VLES OF DRAG REDUCTION FOR A SQUARE CYLINDER WITH SHAPED CORNER BASED ON OPENFOAM

ZHONGYU CHENG[1], ZHAOYANG XIA[1], XINGSI HAN[1,2], JUNKUI MAO[1], FENG JIN[1]
[1] *College of Energy and Power Engineering, Nanjing University of Aeronautics and Astronautics, Aero-engine Thermal Environment and Structure Key Laboratory of Ministry of Industry and Information Technology, Nanjing 210016, China*
[2] *Corresponding author, email: xshan@nuaa.edu.cn*

Keywords: *Very-Large Eddy Simulation, drag reduction, passive flow control, shaped corner, OpenFOAM*

1. Introduction

The drag reduction of high Re number turbulent flow past a bluff body has attracted extensive interests in both academia and industry for decades. For industrial applications, it can reduce the fuel consumption and improve the performance of the vehicles. In academic research, the flow configuration provides abundant fundamental problems, such as massive separated flow, unsteady vortex evolution, flow control strategies, etc.

Generally, the flow control strategies can be divided into two main groups [1]: passive flow control and active flow control. The present study focuses on the passive flow control method which mainly adopts to modify the geometry for reducing the drag. Turbulent flow around a square cylinder has been a classic flow test case [2] for various problems. It was also used for drag reduction studies [3-5]. Previous experiments have found that by shaping the corner of the square cylinder, the drag can be reduced by 50% for some conditions. Although the significant drag reduction result has been observed in the experiments, the underlying physical mechanism has not been well understood. On the basis, the current study aims to reveal some of the key flow mechanics based on high-fidelity numerical simulation.

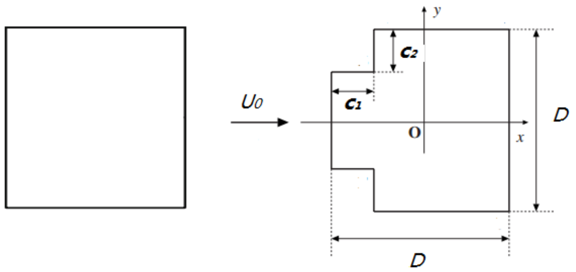

Figure 1: Square cylinder and a shaped-corner square cylinder (a cut plane).

High fidelity simulation, such as Large Eddy Simulation (LES) has been applied for various flow control studies. It shows high potential to investigate the involved flow mechanics. However, LES suffers from the high computation cost for high Re number turbulent flow. In recent years, hybrid turbulence modeling method [6] has been developed rapidely as it combies the advantages of different turbulence modeling approaches, such as hybrid RANS-LES method. One of the hybrid method is VLES method (Very-Large Eddy Simulation) [7, 8], which has been validated in some classic flow cases and performs very well. It is found that the VLES method can provide accurate predictions using quite coarse numrical mesh, and thus suitable for complex high Reynolds turbulent flow. The other objective of the present study is to assess the VLES method for flow control problems.

2. Numerical methods

The VLES method [7] based on the standard k-ε turbulence model is applied. The governing equations of turbulent kinetic energy k and its dissipation rate ε is the same as in the standard k-ε turbulence model. The turbulent viscosity is modelled by introducing a new resolution control function Fr. It has the form shown in Eq. (4), where L_c, L_i and L_k are the cut-off length scale, integral length scale and Kolmogorov length scale, respectively.

$$D\rho k / Dt = P_k - \rho\varepsilon + \partial\left[\left(\mu + \mu_t / \sigma_k\right)\partial k / \partial x_j\right] / \partial x_j \tag{1}$$

$$D\rho\varepsilon / Dt = \left(\varepsilon / k\right)\left(C_{\varepsilon 1}P_k - C_{\varepsilon 2}\rho\varepsilon\right) + \partial\left[\left(\mu + \mu_t / \sigma_\varepsilon\right)\partial\varepsilon / \partial x_j\right] / \partial x_j \tag{2}$$

$$\mu_t = Fr \cdot \rho C_\mu k^2 / \varepsilon \tag{3}$$

$$Fr = \min\left[1.0, \left(\left(1.0 - \exp(-\beta L_c / L_k)\right) / \left(1.0 - \exp(-\beta L_i / L_k)\right)\right)^2\right] \tag{4}$$

With the mesh resolution changing, the resolution control function *Fr* changes and then it determines how much of the turbulence is modelled. Thus, with different mesh resolution, the VLES method can work in different modelling modes, ranging from the RANS, LES to DNS. More details about the VLES method can be found in refs. [7-9].

