European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2006
P. Wesseling, E. Oñate, J. Périaux (Eds)

LARGE EDDY SIMULATION OF 3D TURBULENT FLOW AROUND DEEP-SEA MARINA STRUCTURE

Jisheng Zhang, Yakun Guo

Department of Engineering, University of Aberdeen, Aberdeen, AB24 3UE, UK.
Tel: ++44 1224 272987, Fax: ++44 1224 272497
Email: j.zhang@abdn.ac.uk.

Key words: Large Eddy Simulation, Overlapping Circular Cylinder, Vortex Shedding

Abstract. *The purpose of this paper is to investigate the highly complex interaction between turbulent flow and deep-sea marina structure using a commercial CFD code (FLUENT 6.2) with Large Eddy Simulation (LES) approach. A series of three-dimensional LES of wake flows past overlapping cylinder of finite height are carried out with a range of Reynolds number* $1.0\times10^4 \sim 1.0\times10^5$*. Three different heights of circular mud mat having a fixed diameter are simulated to study the effect of the height of mud mat on turbulent flow field around this kind of bluff body. The complex separated flow structures and wake properties are simulateed and discussed.*

1 INTRODUCTION

The flow around a circular cylinder has been extensively investigated by means of laboratory experiments and numerical simulations for many years mainly due to its industrial relevance and engineering applications. Comprehensive experimental results and reviews of oscillating flow past circular cylinder can be found in references [1-8], while various numerical simulation methods including Large Eddy Simulation (LES) and Reynolds-averaged Navier-Stokes (RANS) model have been developed to study such complex flow field for a range of Reynolds numbers[9-15]. Most of researches focus on the flow around a single circular cylinder.

In practical deep-sea or offshore engineering applications, the phenomena in question that the combination of surface-mounted base, mostly being circular cylinder, of benthic construction and a circular mud mat utilized to protect the resuspension of fine sediment caused by the flow around the cylinder can be frequently encountered. The flow field around this kind of bluff body consists of flow separation at the front corner, recirculation, vortex shedding and is much more complicated than a single circular cylinder case. However, references concerning the flow around this kind of overlapping circular cylinder of finite height are still lacking and substantial research is required.

With the rapid development of computer technique, LES for turbulent flow has achieved great progress in recent years, which makes it possible to numerically investigate the turbulent effect on hydrodynamic characteristics of a bluff body. The purpose of this paper is to investigate the highly complex interaction between turbulent flow and this kind of deep-sea marina structure using a commercial CFD code (FLUENT 6.2) with LES approach.

2 LARGE EDDY SIMULATION

2.1 Numerical simulations

It is significant, crucial and difficult to find a proper numerical simulation method to predict the complex turbulent motions around overlapping cylinder due to their highly three-dimensional, unsteady and fairly irregular characteristics at high Reynolds number. Such details of the turbulent fluctuation motion can be described by the unsteady three-dimensional Navier-Stokes equations together with the continuity equation. The most accurate approach of turbulence simulation, Direct Numerical Simulation (DNS), is to solve the governing equations without averaging or approximation other than numerical discretizations whose errors can be estimated and controlled[16]. Since all motions need to be resolved with this approach, the size of the numerical mesh must be smaller than the size of the small-scale motion where energy dissipation takes place. As a result, the large requirement of computational power is inevitable when DNS is implemented.

Based on RANS, turbulence models, such as the mixing length and k-ε model, have achieved considerable success in numerical simulation of turbulent flow of engineering interest. However, Franke & Rodi (1991) indicated that k-ε model has failed to reproduce unsteady wakes behind bodies[17]. Though the Reynolds stress turbulence model is capable of modelling complex flows, the degree of complexity and the amount of computation required is significantly increased to the level so that the advantage of RANS turbulence modelling compared with LES has been largely lost[9].

In LES approach, only the large-scale turbulent motions that can be resolved on a numerical grid are estimated explicitly by solving the 3D time-dependent Navier-Stokes equations, while the small-scale motions that cannot be resolved need to be considered by a subgrid-scale model. When LES is used to simulate the flow at high Reynolds number, a special near-wall treatment has to be introduced with the shortcoming that the near-wall regions cannot be properly resolved[18]. LES are also three-dimensional, time dependent and expensive but much less than DNS of the same flow. To find the right balance between requirement of computational power and satisfaction of engineering interest, LES method is found to be actually more reasonable and chosen for this study.

2.2 Model settings of LES

A commercial finite volume CFD code, FLUENT 6.2, has been utilized to investigate the interaction between turbulent flow and overlapping circular cylinder. The Navier-Stokes equations for incompressible fluid flow in combination with the Smogorinsky model are used in the present LES.

2.2.1 Model validation

In LES simulation, so many choices of model settings provided by FLUENT 6.2 can be used, which will affect to some extent the accuracy and stability of computation. In order to make sure that the proper settings are chosen, the specific numerical results are validated and compared with the available experimental results at a Reynolds number of 1.4×10^5 reported in [19]. Table 1 lists the three-dimensional simulation settings applied in this study, and no slip condition is used on the surface of circular cylinder. The computational and experimental data of drag coefficient C_d and Strouhal number St are listed in Table 2. The comparisons show a good agreement between LES model and experiment, indicating that the model settings are properly selected.

