NUMERICAL INVESTIGATION OF MECHANISM OF CORNER-CUT IN RECTANGULAR BLUFF BODY FROM THE VIEW POINT OF FLOW FIELD

Yokohama National University Student Member ○ Hoai Thi Ho
Yokohama National University Regular Member Hiroshi Katsuchi
Yokohama National University Regular Member Haeyoung Kim

1. INTRODUCTION

Cutting corner at rectangular bluff body is often used to suppress drag coefficient and wind induced vibration in bridge towers. Present numerical study using Large Eddy Simulation reproduces the flow pattern around a tower cross section from Okajima et al (1991) to investigate the mechanism of corner-cut on aerodynamic stability. The corner-cut modifies the oncoming flow, keeps the shear layer close to side surface and narrows the wake behind rectangular bluff body. Pair of recirculation at front corners accelerate the velocity and change the pressure distribution around bluff body, then reduce the drag coefficient and strengthen dynamic stability. Meanwhile, the recirculation in rear corners formed by the interaction of both upcoming flow and near-wake vortices keep the Karman vortex far away to downstream.

2. NUMERICAL SIMULATION

2.1 Numerical details

In present study, Large Eddy Simulation (LES) with oneEqEddy model is conducted for flow around a corner-cut rectangular of a tower model has side aspect ratio B/D=0.66 and corner cut dimension d/D=0.115, b/D= 0.18, which was experimentally investigated in Okajima et al (1991). A classical O-type mesh with a computation domain of 40D was spatially divided into 130-130 grids and 1D span-wise ratio with grid resolution 0.1D is employed. The radical mesh is clustered to the cut-corner rectangular with a stretching ratio of 1.05, resulting the first normal grid spacing located around wall unit y+<1.The total size of grids is 0.7 million elements.The simulation employs a constant velocity for the upstream boundary, a zero-gradient pressure for the outlet. Non-slip wall condition in corner-cut rectangular and periodic boundary condition in front and back are specified. The study is implemented by OpenFOAM toolbox which was originally developed as a high-end C++ class's library for broad range of fluid dynamics applications. The mesh is generated by Gmsh utilities and then be converted to OpenFOAM by *gmshToFoam* application.

In LES, the velocity vector $\tilde{U}$ is decomposed by the spatial filter velocity U and fluctuation u. The Navier-Stokes (N-S) equations derived from mass and momentum conversion of the incompressible flow take the forms as follow.

$$\nabla.\tilde{U} = 0 \qquad (1)$$

$$\tilde{U}_t + (\tilde{U}.\nabla)\tilde{U} = -\frac{1}{\rho}\nabla\tilde{p} + \nu\Delta(\tilde{U}) - T_{sgs} \qquad (2)$$

where ρ is fluid density and T_{sgs} is sub grid tensor which can be expressed by Boussinesq assumption.

2.2 Numerical verification

The present numerical simulation result has been verified with the previous wind tunnel test in Okajima et al. The result of pressure coefficient at point 1, 2, 3, 4, Strouhal number and wake velocity distribution was shown in Table 1 and Figure 1 as follow. Effects of blockage ratio to numerical results were considered according to proposed equations of Sohankar et al (2000). A fair agreement is found between the numerical simulation results and the experiment data.

Table 1: Pressure coefficient and Strouhal number

	Cp				Str
Points	1	2	3	4	
Exp.Okajima	~1.7	~1.1	~1	~1	0.2
LES.present	1.72	1.27	1.17	1.21	0.203

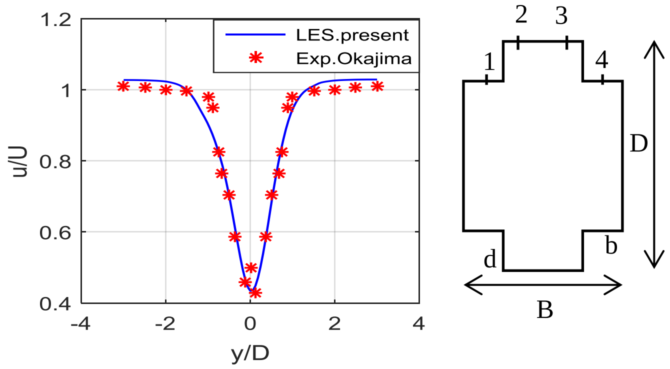

Fig.1 Left -Wake velocity distribution at x/D=2.68
Right-corner-cut rectangular dimensions

3. RESULT AND DISCUSSIONS

Corner cut was proved to be an effective method for drag reduction and dynamic stabilization of rectangular bluff body. Shiraishi et al (1988) stated that at certain corner-cut dimension, the separated shear layer appears considerably close to body side surface, therefore forms rather narrow wake and increases critical velocity for galloping. Similarly, Ueda et al (2009) found out that if the shear layer from the leading corner reattaches from the side surface of the bluff bodies, the high drag reduction could be achieved. In flow pattern visualization of present simulation, the corner-cut rectangular shows lower shear layer height and longer shear layer length in the wake compared to the normal one (Figure 2).

