International Journal of Scientific and Technical Research in Engineering (IJSTRE)
www.ijstre.com Volume 4 Issue 5 ǁ Sep-Oct 2019.
ISSN: 2581-9941

Study of Cylinder Wake Vortex Fluidic Control based on Lattice Boltzmann Method

LipingSun[1], YanfangLv[1,*], WentingWang[2], Cheng Lu[3]
[1]*(Department of Ship BuildingEngineering,Harbin Engineering University, China)*
[2]*(The 703 Institute of China Shipbuilding Industry Corporation, China)*
[3]*(Department of Civil Engineering,Tianjin University, China)*

ABSTRACT: *When the flow past a cylinder, vortex shedding will occur in the near wake behind it. In order to suppress the vortex shedding of the circular cylinders, a numerical simulation program written by the Lattice Boltzmann Method was used based on the jet from the rear stagnation pointof cylinder. According to the numerical simulation of the jet with the same direction as the incoming flow, the effects of suppressing vortex shedding are analyzed. Results demonstrate that: (1) The correctness and validity of the main program and the expansion moduleare verified by simulating the flow around the square cavity and the cylinder. (2) With the increasing of the jet velocity, the effect of the jeton the suppression phenomenon for the cylinder's wake vortex becomes more obviously. Additionally, the fluctuation of the transverse force of the circular cylinder is reduced. (3) Although the method of jet flow can suppress the cylindrical vortex shedding, it needs to control the jet velocity. In addition, it is easy to cause the large value of the drag coefficient of the cylinder and lead to the structural risk if the velocity of jet flow is too high.*

KEYWORDS *- Lattice BoltzmannMethod, Multi-grid, Cylindrical flow, Jet, Suppress vortex shedding.*

I. Introduction

The Flow-Induced Motion (FIM) is a common phenomenon in the marine engineering. And Vortex-Induced Vibration (VIV), which is one of the most well-known phenomenon of FIM, will be developed under the alternating load applied by vortex shedding and will cause fatigue damage of the structures [1]-[2]. The VIV developed frequently in the marine applications such as risers, offshore platforms, bridges and other structures [3]-[4]. Once the structural damage formed, the immeasurable casualties and property losses will be caused. Based on the powerful destruction of VIV, many scholars have focused their attention on the suppression of vortex shedding and have proposed various measures to prevent vortex shedding.

According to the necessity of the energy's input in the process of the vortex shedding, the main methods can be divided into the passive control method and the active control method [5]-[7]. In the recent decades, various methods of passive control method including the devises of helical strakes [8]-[9], the control columns [10]-[11], ribbons [12]-[13] and spoilers [14]-[15] have been proposed by many scholars. Although there are many useful methods to suppress vortex shedding by passive control methods, active control methods are more convenient than that of the former because the optimum effect can be obtained. Active control is a method which should input the external energy to suppress vortex shedding, thus, changing the motion state of the flow field. According to the literature, these methods fall into some categories: (1) Jet Method, which adds injection devices or numerical injection to suppress vortex shedding. (2) MSBC, which suppress the vortex shedding by injecting the momentum in the cylinder's boundary layer. (3) Rotating-sharking Method, which rotates the structure. But the first jet method, here, will be the only consideration. In the recent years, considerable number of scholars focused on suppressing the vortex shedding by the jet flow based on the experimental and numerical method [16]-[19]. Nevertheless, classical experimental method[20] cannot get a more precise result due to the influences and constrains of human factors and physical model. Comparing to the experimental method, the numerical method has gradually become a widely received method to study the problems of hydrodynamic based on its low cost and high precision. However, the numerical method such as using the Navier-Stokes equations and Microscopic Molecular Dynamics, which need a large amount of computational storage and have the extremely long computing time. The Lattice Boltzmann Method (LBM) has many advantages including high operating efficiency, better robustness, excellent parallel performance, lower computational storage and the easily handle boundary. According to the above reasons, this paper employs the LBM to research the cylinder wake vortex fluidic control.