Settings	**Choices**
Simulation	3D (3ddp)
Solver	Segregated Implicit
Temporal discretisation	Second Order Implicit
Pressure	PRESTO!
Momentum equation	Bounded Central Differencing
Pressure-velocity coupling	PISO
Subgrid-Scale model	Smagorinsky-Lilly (Cs=0.1)
Inlet boundary condition	Velocity-inlet
Outlet boundary condition	Outflow
Top boundary condition	Symmetry
Bottom boundary condition	Symmetry
Lateral boundary condition	Symmetry

Table 1: Three-dimensional simulation settings of LES model

	C_d, average	**St**
Simulation	1.224	0.195
Experiment	1.237	0.179

Table 2: Comparison of computational and experimental results at $Re=1.4\times10^5$

2.2.2 Model application

The LES model is carried out to study the turbulent flow around overlapping circular cylinder of finite height, and the same model settings listed in Table 1 are applied, except that no-slip Wall boundary condition is used for bottom boundary condition to consider the effect of sea bottom on flow structure. The overlapping cylinder is located at the origin of the coordinate system, and *x, y, z* represent the stream-wise, lateral and wall-normal direction, respectively. Figure 1 shows the dimensions of computational domain in horizontal and vertical planes, and Values of simulation parameters based on a practical engineering application are: diameter of upper circular cylinder D=0.3m; height of upper circular cylinder H=0.5m; diameter of lower circular cylinder =3D; height of lower circular cylinder h=0.083D (0.025m), 0.167D (0.05m) and 0.25D (0.075m) with free-stream velocity U_0 varying from 0.05m/s to 0.5m/s. Such parameters give rise to Reynolds number (defined as $Re=U_0D/\upsilon$) being $1.0\times10^4 \sim 1.0\times10^5$.

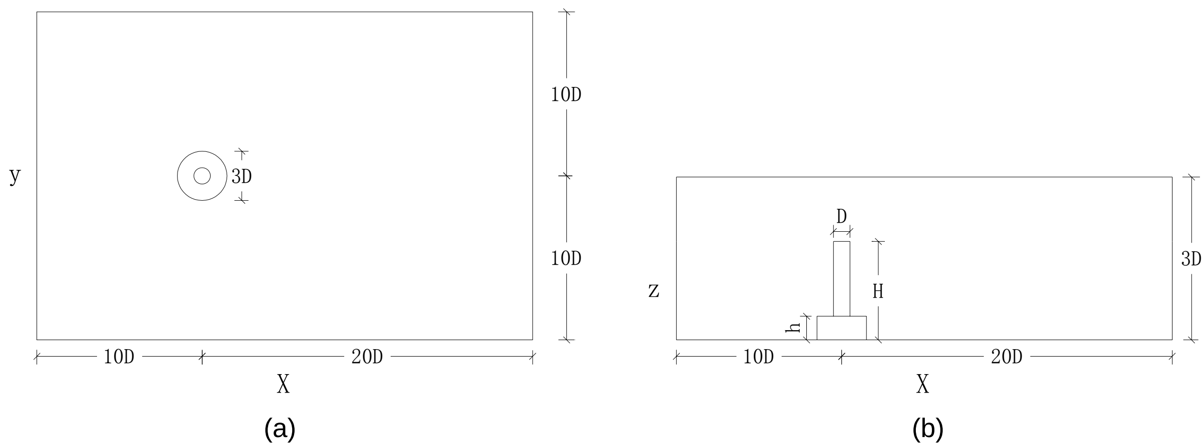


Figure 1: Sketch of computational domain in (a) horizontal and (b) vertical planes

3 RESULTS AND DISCUSSION

To obtain the LES results, the simulations have been implemented more than 400 non-dimensional time units (D/U_0). The statistical data presented and discussed here are collected over the last 300 time units.

3.1 Flow structure

The flow around overlapping circular cylinder with a $Re=1.0\times10^4 \sim 1.0\times10^5$, in contrast to the single cylinder, is highly three-dimensional and complicated, including a laminar boundary layer along the cylinder surface, laminar separation, and transition from lamina to turbulence in the shear layer shortly after separation through a Kelvin-Helmholtz instability and further span-wise instabilities[14]. Figure 2 shows the instantaneous flow structures behind an overlapping cylinder with h=0.167D at a Reynolds number of 5.0×10^4. Laminar separation takes place at the front corners of the overlapping cylinder, while lateral vortex rollers inter-wind and merge upon traveling downstream. As shown in Figure 3, the average pressure of the sidewall surface is greater than that on top surface of upper cylinder and this pressure difference causes some of the fluid to attempt to migrate from the sidewall to top surface. At the same time, the fluid is swept downstream, forming a trailing vortex. At the top rear end the separation process is highly complex and very irregular because of the curved trailing edge and the separation at the sidewalls, so that organized motion can not be detected. The unsteady Necklace vortex is formed at the junction of the overlapping cylinder and flat wall bottom. The vortical motion behind the overlapping cylinder, which is highly influenced by the height of lower cylinder and inflow velocity, will be discussed in average flow. Further downstream in the wake the vertex shedding increases in size and become smoother.

