shown in eq. 3. All these four set anisotropic turbulence is incurred at the inlet of the flow domain.

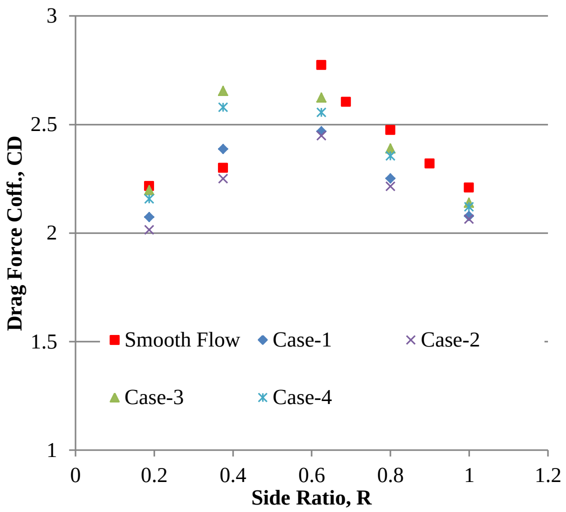
$$I_r = \frac{I_u}{\left(\frac{L_u}{h}\right)^{1/5}} \tag{3}$$

Table 1 Properties of the considered anisotropic turbulences.

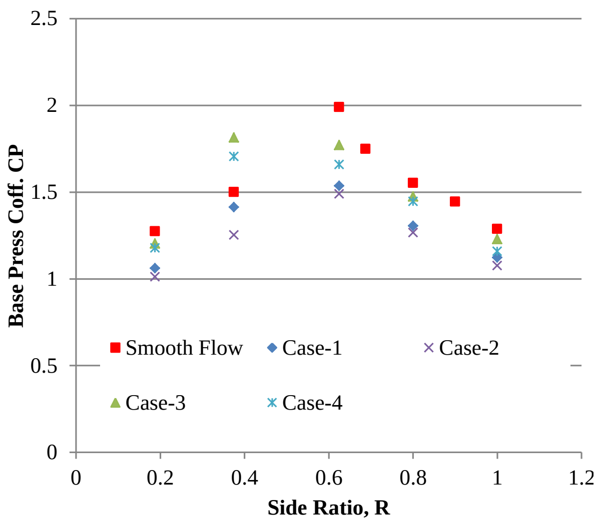
Sl. No.	Along wind Turbulence		Properties of Anisotropy	Vertical Wind Turbulence
	Properties	I_{ru}		Properties
1	L_u=13.97; I_u=9%	0.081	$L_u:L_w$ = 1:0.15; $I_u:I_w$ = 1: 0.2	L_w=2.11; I_w=1.8%
2	L_u=11.56; I_u=8.5%	0.081	$L_u:L_w$ = 1:0.35; $I_u:I_w$ = 1: 0.5	L_w=4.48; I_w=4.25%
3	L_u=8.39; I_u=7.6%	0.081	$L_u:L_w$ = 1:0.55; $I_u:I_w$ = 1: 0.8	L_w=4.52; I_w=6.08%
4	Lu=13.97; I_u=9%	0.081	$L_u:L_w$ = 1:0.15; $I_u:I_w$ = 1: 0.6	L_w=2.11; I_w=5.4%

5. RESULT AND DISCUSSION

Effect of anisotropic turbulence is investigated in this paper, changing the vertical wind turbulence for varying side ratio. Fig. 2 shows the drag and base pressure coefficient for R= 0.1875 to 1, under four different cases of anisotropic turbulences. As can be seen, from Fig. 2 that the vertical wind turbulence has significant effect on the response of bluff body. Specially for the case 3 where the responses of the longer side ratio objects are very close to the response of the smooth flow and for R=0.375, the response of turbulent flow exceed the smooth flow response. Moreover, the peak responses shift toward smaller side ratio and same phenomenon has been confirmed by past researchers for along wind turbulence also. However, the reduced turbulence intensity of along wind turbulence is only 0.081for all the cases, higher value of reduced turbulence intensity may have higher response. Details investigation should be explored, changing the value of reduced turbulence intensity of along wind turbulence.



(a)Drag Force Coefficient, CD



(b)Base Pressure Coefficient, Cp

Fig. 2 Drag and base pressure coefficient under different anisotropic turbulences

REFERENCES

Bearman, P.W., Trueman, D.M., 1971. An investigation of the flow around rectangular cylinders. Aeronaut. Q. 23, 229-237..

Katsuchi, H., Yamada, H., 2010. Study on turbulence partial simulation for wind-tunnel testing of bridge deck. 13th International Conference on Wind Engineering, July 10-15, 2011, Amsterdam.

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).

Xiao, Y.Q., Li, L.X. and Song, L.L., 2009. Study on typhoon wind characteristics based on field measurements. The Seventh Asia-Pacific Conference on Wind Engineering, November 8-12, 2009, Taipei, Taiwan.