The purpose of this paper is to explore the impact of rear stagnation point jet flow on the wake shedding and force condition of original cylinder by developing a numerical simulation program based on the LBM. According to the results of the numerical simulation, the hydrodynamic parameters such as drag coefficients, lift coefficients and shedding vortex have been analyzed.

II. Numerical simulation

2.1. Research method

The Multi-Relaxation-Time Lattice Boltzmann Method (MRT-LBM) [21]-[23] has been used in this paper. The evolution equation can be presented in equation(1):

$$f_i(x+\vec{e}_i\cdot\delta t, t+\delta t) - f_i(x,t) = M^{-1}S[m^{eq}(x,t) - m(x,t)] \quad (1)$$

Where f is the distribution function, S is a relaxation square matrix; where $m^{eq}(x,t)$ expresses the equilibrium state of the moment function.

By combining the MRT-LBM with large eddy simulation (LES) [24]-[26], the flow around the cylinder can be investigated in turbulent mode. The LES employed the Smagorinsky model, which is used to deal with turbulence near the wall, can make the turbulence here satisfy the assumption of isotropy.

A new optimal scheme, which improves the original boundary structure of the thick and thin grids of the multi-block grid, is adopted to make the spatial-temporal multiple interpolation of the original multi-block grid method into a single spatial multi-point interpolation. This method solves the problems including the complex proceduce, waste memory footprint and the accumulation of the interpolation error. The alternating form of the configuration between the original thick and thin grids has been improved.

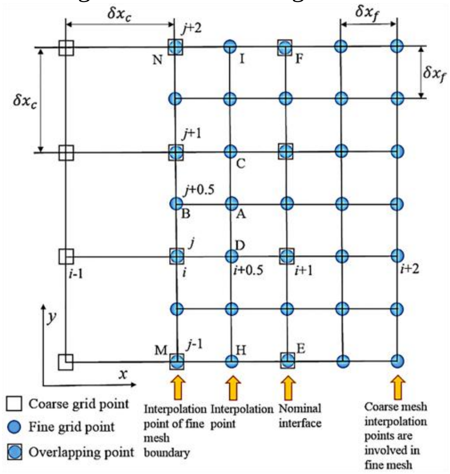

Fig.1: Schematic of overlapping spaces of the grids

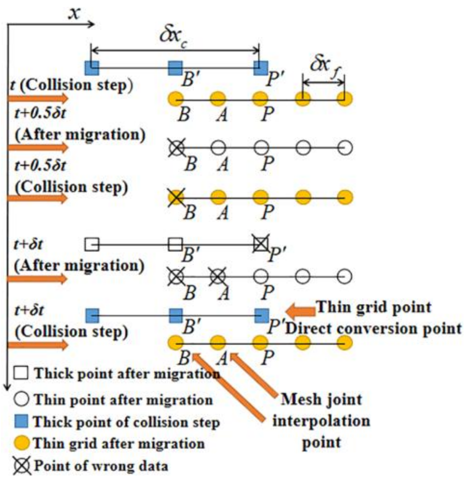

Fig. 2: Schematic of time advancement of the grids

In the overlapping region, the fine grid points have not corresponding thick grid points such as A, B, C and D, so it needs to be spatially interpolated, as shown in Fig. 1. The 16-point Binary Lagrangian Interpolation has been used to ensure the accuracy of interpolation. The equations can be presented as following:

$$\mathrm{f}(x,y) = \sum_{i=1}^{m}\sum_{j=1}^{n}\left[\left(\prod_{\substack{a=1\\a\neq i}}^{m}\frac{x-x_a}{x_i-x_a}\right)\cdot\left(\prod_{\substack{b=1\\b\neq j}}^{n}\frac{y-y_b}{y_j-y_b}\right)\cdot f(x_i,y_j)\right] \quad (2)$$

The Fig. 2 is the case of n = 2. For the configuration structure of n> 2, the continuity can be guaranteed only by evenly dividing the number of n-1 grid points between BP, interpolating these points and adding n-1 collision migration operation in the fine grid.

