

Wind flow around rectangular obstacles and the effects of aspect ratio

Dan Gu^a, Hee-Chang Lim^b

^a*Graduated student of school of mechanical engineering, Pusan Nat'l Univ., san 30, Jangjeon-dong, Geumjeong-gu, Pusan, South Korea*

^b*Assistant professor of school of mechanical engineering, Pusan Nat'l Univ., san 30, Jangjeon-dong, Geumjeong-gu, Pusan, South Korea*

ABSTRACT: It has long been studied about the flow around bluff bodies, but the effect of aspect ratio on the sharp-edged bodies in a thick turbulent boundary layer is still argued. The study is aiming to understand the surface pressure distribution around the bodies such as the suction pressure in the leading edge on the top surface when the aspect ratio of bodies is changed. Therefore, the authors firstly carried out the wind tunnel measurement around a series of rectangular bodies ($40^d \times 80^w \times 80^h$, $80^d \times 80^w \times 80^h$, $160^d \times 80^w \times 80^h$, $80^d \times 40^w \times 80^h$ and $80^d \times 160^w \times 80^h$ in mm) placed in a deep turbulent boundary layer. With modern numerical calculations, the RANS based on the k- ϵ model and the DES turbulence model were used, and the numerical results were compared with wind-tunnel experiments. The results show that the transverse width has a substantial effect on the surface pressure around the bodies, while the longitudinal length has a little influence on the surface pressure.

KEYWORDS: Rectangular bodies, Wind environment, Aspect ratio, Surface pressure distribution, Wind-tunnel measurement, CFD

1 INTRODUCTION

The flow characteristics on a bluff body immersed in a turbulent flow has long been of fundamental interest in the study of fluid dynamics. Such investigations are critical in the design and development of practical objects such as windmills, buildings and bridges, etc. Above all, the study of flow characteristics around a bluff body is generally considered to be important in academic circles as well as in the engineering applications and now there are still lots of topics left to be studied.

Regarding to the flow around buildings or all kinds, there have been numerous empirical data and comparisons between wind-tunnel and full-scale (field) data. One of the well-cited papers in this area is the wind-tunnel experiments of Castro & Robins (1977)^[1] (hereafter denoted by CR), which measured the flow around the surface-mounted cubes. In addition, Tieleman & Atkins (1996)^[2] reported that the variation of the base/side surface pressure of surface-mounted rectangular prisms was determined by the interaction of the incident turbulence with the separated shear layers. More recently, Cigada et al. (2006)^[3] investigated the fluid-dynamic forces acting on a rectangular cylinder for unbounded flow conditions. Especially, they focused on the effects of a fixed wall placed at various distances from a rectangular cylinder and observed that the force component acting on the cylinder depends on the periodical motion and provides the dynamic characterization of the loading and of the wake shedding. In addition, the aspect ratio of the body governs effects of the wall condition on the force coefficients and the Strouhal number. Then, Larose & Auteuil (2008)^[4] made a wind tunnel measurement on rectangular prisms with aspect ratio of 2, 3 and 4 at high Reynolds number.

With regard to the CFD techniques, there have been a substantial change in the numerical modeling as well as the hardware development and it still develops rapidly. There have been a lot of attempt for simulating the precise flow around a various bodies by solving the governing Navier–Stokes equations. Especially, in the early stage, there were a lot of turbulence models to solve the complicate turbulent flow around a body and some of them were compared each other for the systematic efficiency of the models (i.e. Murakami (1993)^[5]; Zhang et al. (1996)^[6]; Meroney et al. (1999)^[7]). What they found is that the accuracy of the numerical calculation was highly dependent of the choice of the turbulence model. However, most previous studies have been focused on the typical Reynolds-Averaged Navier–Stokes (RANS) method, especially the standard $k-\epsilon$ model (see e.g. Iaccarino et al. (2003)^[8]; Li and Stathopoulos (1997)^[9]; Tominaga and Stathopoulos (2009)^[10]) was one of kinds. However, it was reported that the RANS simulation was unsuccessful as usual. In contrast to RANS method, rather the large-eddy simulation (LES) resolved large-scale unsteady motions and it showed a good agreement with wind tunnel experiments (i.e. Krajnovic' and Davidson (1999)^[11], (2001)^[12]; Shah and Ferziger (1997)^[13]), whereas the LES simulation was a bit costly and spent much more time and had a high request for the CPU time. Therefore, in a very recent time, there has been a trend to use an eclectic approach to balance the computational costs and the precision of calculation, namely Detached Eddy Simulation (DES) method. Due to the limited number of papers, Jochen & Dominic (2008)^[14] was one of them describing the usefulness and high potential of the LES/RANS turbulence models for using a practical application.