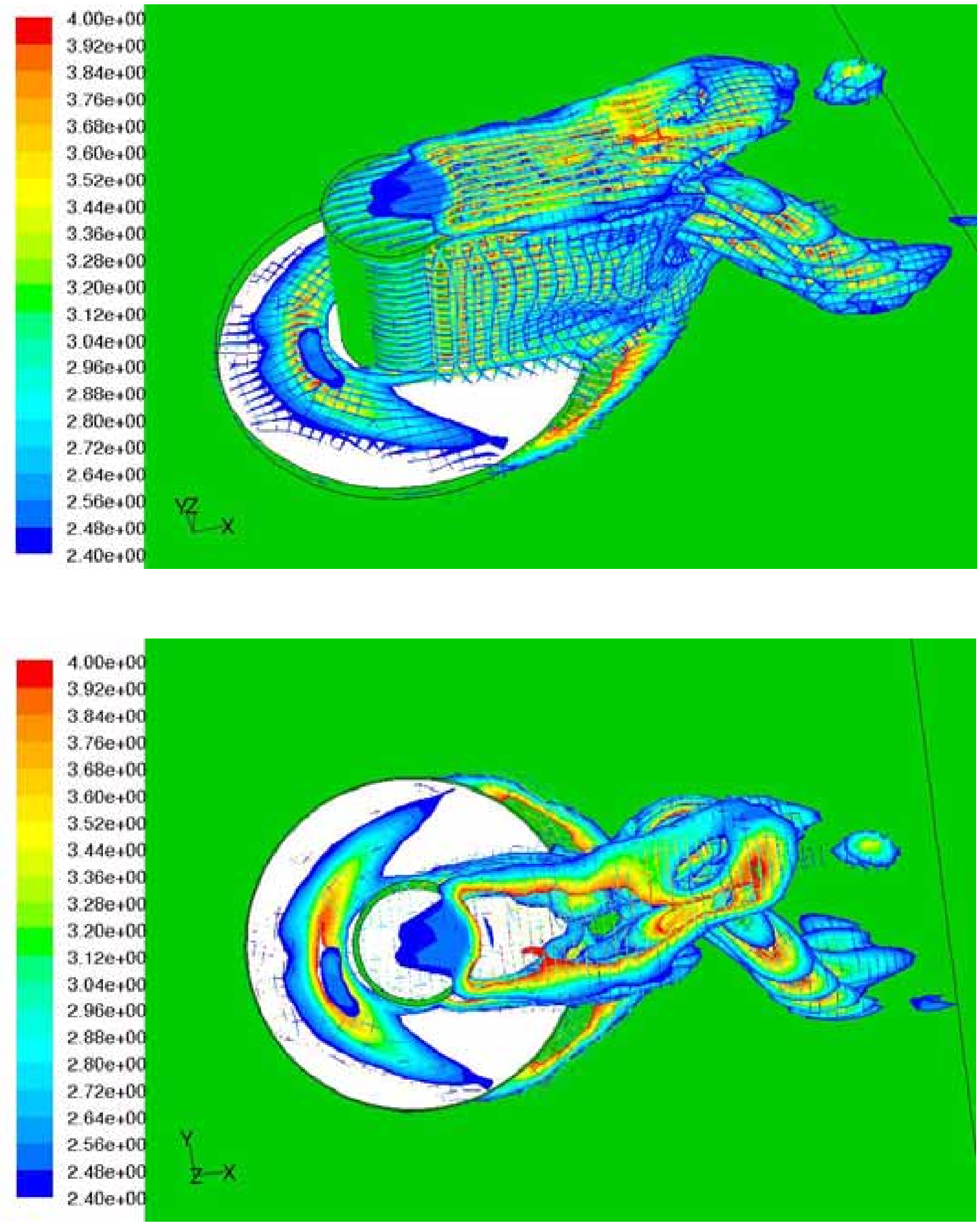


Figure 2: Instantaneous flow structures behind an overlapping cylinder with h=0.167D at Re=5.0×10^4; vortical structures are colored by vorticity magnitude. Unit is 1/s.