2.2. Calculation model

The LBM is used to simulate two-dimensional flow over a stationary circular cylinder with jet flow. The coordinate system of this paper is established as shown in Fig. 3. The x and y represent the directions of stream-wise and span-wise, respectively. Where D isthe diameter of the cylinder and h is the diameter of the jet hole, $U$is the velocity of the incoming flow, and $U_j$is the jet velocity in the range of $0.0U \leq U_j \leq 10.0U$. The location of the jet flow is situated at the rear stagnation point and the jet flow has the same direction with incoming flow. The free-lip boundary condition is applied on the top and bottom lids. The left inlet and right outlet are set as the characteristic and relative velocities, respectively, and keep the gradient of pressure at zero. According to the complex boundary formula, the wall surface of cylinder is set as no-slip boundary.

The C_D is the drag coefficient, C_L is the lift coefficient. The equations can be presented in equation(3) and equation(4) as:

$$C_D = \frac{F_x}{0.5\rho U^2 D} \tag{3}$$

$$C_L = \frac{F_y}{0.5\rho U^2 D} \tag{4}$$

Where F_x and $F_y$are the force of the downstream and cross flow directions, respectively. And ρ is the density of the flow.

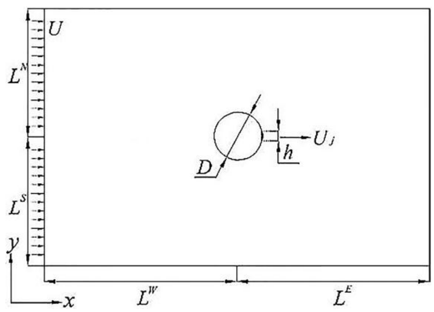

Fig. 3: Schematic diagram of jet flow of cylinder

2.3. Implementation and verification of the program

The improved multi-block grid method was embedded into the basic program by adding self-loop module. The large eddy model of Smagorinsky model was introduced into the computer program by adding turbulence module to the collision module. The rear stagnation point jet has been simulated by adding the jet module to the migration module.Using the OpenMP to compile the processing scheme, rewrite the program code, achieve parallel computing, improve computing efficiency and save computing time. The output module has been used to realize the connection between the program and the post-processing software of the flow field, and then realize the visualization of the flow field. The Fig. 4 is the flow chart of the main program.

The feasibility of method was verified by the square cavity around the flow and the unsteady cylindrical flow around. In order to verify the correctness of the program code which used the improved multi-block grid module, a classical square cavity model and a circular cylinder have been used to simulate the laminar state at low Re. The results from Fig. 5 and Fig. 6 show that the standard solution of the numerical simulation results are in good agreement with the literature data [27]. In order to verify the correctness of the program code which added the Smagorinsky model, a circular cylinder has been used to simulate the turbulence state at Re = 3900. The results from Table 1 show a good fit to the literature [28]-[29]. To sum up, this research method is feasible.