The immediate emphasis in this paper is on the flow characteristics around the rectangular bodies with various aspect ratio. The aspect ratio is responsible for the type of generated wake and, ultimately, for the structural loading and pressure and especially structure excitation. For example, the drag coefficient of an elongated rectangular-section body in a flow is a function of the width of the body. (see Fig.1) As shown in the figure, the drag is a function mainly of the elongation d/w of the body. Therefore, the paper consists of a carefully designed set of experiments and numerical simulation on a simulated turbulent boundary layer flow, which is tailored in the atmospheric environment condition, over a surface-mounted various rectangular obstacles. In the numerical calculation, the standard $k-\epsilon$ and the DES model were used and finally compared with the experimental data (i.e., the wind tunnel and the field study).

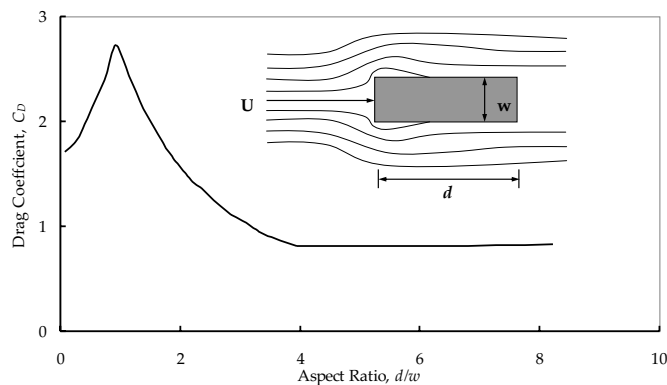


Figure 1. Drag coefficient versus aspect ratio of rectangular body. (Simiu and Scanlan, 1996^[15]).

The following section §2 outlines the experimental techniques, §3 describes the computational techniques, §4 summarizes the surface pressure characteristics, as well as some findings and discussions about the major results around the models under a simulated turbulent boundary layer, finally §5 gives the major conclusions.

2 EXPERIMENTAL TECHNIQUES

2.1 Atmospheric boundary layer wind tunnel

Figure 2 illustrates the set-up, showing the model location in the wind tunnel. Experiments were conducted in the middle closed-circuit subsonic wind tunnel, whose working section dimension is $0.6\text{m}^{\text{wide}} \times 0.72\text{m}^{\text{high}} \times 6\text{m}^{\text{long}}$ having a maximum wind speed of about 40m/s at the Pohang University of Science and Technology in South Korea. It is suitable for generating an artificial boundary layer and is also equipped with modern hot-wire anemometry (IFA100) and a PIV system for optical measurements of the airflow. The rectangular models used in the study are made of plexiglass, and they consist of 3 bodies - a cube ($80^{\text{d}} \times 80^{\text{w}} \times 80^{\text{h}}$ in mm for 1×1) for comparing the existing results in a reference, two rectangular bodies ($40^{\text{d}} \times 80^{\text{w}} \times 80^{\text{h}}$ for 2×1 s and $160^{\text{d}} \times 80^{\text{w}} \times 80^{\text{h}}$ for 1×2), and especially the flow around rectangular bodies could make two more aspect-ratios rotating it 90° (e.g., $80^{\text{d}} \times 40^{\text{w}} \times 80^{\text{h}}$ for 1×2 s and $80^{\text{d}} \times 160^{\text{w}} \times 80^{\text{h}}$ for 2×1).