The present VLES method has been implemented in the OpenFOAM toolbox [10]. The unsteady simulations are performed using the pimpleFOAM solver. The convective terms are discretized using a second-order central differencing scheme coupled with a small fraction of first order upwind scheme. The temporal advancement was approximated using a second-order implicit Crank-Nicolson scheme.

3. Results and discussions

Firstly, the classic turbulent flow around a square cylinder at Re=22000 is carried out to assess the performance of the VLES turbulence modelling using OpenFOAM toolbox. Two meshes with around 0.7 million and 1.2 million cells are used in the present simulations. The mean and RMS velocities at the central line downstream the square cylinder are shown in Fig. 2, with the experimental data [2] included. It can be seen that the present VLES model gives quite good predictions compared with experimental data for both the mean and RMS velocities. Also, with decreasing the mesh resolution, the results changes slightly, which demonstrates that the present VLES method is not sensitive to the mesh resolutions, and coarse meshes can be used for VLES studies.

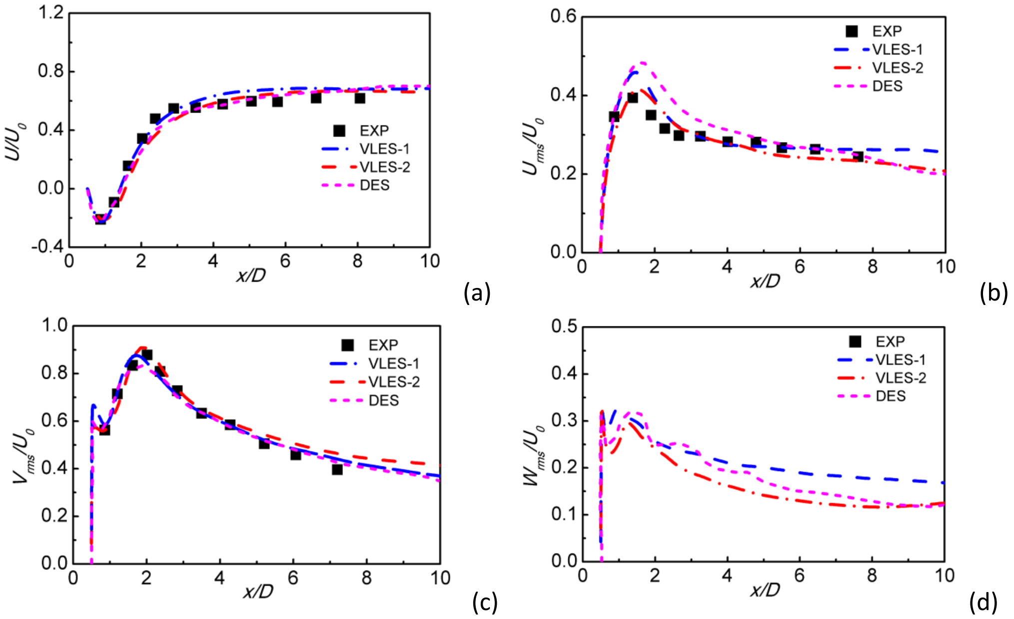

Figure 2: Comparisons of mean and RMS velocities along the central line in the middle plane of the square cylinder. (a) mean velocity in the x direction; (b) RMS velocity in the x direction; (c) RMS velocity in the y direction and (d) RMS velocity in the z direction. Experimental data is from ref. [2] and DES results from ref. [9].

The comparisons in Fig. 2 demonstrate that the present simulations with VLES modelling can give satisfactory results for the flow around a square cylinder. On the basis, the simulations are applied for the flow around the shaped-corner square cylinder, where the two corners at the front (as in Fig. 1) are cut-off. In the present study, two test cases are applied, i.e., case F1 with cut-corner scales of c_1/D=0.15 and c_2/D=0.15, case F2 with c_1/D=0.20 and c_2/D=0.15.