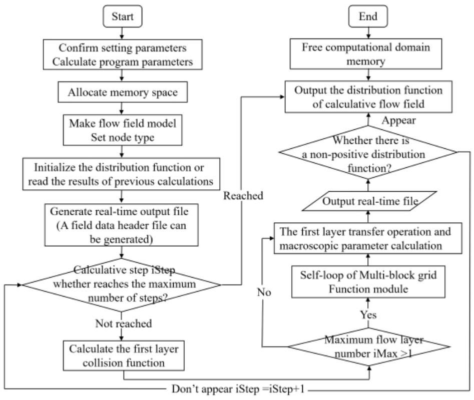

Fig. 4: The flow chart of the main program

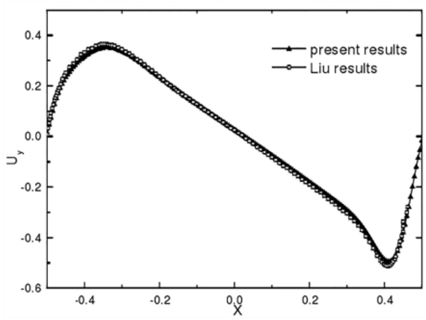

Fig. 5: The velocity distribution in X Direction

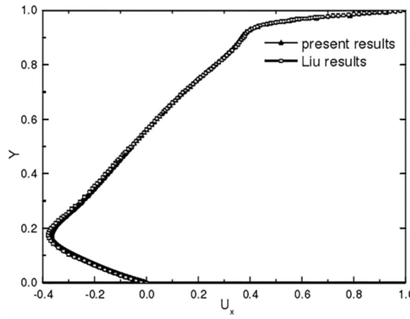

Fig. 6: The velocity distribution in Y Direction

Table 1. The comparative analysis between literature [28]-[29] and this paper at *Re*=3900.

Article	Method	Cd-Mean	Cl-RMS	St
This paper	LES	1.396	1.097	0.226
Reference [28]	URANS	1.330	-	0.210
Reference [29]	URANS	1.590	1.170	0.235

III. Results and Analysis

The main assignment of this paper is to analyze the phenomenon of vortex shedding by injecting momentum from the rear stagnation point of the cylinder and is to study the hydrodynamic parameters such as drag and lift coefficients at different *Re*.

3.1. The analysis of drag and lift coefficients

For *Re* = 200 and 300, the state of flow field is simulated without adding the turbulence module due to the leading function of the viscous force. For *Re* = 10000, there is a turbulent state around the cylinder, so the turbulence module is considered for simulation.

With the increasing of the jet velocity, the mean value of drag coefficients show the downward trend in Fig. 7 and Fig. 8. The sign of the resistance changes as the U_j changing value from 5U to 6U. The main reason of this phenomenon is that the jet exerts a reaction force on the cylinder, furthermore, the dimensionless coefficient (*Cs*) of jet recoil is quadratic correlation with jet velocity. Additionally, when $U_j \geq 8.0U$,the absolute value of the drag coefficients, which is under the influence of the jet, will greater than that of the case without jet flow. Therefore, the effect of jet recoil on the cylinder should be emphasized in the practical application.The extreme fluctuation of lift coefficients (*Cl-Max*) at different jet velocities and the corresponding root mean square of lift coefficients (*Cl-RMS*) are shown in Fig. 9 and Fig. 10. The curves show the downward trend as the jet velocities increase in the certain range. The reason for this phenomenon is that the jet flow compensates for the flow loss in the vortex region of the cylinder and improves the stability of the flow field.

In the range of $U_j \geq U_{jcrit1}$, the value of the extreme fluctuation and root mean square for lift coefficients both begin to rapidly decline. In the case of *Re* = 200 and 10000, which have the same critical jet velocity of U_{jcrit1} = 4.0. In the case of *Re* = 300, resulting in U_{jcrit1} = 5.0. The main reason for the suddenly drop is that the jet flow separates the wake field and suppresses the interaction of the wake vortex. In the range of $U_j \geq U_{jcrit2}$, as the jet velocities increase, the value of lift coefficients decreases gently. Where U_{jcrit2}= 5.0 at *Re* = 200 and 10000 and U_{jcrit2}= 6.0 at *Re* = 300.