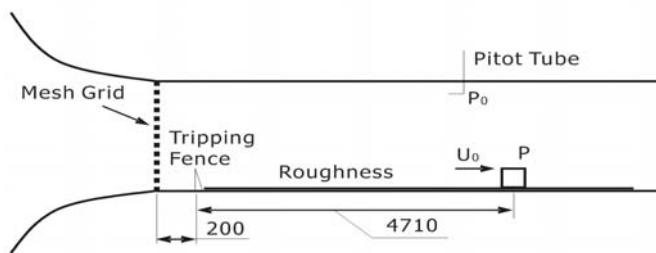


Figure 2. The $0.6\text{m} \times 0.72\text{m} \times 6\text{m}$ wind tunnel test section and model set up.

2.2 Simulated atmospheric boundary layer

Thick boundary layers were generated using a technique often employed by wind-engineering practitioners, first devised by Cook (1978)^[16]. Toothed barriers spanning the floor of the working section near its entry, followed by a square section, bi-planar mesh across the entire working section and an appropriate rough surface thereafter can together be designed to yield mean-velocity profiles which are closely logarithmic over a significant portion of the working-section height, with turbulence stresses and spectra similar to those found in atmospheric neutrally stable boundary layers. Since it was intended to make comparisons with the existing data such as wind tunnel and field data (e.g. a 6m height cube in Silsoe, UK^[17]) - less than one-tenth of the height of the logarithmic region - maximizing the depth of this region was deemed the most important. It is crucial to design the barrier wall and mixing grid geometries in tandem with the intended roughness, since any mismatch will yield unacceptably long fetches before reasonably well-developed flows are attained. In the present case, commercially available artificial plastic artificial grass was used to provide the surface roughness. This gave a roughness length, z_0 , of 0.17mm, where z_0 is defined in the usual way via the mean velocity log law expressed as:

$$\frac{U}{u_*} = \frac{1}{K} \ln \left(\frac{z-d}{z_0} \right) \quad (1)$$

where u^* = the friction velocity; d = 'zero plane displacement'.

Obtaining the three unknowns (u^* , d and z_0) from the mean velocity profile alone is, as is well-known, a very ill-conditioned process. In the present study, u^* was deduced from an extrapolation of the measured turbulence shear stress to the surface (see Lim et al., 2007^[18]), with d and z_0 then following from a best fit of the mean velocity data to eq.(1). In the tunnel, the barrier wall had a height of 50mm, with triangular cut-outs at the top, of pitch 50mm and depth 50mm, and the mixing grid consisted of a biplanar grid of 10mm bars at a pitch of 50mm.

3.2 Turbulence models

The present numerical simulation has been carried out according to the experiment study. The Reynolds number based on h (model height) and U_h (mean inlet velocity at h) was 4.6×10^4 . The inlet mean-velocity profiles were compiled by UDFs, which are shown in Fig.5 (a) compared with the experiment profiles. As shown in the vertical wind profiles, it can be seen that a fully developed shear flow was made in the wind tunnel as well as numerical tunnel, which were specifically designed to be similar to the (rural) atmospheric boundary layer. Two approaching methods (i.e., Reynolds-averaging (RANS model) and filtering methods (LES model)) were employed to render the Navier-Stokes equations tractable so that the small-scale turbulent fluctuations do not have to be directly simulated. Firstly, what we consider in this study is the standard $k-\epsilon$ model. Here, the self-developed code of the turbulence kinetic energy (k) and its dissipation rate (ϵ) was used to obtain the turbulence intensity, which should be consistent with the experiment, see Fig.5 (b). As the LES simulation takes usually a high computational cost, herein, we select the DES model which is often referred to as the hybrid LES/RANS models. In this case, the DES model is based on the one-equation Spalart-Allmaras model and the transient calculations were carried out with the time step 0.0001s.