Table 1: The global flow parameters for turbulent flow around a square cylinder

		St	Cd_mean	Cd_rms	Cl_rms	ΔCd%
Square cylinder (F0) c_1/D=0, c_2/D=0	VLES	0.12	2.3	0.26	1.03	—
	Exp. [2]	0.13	2.2	—	—	—
	LES [11]	0.12-0.13	2.0-2.3	0.16-0.20	1.2-1.5	—
Shaped cylinder (F1) c_1/D=0.15, c_2/D=0.15	VLES	0.14	1.4	0.20	0.48	-39%
	Exp. [3]	0.16	1.3	—	—	-41%
Shaped cylinder (F2) c_1/D=0.20, c_2/D=0.15	VLES	0.17	1.2	0.18	0.37	-48%
	Exp. [3]	0.22	1.1	—	—	-50%

The global flow parameters for the baseline case F0 and two shaped corner cases, F1 and F2, are summarized in Table 1 with experimental data and previous simulation results. The comparisons show that the present VLES method generally gives good predictions compared with experimental data for all the three cases. The present simulation predicts a significant drag reduction of around 39% for case F1 and 48% for case F2, which are quite close to experimental data. The results show that with the drag reduction, the Strouhal number is increasing and the RMS drag and RMS lift decreasing.

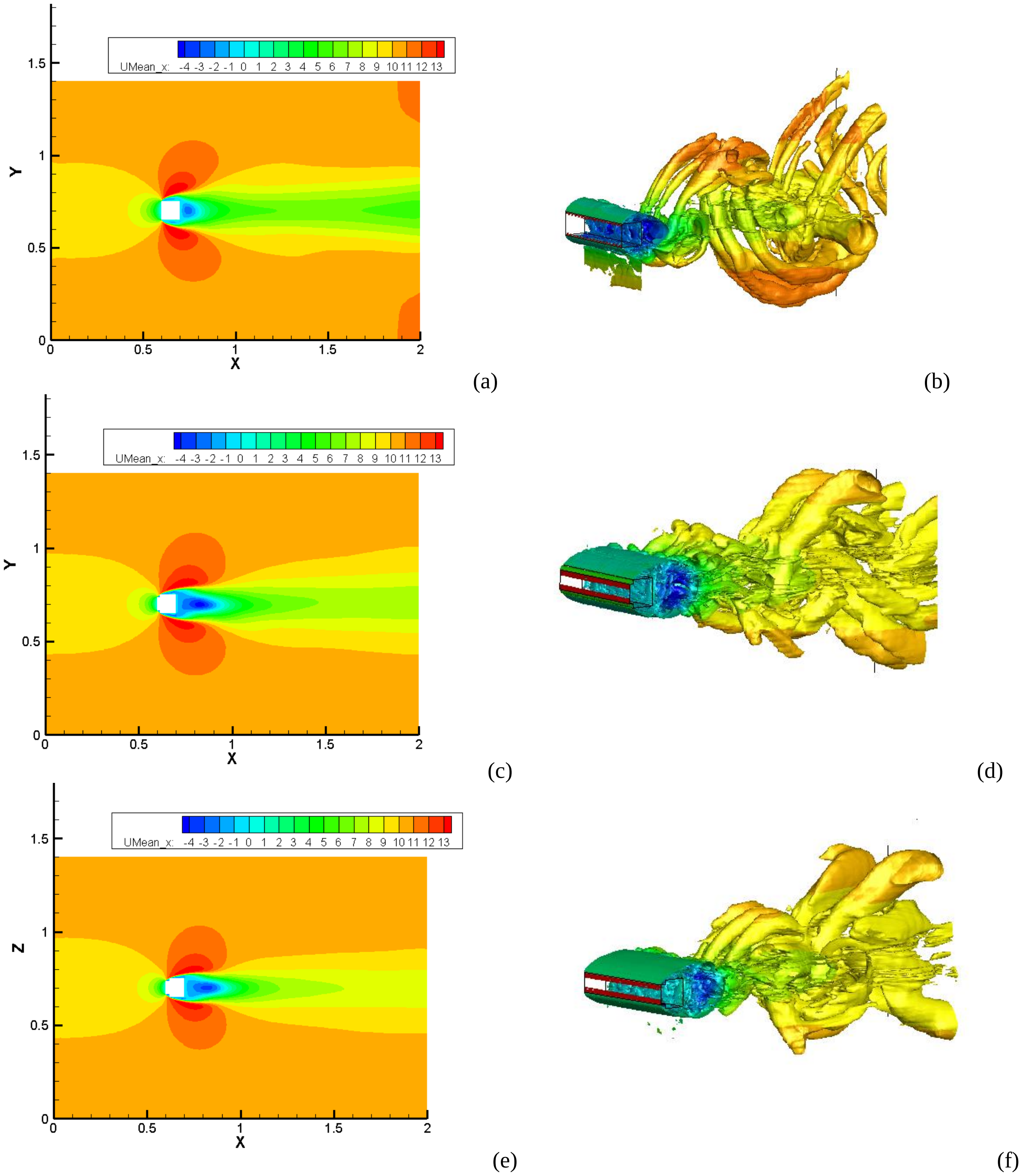

Figure 3: Results of the mean velocity (*Ux*) and the iso-surface of the second invariant of the velocity gradient (*Q*=100) for different cases: (a) *Ux* for case F0; (b) *Q* for case F0; (c) *Ux* for case F1; (d) *Q* for case F1; (e) *Ux* for case F2; (f) *Q* for case F2.

The mean velocity and the second invariant of the velocity gradient (Q) for different cases are shown in Fig. 3, where case F0 is the baseline case, F1 and F2 are the shaped corner cases. The mean velocity shows that cases F1 and F2 give a longer recirculation region compared with the baseline F0, which are consistent with previous observations. For the cases F1 and F2, a larger drag reduction is observed in case F2. However, the recirculation region behind the cylinder is not bigger than the case F1, while the width of the flow around the cylinder seems become narrow in case F2. The results imply that the drag reduction mechanisms of the case F1 and F2 are not the same. Detailed analysis will be carried out in the full length paper. The results of Q in Fig. 3 show that with the drag reduction, the wake region seems to become narrow and more stable. It also shows that bigger vortex structures can be observed in cases F1 and F2 compared with the baseline F0.

The results show that the present VLES modelling can give satisfactory results for the passive flow drag reduction simulations. The physical mechanism can be explored with the obtained simulation results. The study demonstrates that the OpenFOAM toolbox provides a very good platform for perform such kind of high-fidelity simulation studies.