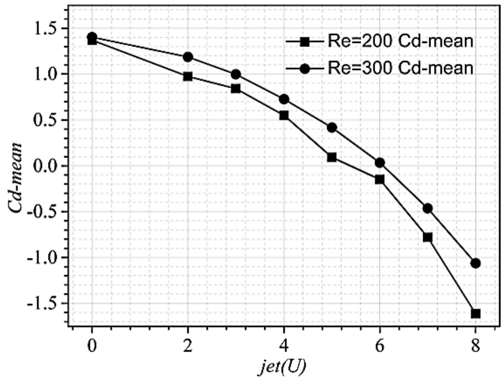

Fig. 7: Mean value of drag coefficients at *Re*=200 and 300

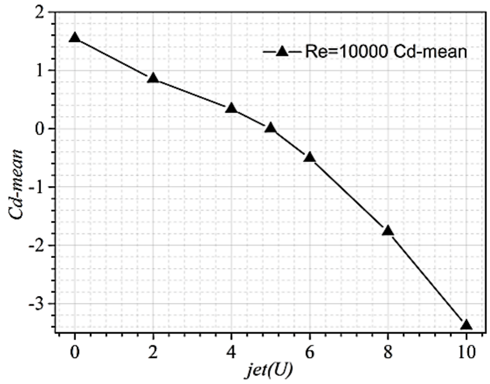

Fig. 8: Mean value of drag coefficients at *Re*=10000

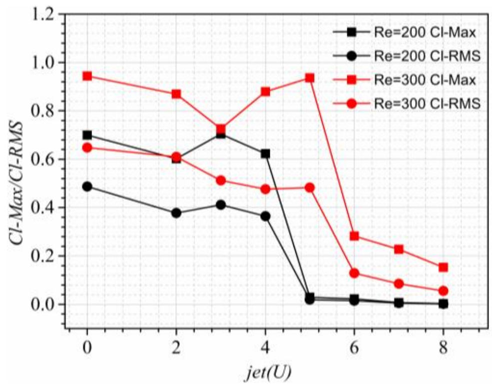

Fig. 9: The diagram of extreme fluctuations and root mean square for lift coefficients at *Re*=200 and 300

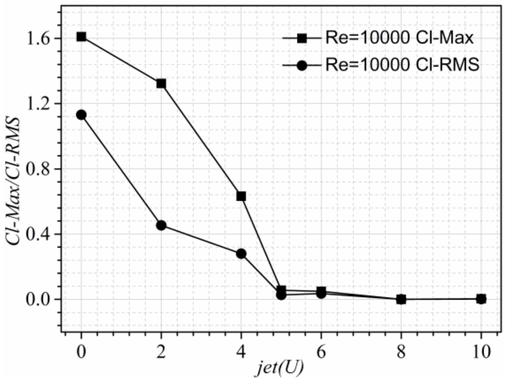

Fig. 10: The diagram of extreme fluctuations and root mean square for lift coefficients at *Re*=10000

3.2. The analysis of the flow field

Some typical jet velocities have been selected based on the results presented in Fig. 9 and Fig. 10. It is notable that the curves fall sharply when the value of U_j varies from 4*U* to 5*U*. Therefore, it is used to the further study. In the range of $U_j \geq 8U$, the situation of the flow is steady, so it is included in the analysis. The jet velocities are 0.0*U*, 4.0*U*, 5.0*U* and 8.0*U* from left to right in the Fig. 11-Fig. 14.

3.2.1. The analysis of velocity field

As shown in Fig. 11, for*Re* = 200, the results indicate that the jet flow produces a separated function for the wake flow field, and this phenomenon become obviously with the increasing of the jet velocity. The wake vortex of the cylinder, which is formed near the jet flow, has the opposite direction compared to the jet flow as the jet velocity reaching a certain value, indicating that the jet has a suppression effect for the wake vortex of cylinder. In the meantime, increasing of the distance between the jet flow and rear stagnation point is accompanied by the dissipation of hydrodynamic power of the jet flow. The jet velocity decreases as the distance increases and the length of the stable region increases as the jet velocity increases. The location of the instability and the length of the stable region will affect the vortex state of the original cylinder and the transverse force of the cylinder.