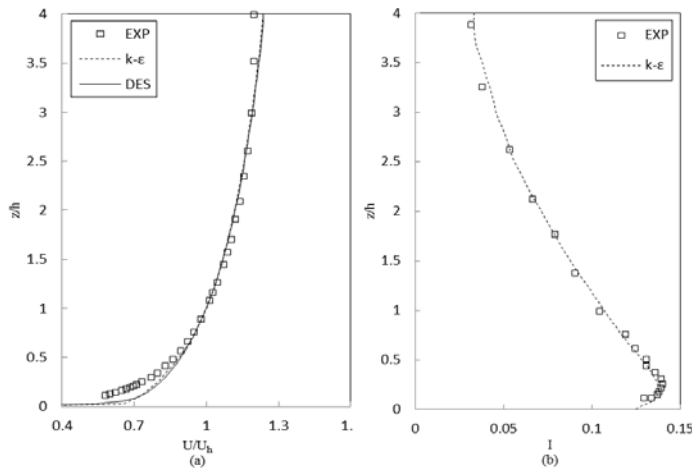


Figure 5. Mean velocity (a) and turbulent intensity (b) profiles.

4 RESULTS AND DISCUSSION

4.1 Surface pressure distribution - a cubical model (1×1)

Firstly, the surface pressure distributions along the centerline around the 1×1 model are shown in Fig.6. The variation of the mean static pressure coefficient $C_p = (p - p_r) / (0.5 \rho U_h^2)$, where p_r is the mean static pressure in the upstream flow) is compared between numerical and experimental results. In addition, the figure also includes the existing results - wind tunnel (WT) and full field scale (FS) measurement, which is as a function of the measurement location x/h ; $x=0$ corresponds to the foot at the front face of the model body - actually depending on the wind direction (e.g., see the solid line on the right figure). Note that the rest of the subsequent figures are arranged in a similar manner to this figure. It is no doubt that the profiles in the figure has an expected shape, in that the largest negative pressures occur just beyond separation at the leading edge and are followed by a substantial pressure recovery on the top surface, as shown frequently by previous studies. Note that the experiment data agree well with the earlier field data of LCH's (Lim, Castro & Hoxey, 2007^[19]) but are significantly different from the wind-tunnel data of CR (Castro &

Robins, 1977). The latter are similar to those of Murakami & Mochida (1988)^[20] and, in agreement with CR's discussion, are undoubtedly a result of very much higher upstream turbulence levels, leading to much earlier attachment and pressure recovery on the top surface. In addition, the k- ϵ model data are nearly similar except the region on the just right corner of the position $x/h=1$ which has a negative peak. This seems to be caused by the turbulence model itself so that in this regard, the DES model seems to have better performance than the k- ϵ model when simulating the flow around a sharp-edged bluff body.

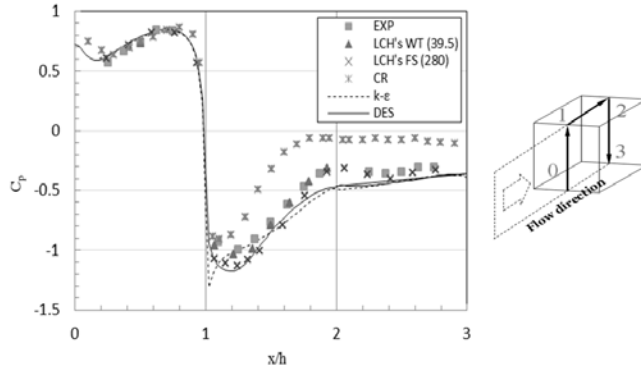


Figure 6. Mean surface static pressure coefficient along the central section with wind normal to face.

Figure 7 shows the mean surface pressure along the mid-height of the cube. As shown in the figure, the experiment data agree well with the field measurement data except the CR's result which has been explained before. The k- ϵ model data are similar to the experiment data, but still has a significant difference on the just right corner of the position $x/h=1$, whereas the DES model data seem to improve the accuracy of the prediction in this region - a negative peak. However, the numerical data show relatively low value compared to the other experiment data. Perhaps, this can be explained that the upstream turbulence level in the approaching flow is not high enough so that the attachment and pressure recovering seem to be delayed.

Even though the current experiment and numerical data in this section are showing the simple comparison with the existing results, the immediate implication is that the comparison with the existing results confirms the high accuracy and reliability of the current study, and the identical condition with the field measurement yields the significant reduction of the measurement and calculation errors. Therefore, the current section could be a precursor of the rest of the pressure profiles and the next section will discuss the pressure profiles around the rectangular obstacles with the same boundary layers.