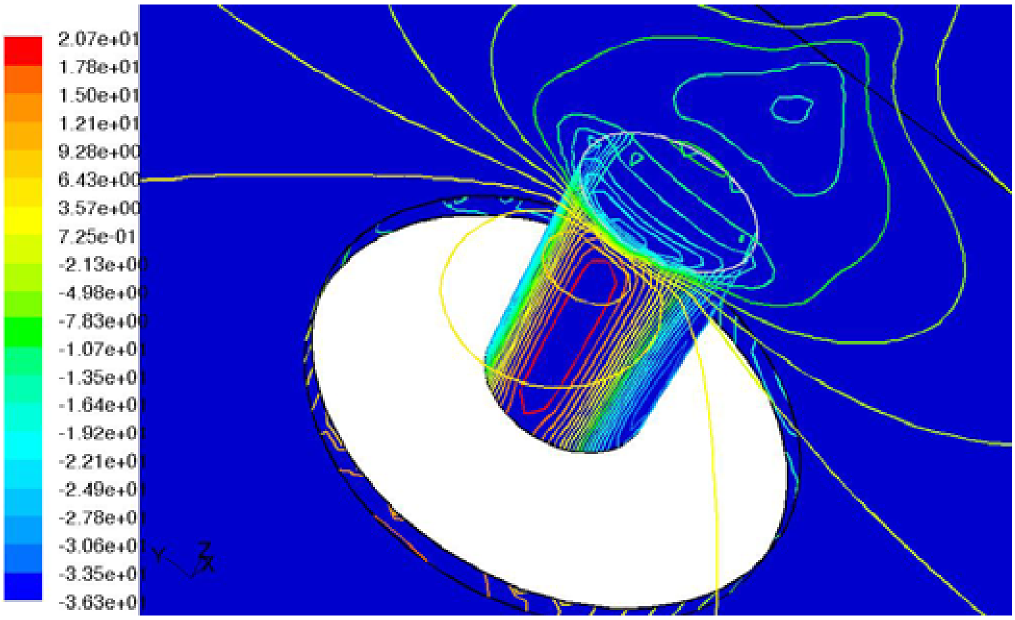


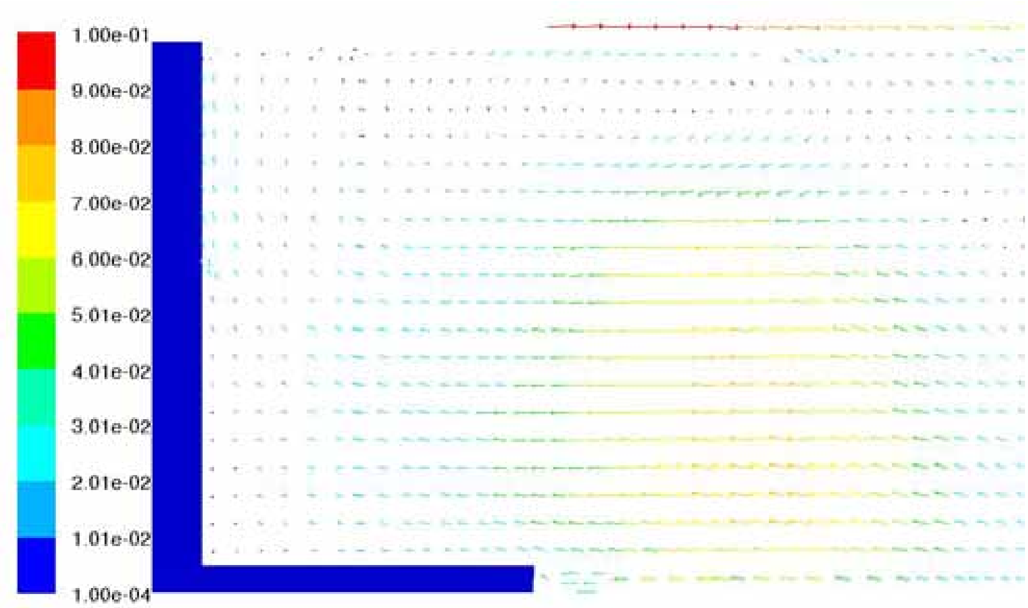
Figure 3: The average pressure distributions on the top and sidewall surfaces of upper cylinder. Unit is Pa.

3.2 Effect of height h on the flow field

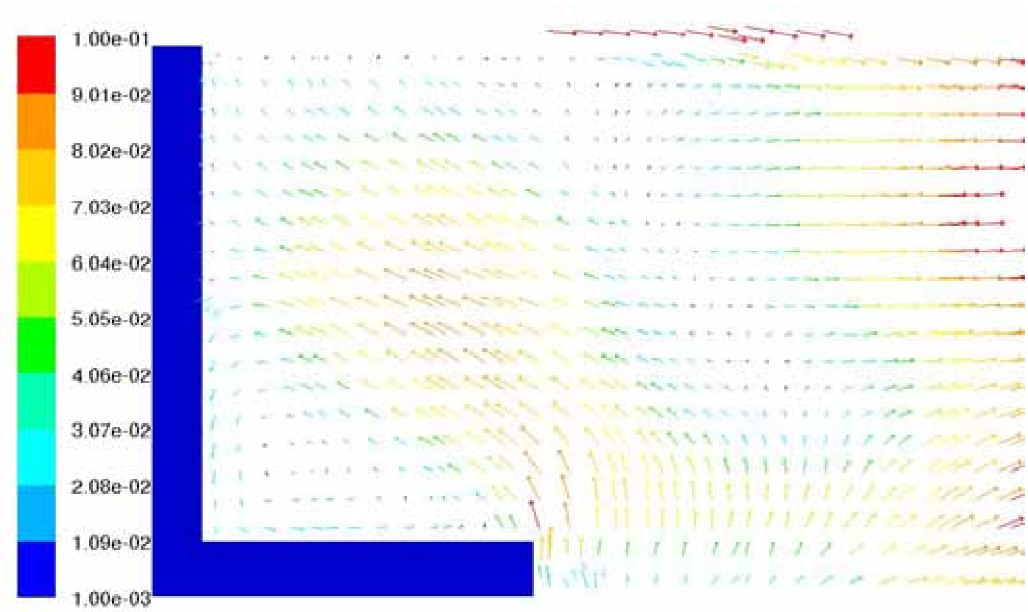
Figure 4 displays the average flow velocity, $\sqrt{u^2+w^2}$, in the center-plane $y/D=0$ behind the overlapping cylinder of different h, showing that a recirculation occurring behind the upper cylinder. Comparing with the results from the single circular cylinder of finite height (h=0)[14], where a large recirculation is generated behind the top corner of cylinder in the center-plane, the recirculation structure investigated in this study (h>0) is totally different due to the existence of lower large cylinder. The flows along circumference of lower cylinder meet together near the rear tip, generating a stream revealed by Figure 5 with relatively high velocity in wall-normal direction. This generated stream lifts until encounter the trailing vortex, and attempt to migrate to the region where the fluid has been rolled away by the Bound vortex. As a result, a recirculation region is formed. Since h=0.083 is too short to generate a strong lifting stream, a circulation is still formed in the downstream of top end of upper cylinder.