As shown in Fig. 12, for*Re* = 10000, the diagrams reveal similar phenomenon to the case of low *Re*. However, in the case of U_j= 8*U*, the unstable phenomenon of jet flow has not appeared. It is speculated that the jet flow is far away from the cylinder wake area when the jet kinetic energy is exhausted. And the inertia plays a dominant role for the shaping of flow state but the influence of viscous force is small. Therefore, the jet flow can continue to maintain a long distance and can be not easy to be unstable.

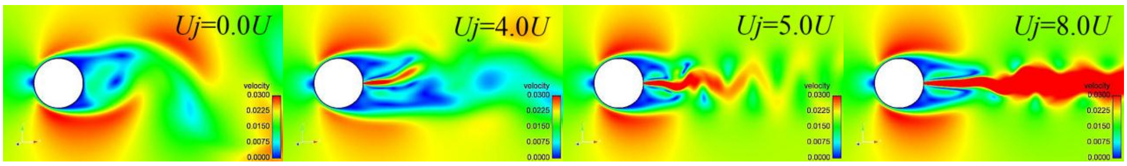

Fig. 11: The comparison diagram of velocity at *Re* = 200

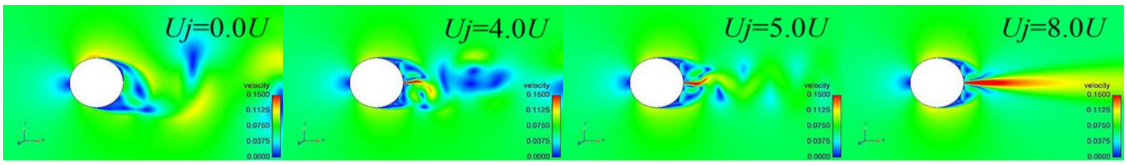

Fig. 12: The comparison diagram of velocity at *Re* = 10000

3.2.2. The analysis of vorticity field

As shown in Fig. 13, for*Re* = 200, in the case of U_j<4*U*, the wake vortex of the cylinder has been stimulated due to the short length of the jet flow stable region. As U_j increases in the range of $4U \leq U_j \leq 5U$,based on the phenomenon that the length of the stable regionhas thegreatervaluethan that of the main region of the cylinder wake vortex, the cylinder wake vortex has not been stimulated. In the range of U_j>5*U*, the separated function of jet flow becomes more obviously, which induces a pair of anti-phase symmetric wake vortex and suppresses the shedding of the cylinder wake vortex. It should be noted that, although the vortex of jet flow has been formed in the wake jet flow, it causes a little influence on the transverse force of the cylinder because of the far distance between the area of the alternating vortex and the wall of cylinder. It can be seen that the stretch of the cylinder wake vortex is not obvious in the case of U_j = 5U but obvious in the case of U_j = 8U.

As shown in Fig. 14, for*Re* = 10000, the results of this case is similar to the case of low *Re*. And the suppression phenomenon of the cylinder wake vortex is more obviously due to the larger power of the jet flow. Therefore, as the jet velocity reaching up to the value of U_j= 4U, although the cylinder wake vortex will still be stimulated because of the close unsteady position of the jet flow, the transverse force of the cylinder will also be significantly weakened due to the large suppression effect of the jet flow.

According to the analysis in the cases of *Re* = 200 and 10000, the results show that the jet flow on the rear stagnation point can suppress the cylinder wake vortex obviously. And it is more appropriate to select the jet velocity in the range of $6U \le U_j \le 8U$ after considering the jet recoil and suppression effect comprehensively.