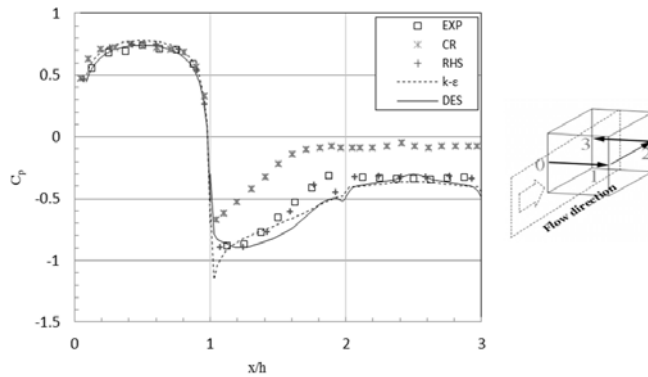


Figure 7. Mean surface static pressure coefficient at the mid-height of the cube.

4.2 Surface pressure distribution - models with aspect ratio

Mean surface static pressure profiles along the centerline of the models with the three different aspect ratios are plotted in Fig.8. The figure shows how the surface pressure on the top as well as on the front and rear face changes with changing of the width of the body. Note that several subsequent figures are arranged in a similar manner to Figure 8, i.e. the experiment pressure profiles are shown on the upper-left and the CFD ($k-\epsilon$ model) pressure profiles are shown on the upper-right. In addition, models to compare each other are shown underneath, to easily visualize the measurement locations around the body. The immediate implication of the data in Fig.8 is that the width variation while maintaining the depth makes the surface pressure on the top surface more negative, whereas the front and rear face are almost consistent for the width variation. In addition, as it can be seen, the solid arrow shows the direction of the pressure drop as the width changes.

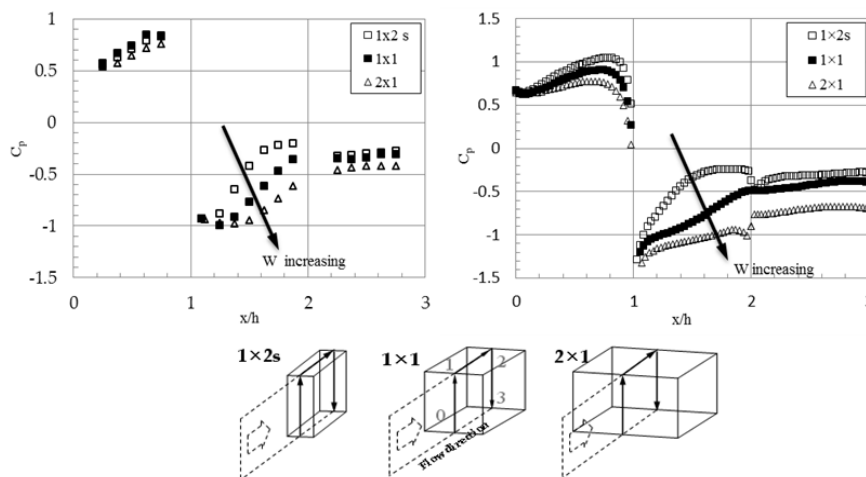


Figure 8. Mean surface static pressure coefficient along the centerline with changing transverse width.

In Figure 9, the mean surface static pressure along the side face at the mid-height of the three different boxes with changing transverse width is presented. The abscissa in the figure is normalized with the body height. As consistent with the previous figure, the pressure drop with increasing width was noticeable to the negative direction. These results demonstrate that with an increase in the horizontal width, i.e. as increasing the aspect ratio, there is a concurrent suction pressure drop on the side face.