At the same Reynolds number, the size of the recirculation region is significantly dependent on h. When h=0.083D, a small recirculation takes place at the rear corner of upper cylinder, but it is too small to display clearly. With the increasing h, the dimensions of the recirculation grow quickly. With the condition of h=0.167D, the height and width of the recirculation region approximately reach 0.5H. A large scale recirculation almost occupies the entire corner behind the upper cylinder for h=0.25D.

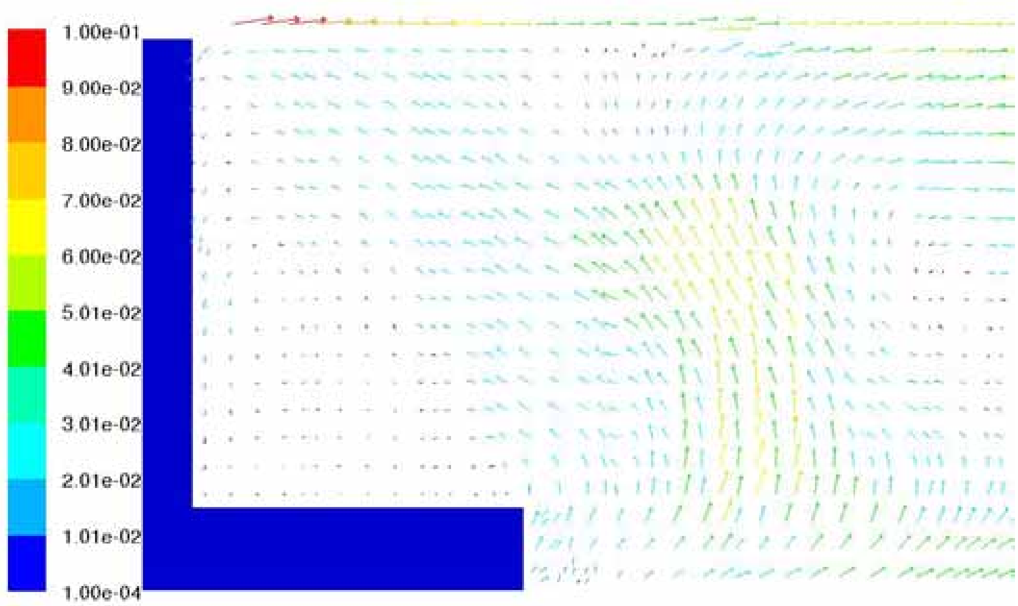
Another interesting feature is that the velocity magnitude of average flow behind overlapping cylinder of h=0.167D is much higher than those of h=0.083D and h=0.25D. This phenomenon may be ascribed to the stronger turbulent vortex generated near the top surface of lower cylinder, which is disadvantageous to cylinder structure.


1.00e-01
9.00e-02
8.00e-02
7.00e-02
6.00e-02
5.01e-02
4.01e-02
3.01e-02
2.01e-02
1.01e-02
1.00e-04

(a)


1.00e-01
9.01e-02
8.02e-02
7.03e-02
6.04e-02
5.05e-02
4.06e-02
3.07e-02
2.08e-02
1.09e-02
1.00e-03

(b)



(c)

Figure 4: The flow velocity $\sqrt{u^2 + w^2}$ of average flow behind the overlapping cylinder in the center-plane y/D=0: (a) h=0.083D; (b) h=0.167D; (c) h=0.25D. Unit is m/s.