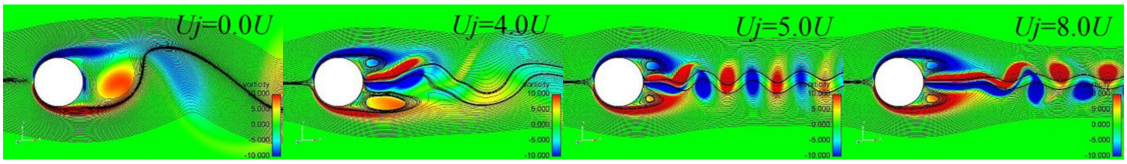

Fig. 13: The comparison diagram of vorticity at *Re*=200

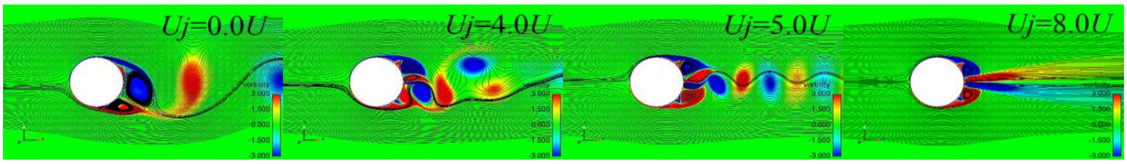

Fig. 14: The comparison diagram of vorticity at *Re*=10000

IV. Conclusion

In this paper, a numerical simulation program, which was developed by Fortran based on LBM, was applied to the research of the influence of cylinder which added the jet flow. And the main conclusions are obtained as follows:

With the increasing of the jet flow velocity, the cylinder lift coefficients (such as the extreme fluctuation and the root mean square) show a downtown tendency. Particularly, when the jet velocity reaches up to a critical value, the value of lift coefficients decreases gently and closes to zero. Nevertheless, the results confirm that it needs to control the jet velocity. If the velocity of jet flow is too high, it is easy to cause the large value of the drag coefficient (resulting in the structural risk).

The influence of jet flow on the rear stagnation point for the cylinder wake vortex are mainly reflected in two aspects: the jet flow, which develops a separated function for the cylinder wake field, suppresses the vortex shedding by preventing the interaction between the wake vortex of the original cylinder; the jet flow compensates the velocity loss in the area of cylinder wake vortex, thus increasing the stability of flow field and suppressing the original cylinder wake vortex. The combined action of these two aspects can restrain the vortex shedding and weaken the transverse force fluctuation of the cylinder.

Reference

[1] C.H.K. Williamson andR. Govardhan,Vortex-induced vibration,*Annual Review of Fluid Mechanics, 36*, 2004,413-455.

[2] R. Govardhan andC.H.K. Willliamson, Modes of vortex formation and frequency response of a freely vibrating cylinder, *Journal of Fluid Mechanics*, *420*, 2000,85-130.

[3] M. Lou, *Experimental study and numerical simulation of vortex-induced vibration of Marine transport riser*, doctoral diss., Ocean University of China, Qingdao, Shandong, 2007.

[4] T.C. Lu, *Numerical simulation of vortex-induced vibration of marine riser*,masteral diss., Harbin Institute of Technology, Harbin, Heilongjiang, 2014.

[5] H. Choi, W.P.Jeon, J. Kim, Control of flow over a bluff body,*Annual Review of Fluid Mechanics*, *40*,2008, 113-139.

[6] S.Kumar, B.Gonzalez, O. Probst, Flow past two rotating cylinders,*Physics of Fluids*, *23(014102)*, 2011.

[7] S.Muddada andB.S.V.Patnaik, An active flow control strategy for the suppression of vortex structures behind a circular cylinder,*European Journal of Mechanics B/Fluids*, *29*, 2010, 93-104.

[8] L.R. Wooton andD.E. Walshe, Preventing wind-induced oscillations of structures of circular section, *Proceedings of the Institution of Civil Engineers*, *47(1)*, 1970, 1-24.