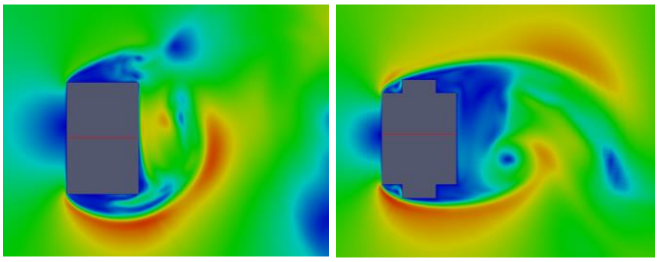

Fig. 2 Instantaneous velocity around bluff body (t=3.8s)

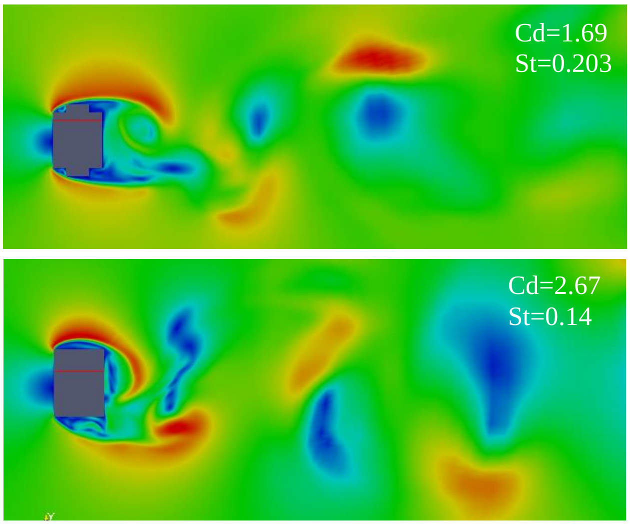

Fig. 3 Wake velocity behind bluff body (up-with cut corner, down-without cut corner)

In literature, it is said that a rectangular in critical aspect ratio (B/D~0.6) has extremely strong curvature vortex shedding, causing large separation and highest drag coefficient. In present simulation, the coming flow attaches the 1st corner-cut leading edge, goes around or reattaches the 2nd corner-cut leading edge, then controls the shear layer close to the side surface and keeps the Karman vortex form far from the rear side of bluff body. As a result, the size of wake structures is much smaller in corner-cut case, also the value of Cd and Strouhal number changes significantly as shown in Figure 3. Moreover, a pair of recirculation formed at front corner-cut works as a reshaping countermeasure, strengthens the flow energy and velocity and allows the flow goes around the bluff body easily. Thus, the separation and wake width are suppressed. The recirculation in rear corner was form by the interaction of incoming flow and near wake vortices structure, changing the total base pressure around rectangular bluff body and induced drag force. Compared to normal rectangular, the velocity profile at x=0 on the top of corner-cut bluff bodies has higher gradient value and lower separation (Figure 4). Also, the significant change of pressure distribution is found at top side surface of corner-cut bluff body (Figure 5).

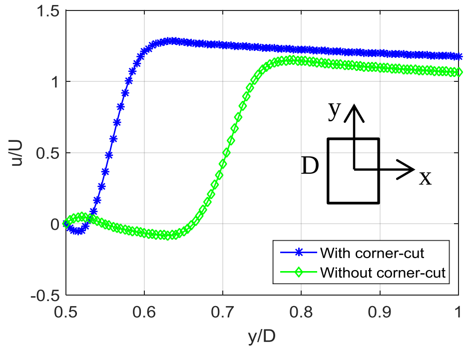

Fig. 4 Velocity profile on top surface, x=0

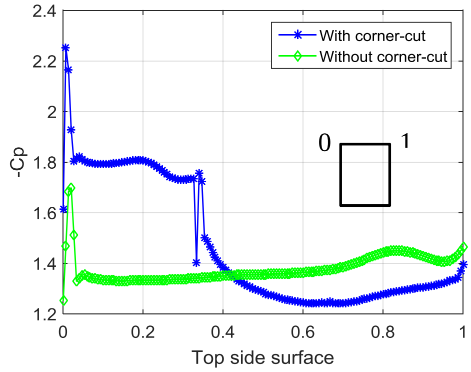

Fig. 5 Pressure distribution top surface

4. CONCLUSION

The present numerical simulation clarifies the effect of corner-cut in aerodynamic stability from the viewpoint of flow field around bluff body and at wake. It is said that the corner-cut not only reshapes the upstream flow but also changes pressure distribution and dissipates the wake structure at downstream.

REFERENCES

1. Okajima, A., Ueno, H., Abe, A., Influence of Reynold number on flow and aerodynamic characteristics of bluff bodies with rectangular section of cut corners, Journal of Wind Engineering and Industrial Aerodynamic, 49, 1992,p1-13.
2. Sohankar, A., Davison, L.,Norberg, C., Large Eddy simulation of flow past a square cylinder: Comparison of Different Sub grid scale models, Journal of Fluid Engineering, 122(1). 1999, P39-47
3. Shiraishi,N.,Mastumoto,M.,Shirato,H.,Ishizaki, H., On aerodynamic stability effect for bluff rectangular cylinder by their corner-cut, Journal of Wind Engineering and Industrial Aerodynamic, 1988, p371-380
4. Ueda,Y., Kurata, M., Kida,T., Iguchi, M., Visualization of flow past a square prism with cut-corners at the front-edge, Journal of Visualization, 12(4), 2009, p383-393.