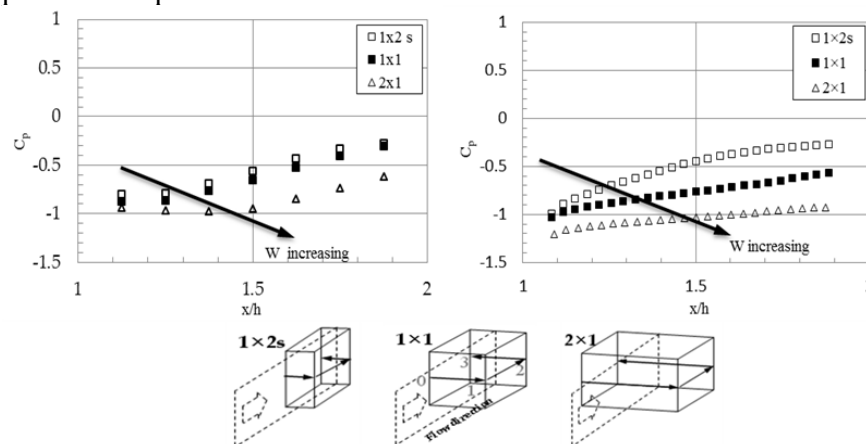


Figure 9. Mean surface static pressure coefficient at the mid height with changing transverse width.

Figures 10 and 11 show the mean surface static pressure profiles along the centerline and the mid-height of the models with changing longitudinal length(i.e. the depth). The different size of the depth maintaining the front face area makes different length of the side face - the ratio of 1:2:4, respectively. The flows around wall-mounted sharp-edged models usually have separation and often attachment around the body so that the pressure profiles in the figure have the similar shape, as shown in the above results. The pressure profiles seem to have a similar trend and it is noted that the overall distribution of the surface pressure makes good agreement and the longitudinal length of the body only affects the recovery region on the top surface, and there appears a little discrepancy in the side face.

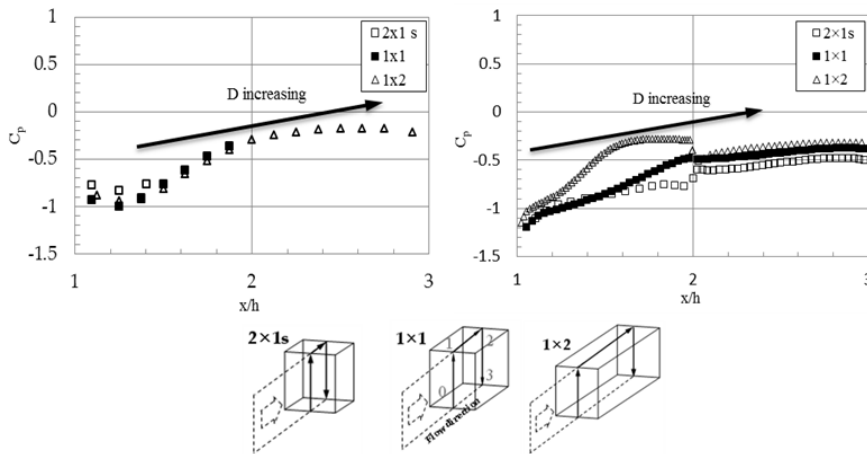


Figure 10. Mean surface static pressure coefficient along the centerline with changing longitudinal length.

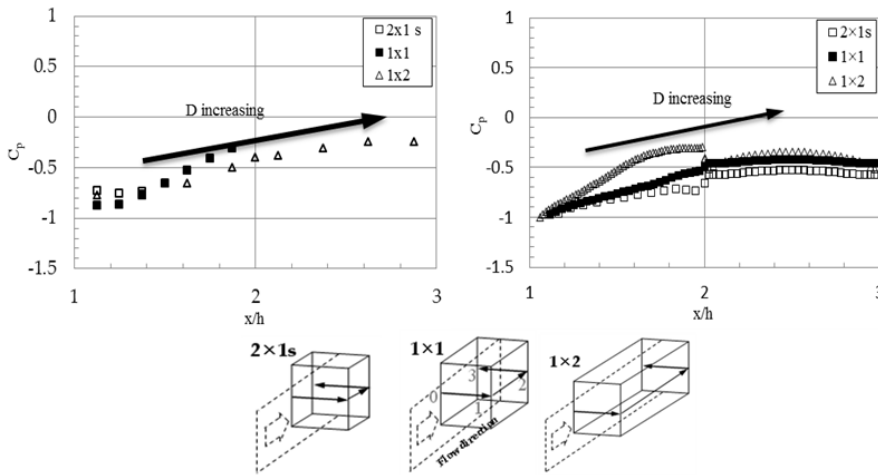


Figure 11. Mean surface static pressure coefficient at the mid height with changing longitudinal length.