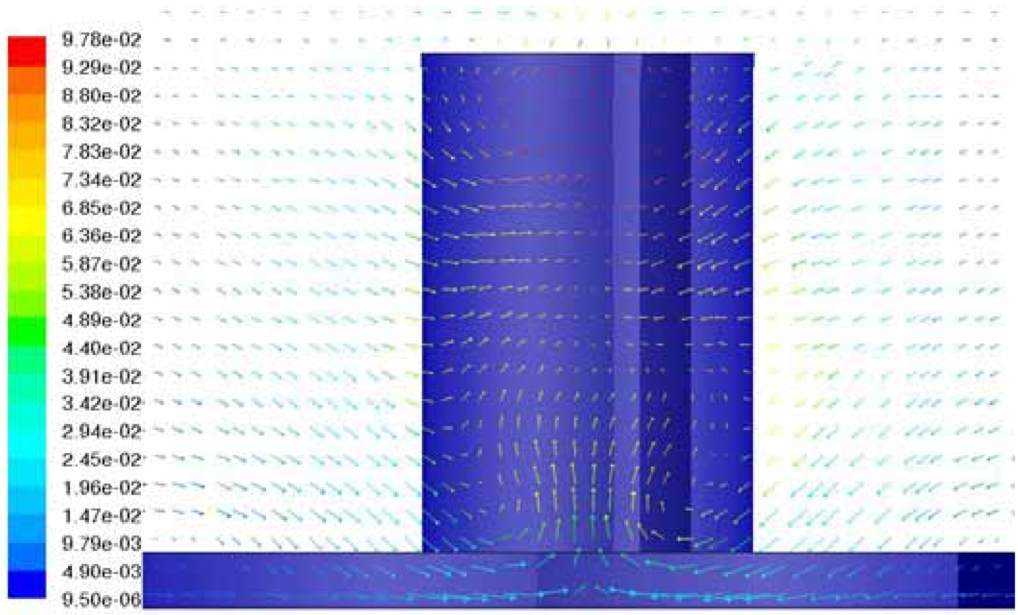


Figure 5: The secondary flow velocity $\sqrt{v^2 + w^2}$ of average flow behind the overlapping cylinder of h=0.167D at the x/D=5. Unit is m/s

3.3 Effect of inflow velocity on flow field

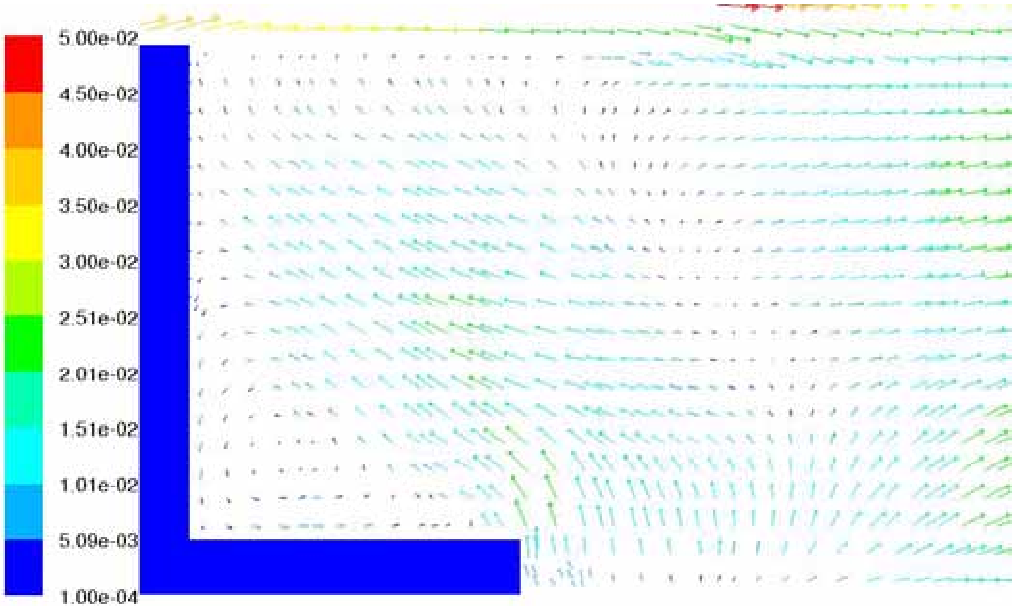
A series of simulations with different inflow velocity varying from 0.05 to 0.5m/s are implemented to study the effect of inflow velocity on flow structure. Figure 6 and 7 are typical examples of velocity $\sqrt{u^2+w^2}$ and $\sqrt{u^2+v^2}$ for h=0.167D and inflow U_0=0.05m/s and U_0=0.4m/s, respectively.

As discussed in section 3.2, a lifting stream shown in Figure 6 is generated near the rear tip of lower cylinder, and it is separated into two main components when encountering trailing vortex. One component migrates to recirculation region behind upper cylinder, while the other goes downstream along trailing vortex. Due to the interaction between flow with relatively high velocity from both sides of upper cylinder and upstream component of lifting stream, two recirculation regions, which are approximately symmetrical about center-plane y/D=0, are formed at horizontal plane z/D=1 shown in Figure 7.

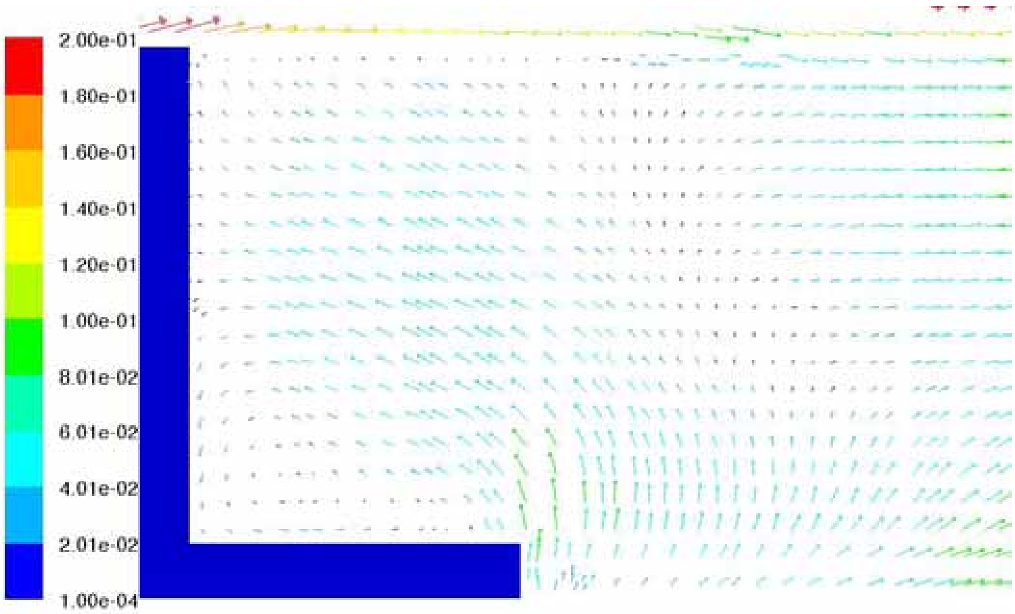
Within the Reynolds number carried out in this study, the increase of U_0 consequently leads to an augment of turbulent velocity behind overlapping cylinder, but the formation of recirculation region does not change significantly. The diameter of recirculation represented in Figure 6 is approximately 0.5H, which is similar to that shown in Figure 4 (b). In other words, compared with the deformation of recirculation region caused by h, the influence of U_0 is negligible.