[9] M.Branković andP.W.Bearman, Measurements of transverse forces on circular cylinders undergoing vortex-induced vibration,*Journal of fluids and structures*,*22(6)*, 2006, 829-836.
[10] F. Lesage, A method of reducing drag fluctuating side force on bluff bodies, *Journal of Wind Engineering and Industrial Aerodynamics*, *25(2)*, 1987, 229-245.
[11] S.Mittal and A.Raghuvanshi, Control of vortex shedding behind circular cylinder for flows at low Reynolds numbers,*International Journal for Numerical Methods in Fluids*, *35(4)*, 2001, 421-447.
[12] R.D. Blevins,*Flow Induced Vibration* (NYC: Van Nostrand Reinhold, 1977).
[13] S.H.Kwon,J.W.Cho, J.S.Park, H.S.Choi, The effects of drag reduction by ribbons attached to cylindrical pipes,*Ocean Engineering*, *29(15)*, 2002, 1945-1958.
[14] P.K.Stansby, J.N.Pinchbeck, T.Henderson, Spoilers for the suppression of vortex-induced oscillations (Technical Note),*Applied Ocean Research*, *8(3)*, 1986, 169-173.
[15] C.P. Shao, X.Q. E, Q.D. Wei, F.R. Zhu,Control of vortex shedding moderate Reynolds numbers,*Theoretical and Applied Mechanics*,*34(4)*, 2002, 609-615.
[16] L.H. Feng andJ.J. Wang,Circular cylinder vortex-synchronization control with a synthetic jet positioned at the rear stagnation point,*Journal of Fluid Mechanics*, *662*, 2010, 232–259.
[17] L.H. Feng,J.J. Wang, C. Pan, Effect of novel synthetic jet on wake vortex shedding modes of a circular cylinder,*Journal of Fluids Structure*, *26*, 2010, 900–917.
[18] L.H.Feng and J.J.Wang, Synthetic jet control of separation in the flow over a circular cylinder, *Experiments in Fluids*, *53(2)*, 2012, 467-480.
[19] Y.Qin, Y.P.Song, F.Chen, R.Y.Wang, H.P.Liu,Active flow control by means of end-wall synthetic jet on a high-speed compressor stator cascade,*Institution of Mechanical engineers*, *232(6)*, 2018, 641-659.
[20] Y.J. Chen and C.P. Shao, Suppression of vortex shedding from an oscillating cylinder by base blowing,*Scientia Sinica:Physica, Mechanica&Astronomica*, *42(4)*, 2012, 406-420.
[21] Z.L.Guo, *The principle and application of Lattice Boltzmann Method* (Beijing: Science Press, 2009).
[22] S.Chen and G.D. Doolen, Lattice Boltzmann Method for Fluid Flows,*Annual Review of Fluid Mechanics*, *30(1)*, 2012, 329-364.
[23] S.Tao, *The application of the Lattice Boltzmann Method in computational fluid dynamics*,masteral diss., Dalian University of Technology, Dalian, Liaoning, 2013.
[24] W.Cheng, D.I.Pullin, R.Samtaney, Large-eddy simulation of flow over a grooved cylinder up to transcritical Reynolds numbers,*Journal of Fluid Mechanics*,*835*, 2008, 327-362.
[25] A.Dmitry, S.Lysenko·Ivar, E.R. Ertesvåg·Kjell,Large-Eddy Simulation of the Flow over a Circular Cylinder at Reynolds Number 3900 Using the OpenFOAM Toolbox,*Flow Turbulence Combust*, *89*, 2012, 491-518.
[26] Z.S.Zhang,*Theory and application of numerical simulation of large eddy in turbulent flow* (Beijing: THU Press, 2008).
[27] S. Mittal and R. Balachandar, On the inclusion of the dimensional effects in simulations of two-dimensional bluff body wake flows, *Proc. the 1997 ASME Fluids Engineering Division Summer Meeting*, 1997, 1-6.
[28] E. Palkin, R. Mullyadzhanov, K. Hanjalic, International Conference on Mechanical, Automobile and Robotics Engineering: URANS and LES of a flow over a cylinder at Re= 3900, Singapore, 2014.
[29] M.E. Young and A. Ooi, Comparative Assessment of LES and URANS for Flow Over a Cylinder at a Reynolds Number of 3900, *Proc.16th Australasian Fluid Mechanics Conference*, Crown Plaza, Gold Coast, Australia, 2007, 1063-1070.