Figure 12 presents the variation of the mean static pressure C_p appropriately normalized as making a proper scaling. Note that the above two pictures are the experiment data and the following two pictures are the CFD (k- ϵ model) data. It is also noted that the abscissa in the previous section was presented as a non-dimensional axis normalised by a body height h , whereas the current axes are normalized along the the axial centerline of the models by the characteristic length - the width(W , width effect) and the depth(D , the depth effect), respectively. In the figure, they are generally in good agreement. Therefore, it can be described that even

though the length and width of the rectangular body are changed, the characteristics of the static surface pressure profiles along at least the centerline itself remains unchanged, if any, it ought to be a litter scatter.

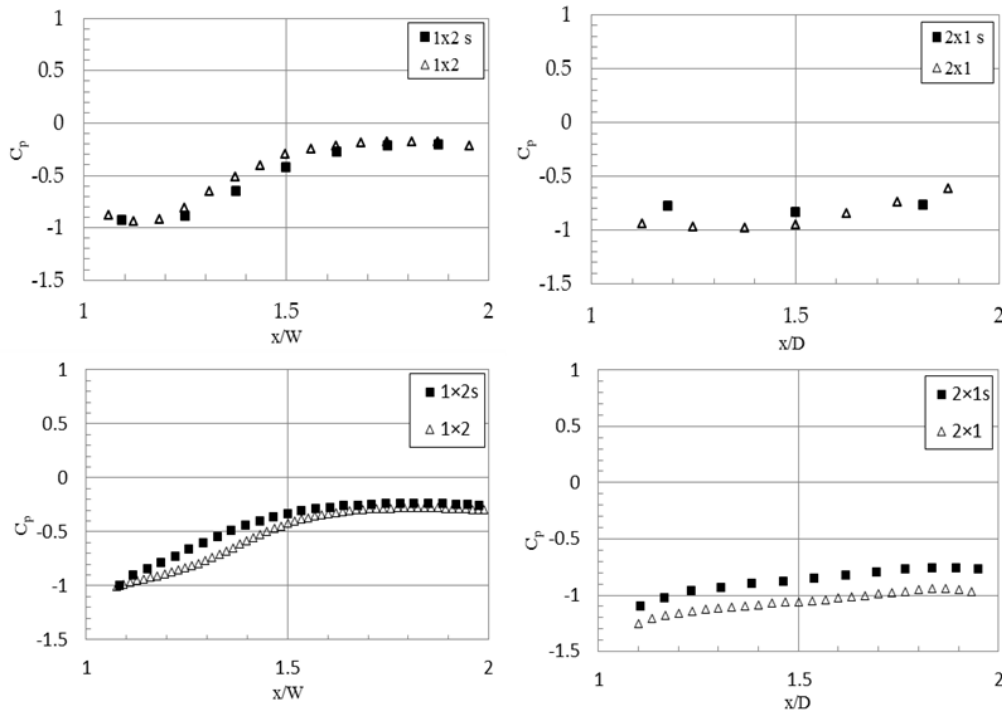


Figure 12. Mean surface static pressure coefficient along the centreline on the various boxes with the normalization of width (W) and the depth (D), respectively.

5 CONCLUDING REMARKS

Although more comparisons are needed to end up with the conclusions, the study has contributed in making the following conclusions:

- (1) As a usual, the CFD results agree well with the experiment results, as well as the earlier field results, but there still appears a litter discrepancy, which should be discussed in detail in the next study.
- (2) In this comparison, the DES model seems to be better than $k-\epsilon$ model when simulating the wind flow around a variety of bluff body.
- (3) When the aspect ratio of a rectangular body changes, the transverse width has a substantial effect on the surface pressure around the bodies, while the longitudinal length shows a little influence on the surface pressure.

6 ACKNOWLEDGEMENTS

This work was supported by the Human Resources Development of the Korea Institute of Energy Technology Evaluation and Planning (KETEP) grant funded by the Korea government Ministry of Knowledge Economy (No. 20114030200070, 20114010203080). In addition, this work was supported by the Ministry of Education, Science and Technology (MEST) in 2011.