4 CONCLUSION

A series of LES are performed to investigate the highly complex interaction between turbulent flow and the overlapping circular cylinder of finite height. The main features of the complex flow around this kind of bluff body are simulated and discussed. Laminar separation takes place at the front corner, and a trailing vortex is formed on the top due to the pressure difference between sidewall surface and top surface of upper cylinder. Because of the interaction between upstream component of lifting stream and flow from both sides of upper cylinder, two recirculation regions symmetrical about center-plane y/D=0 are formed at the horizontal plane z/D=1. Several vortex structures, including Necklace vortex near wall bottom and Bound vortex in the middle of upper cylinder, are also investigated. Influences of height of lower cylinder and inflow velocity on the flow structure are discussed. The results of average flow reveal that the formation of recirculation region behind the upper structure is significantly affected by the height of lower cylinder rather than inflow velocity in this study.

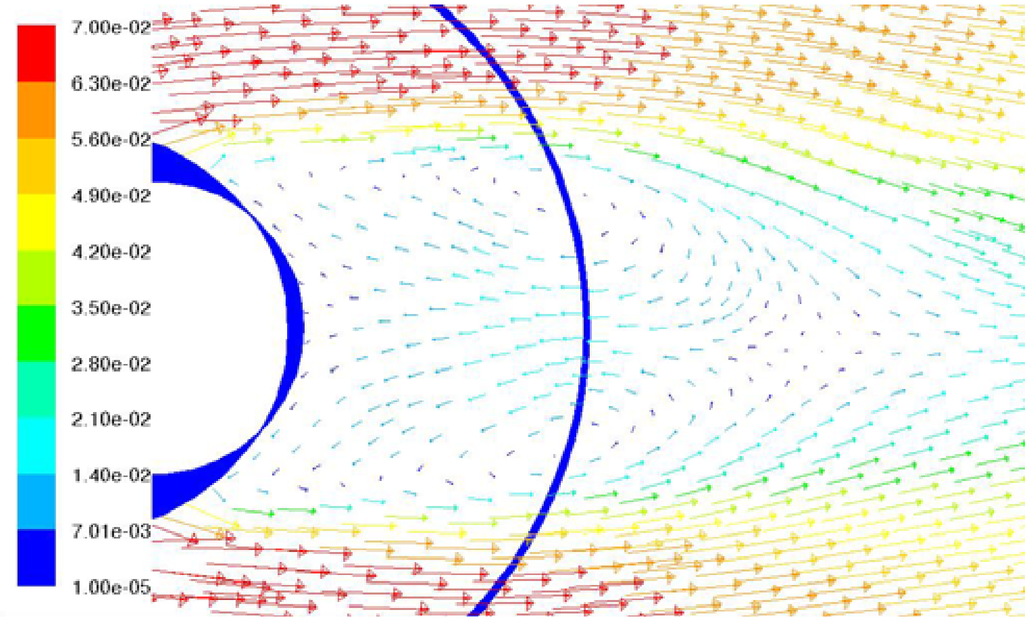


(a)