More simulation results, comparisons and detailed analysis will be given in the full length paper.

Acknowledgements
This work was financially supported by the National Natural Science Foundation of China (grant No: 51606095), the Jiangsu Provincial Natural Science Foundation (grant No: BK20160794), and the Aerospace Power Foundation of China. X.S. Han acknowledges the support of the Jiangsu Specially-Appointed Professor Program

References

[1] M. Gad-El-Hak, A. Pollard, J.P. Bonnet, Flow Control: Fundamentals and Practices, Assembly Automation, 42(2): 118 – 124, 1998.

[2] D.A. Lyn, S. Einav, W. Rodi, et al. A laser-Doppler velocimetry study of ensemble-averaged characteristics of the turbulent near wake of a square cylinder, Journal of Fluid Mechanics, 304: 285–319, 1995.

[3] M. Kurata, Y. Ueda, T. Kida, et al., Drag Reduction Due to Cut-Corners at the Front-Edge of a Rectangular Cylinder With the Length-to-Breadth Ratio Being Less Than or Equal to Unity, ASME Journal of Fluids Engineering, 131: 064501, 2009.

[4] G.S. He, N. Li, J.J. Wang, Drag reduction of square cylinders with cut-corners at the front edges, Experiments in Fluids, 55: 1745, 2014.

[5] A. Sohankar, M. Khodadadi, E. Rangraz, Control of fluid flow and heat transfer around a square cylinder by uniform suction and blowing at low Reynolds numbers, Computers & Fluids, 109:155-167, 2015.

[6] P. Sagaut, S. Deck, M. Terracol, Multiscale and Multiresolution Approaches in Turbulence, LES, DES and Hybrid RANS/LES Methods: Applications and Guidelines, 2nd ed., Imperial College Press, London, 2013.

[7] X. Han, S. Krajnovic, An efficient very large eddy simulation model for simulation of turbulent flow, International Journal for Numerical Methods in Fluids, 71(11):1341-1360, 2013.

[8] X. Han, S. Krajnovic, Very-Large Eddy Simulation Based on k-ω Model. AIAA Journal, 53: 1103-1108, 2015.

[9] X. Han, T. Ye, Y. Chen, Calibration of a new very large eddy simulation (VLES) methodology for turbulent flow simulation, Science China: Physics, Mechanics and Astronomy, 55: 1905-1914, 2012.

[10] OpenFOAM: The Open Source CFD Toolbox. User Guide Version 4.0, the OpenFOAM Foundation, 2017.

[11] A. Sohankar, L. Davidson, Large eddy simulation of flow past a square cylinder: comparison of different subgrid scale models, ASME Journal of Fluids Engineering, 122:39–47, 2000.