7 REFERENCES

- 1 I.P. Castro and A.G. Robins, The flow around a surface mounted cube in uniform and turbulent streams, *J. Fluid Mech.*, 79 (1977) 307-335.
- 2 H.W. Tieleman and R.E. Akins, The effect of incident turbulence on the surface pressures of surface-mounted prisms, *J. Fluids and Structures*, 10 (1996) 367-393.
- 3 A. Cigada, S. Malavasi and M. Vanali, Effects of an asymmetrical confined flow on a rectangular cylinder, *J. Fluids and Structures*, 22 (2006) 213-227.
- 4 G.L. Larose and A.D'Auteuil, Experiments on 2D rectangular prisms at high Reynolds numbers in a pressurised wind tunnel, *J. Wind Engng Ind. Aero.*, 96 (2008) 923-933.
- 5 S. Murakami, Comparison of various turbulence models applied to a bluff body, *J. Wind Eng. Ind. Aerodyn.*, 46-47 (1993) 21-36.
- 6 Y.Q. Zhang, S.P. Arya and W.H. Snyder, A comparison of numerical and physical modeling of stable atmospheric flow and dispersion around a cubical building, *Atmos. Environ.*, 30 (1996) 1327-1345.
- 7 R.N. Meroney, B.M. Leitl, S. Rafailidis and M. Schatzmann, Wind tunnel and numerical modeling of flow and dispersion about several building shapes, *J. Wind Eng. Ind. Aerodyn.*, 81 (1999) 333-345.
- 8 G. Iaccarino, A. Ooi, P. Durbin and M. Behnia, Reynolds averaged simulation of unsteady separated flow, *Int. J. Heat Fluid Flow.*, 24 (2003) 147-156.
- 9 Y. Li and T. Stathopoulos, Numerical evaluation of wind-induced dispersion of pollutants around a building, *J. Wind Eng. Ind. Aerodyn.*, 67-68 (1997) 757-766.
- 10 Y. Tominaga and T. Stathopoulos, Numerical simulation of dispersion around an isolated cubic building: comparison of various types of k- ϵ models, *Atmospheric Environment.*, 43 (2009) 3200-3210.
- 11 S. Krajnovic' and L. Davidson, Large-eddy simulation of the flow around a surface-mounted cube using a dynamic one-equation subgrid model, In S. Banerjee and J. Eaton (Eds.), *The First Int. Symp. on Turbulence and Shear Flow Phenomena*, Beggel House, Inc, New York, 1999.
- 12 S. Krajnovic' and L. Davidson, Large eddy simulation of the flow around a three-dimensional bluff body, *AIAA 2001-0432*, 39th AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada, 8-11 January, 2001.
- 13 K. Shah and J. Ferziger, A fluid mechanics view of wind engineering: large eddy simulation of flow past a cubic obstacle, *J. Wind Eng. Ind. Aero.*, 67-68 (1997) 211-224.
- 14 F. Jochen and V T. Dominic, Hybrid LES/RANS methods for the simulation of turbulent flows, *J. Progress in Aerospace Sciences*, 44(5) (2008) 349-377.
- 15 E. Simiu and R.H. Scanlan, *Wind effects on Structures*, Wiley, New York, 1996
- 16 N.J. Cook, Wind tunnel simulation of the adiabatic atmospheric boundary layer by roughness, barrier and mixing device methods, *J. Wind Engng Ind. Aero.*, 3 (1978) 157~176.
- 17 P.J. Richards, P.J. Hoxey, and L.J. Short, Wind pressure on a 6m cube, *J. Wind Engng Ind. Aero.*, 89 (2001) 1553-1564.
- 18 H.C. Lim, Generation of a turbulent boundary layer using LES, *Trans of the KSME (B)*, 31 (2007) 680-687.
- 19 H.C. Lim, Ian P. Castro and R.P. Hoxey, Bluff bodies in deep turbulent boundary layers Reynolds-number issues, *J. Fluid Mech.*, 571(2007) 97-118.
- 20 S. Murakami and A. Mochida. 3D numerical simulation of airflow around a cubic model by means of k- ϵ model, *J. Wind Engng Ind. Aero.*, 31 (1988) 283-303.