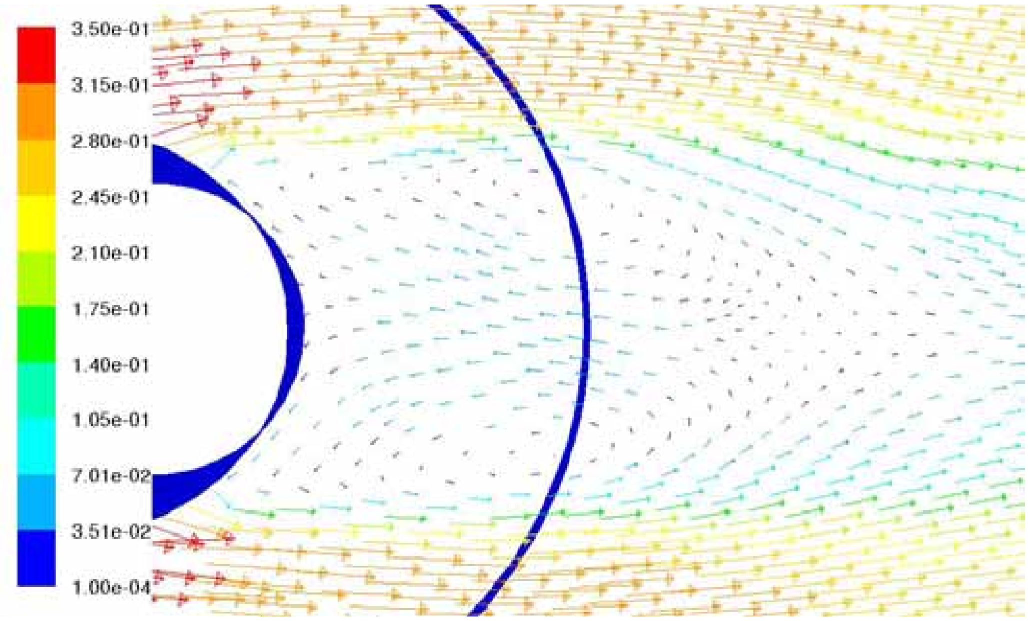


(b)

Figure 6: The flow velocity $\sqrt{u^2 + w^2}$ of average flow behind the overlapping cylinder in the center-plane y/D=0: (a) U_0=0.05m/s; (b) U_0=0.4m/s. Unit is m/s.



(a)



(b)

Figure 7: The flow velocity $\sqrt{u^2 + v^2}$ of average flow behind the overlapping cylinder at horizontal plane z/D=1: (a) U_0=0.05m/s; (b) U_0=0.4m/s. Unit is m/s.

ACKNOWLEDGEMENT: This study is supported by University of Aberdeen.

REFERENCES

[1] P.W. Bearman, M.J. Downie, J.M.R. GrahamM and E.D. Obasaju, "Force on cylinder in viscous oscillatory flows at low Kuelegan-Carpenter numbers." *J. Fluid Mech.*, 154: 337-352 (1985).

[2] C.H.M. Williamson, "Sinusoidal flow relative to circular cylinders." *J. Fluid Mech.*, 155: 141-174 (1985).

[3] T. Sarpkaya, "Force on a circular cylinder in viscous oscillatory flow at low Kuelegan-Carpenter numbers." *J. Fluid Mech.*, 165: 61-71 (1986).

[4] N. Fujisawa, K. Ikemoto and K. Nagaya, "Vortex shedding resonance from a rotationally oscillating cylinder." *J. Fluids Struct.*, 12: 1041-1053 (1998).

[5] D. Rocchi and A. Zasso, "Vortex shedding from a circular cylinder in a smooth and wired configuration: comparison between 3D LES simulation and experimental analysis." *J. Wind Eng. Ind. Aerodynam.*, 90: 475-489 (2002).

[6] M.H. Wu, C.Y. Wen, R.H. Yen, M.C. Weng and A.B. Wang, "Experimental and numerical study of the separation angle for flow around a circular cylinder at low Reynolds number." *J. Fluid Mech.*, 515: 233-260 (2004).

[7] M. Kappler, W. Rodi, S. Szepessy, and O. Badran, "Experiments on the flow past long circular in a shear flow." *Exp. Fluids*, 38: 269-284 (2005).

[8] R.D. Gabbai, and H. Benaroya, "An overview of modeling and experiments of vortex-induced vibration of circular cylinders." *J. Sound Vib.*, 282: 575-616 (2005).

[9] X. Sun and C. Dalton, "Application of the LES method to the oscillating flow past a circular cylinder." *J. Fluids Struct.*, 10: 851-872 (1996).

[10] R.P. Selvam, "Finite element modelling of flow around a circular cylinder using LES." *J. Wind Eng. Ind. Aerodynam.*, 67&68: 129-139 (1997).

[11] W. Rodi, J.H. Ferziger, M. Breuer and M. Pourquie, "Status of Large Eddy Simulation: Results of workshop." *J. Fluids Eng., Trans. SAME*, 119: 248-262 (1997).

[12] M. Breuer, "A challenging test case for large eddy simulation: high Reynolds number circular cylinder flow." *Int. J. HeatFluid Flow*, 21: 648-654 (2000).

[13] P. Catalano, M. Wang, G. Iaccarino and P. Moin, "Numercial simulation of the flow around a circular cylinder at high Reynolds numbers." *Int. J. Heat Fluid Flow*, 24: 463-469 (2003).

[14] J. Frohlich and W. Rodi, "LES of the flow around a circular cylinder of finite height." *Int. J. Heat Fluid Flow*, 25: 537-548 (2004).

[15] N. Fujisawa, Y. Asano, C. Arakawa and T. Hashimoto, "Computational and experimental study on flow around a rotationally oscillating circular cylinder in a uniform flow." *J. Wind Eng. Ind. Aerodynam.*, 93: 137-153 (2005).

[16] J.H. Ferziger and M. Peric, *Computational Methods for Fluid Dynamics*. Springer, Germany (1999).

[17] R. Franke and W. Rodi, "Calculation of vortex shedding past a square cylinder with various turbulence models." *Proceedings 8th Symposium on Turbulent Shear Flows*,

Munich, Germany. 189-204 (1991).
[18] W. Rodi, "DNS and LES of some engineering flows." *Fluid Dyn. Res.*, 38: 145-173 (2006).
[19] B. Cantwell and D. Coles, "An experimental study on entrainment and transport in the turbulent near wake of a circular cylinder." *J. Fluid Mech.*, 136: 321-374 (1983).