Large Eddy Simulation of the Vortex-Induced Vibration of a Circular Cylinder by using the Local Domain-Free Discretization Method

Tianmei Pu¹, Yang Zhang² and Chunhua Zhou^{3,*}

 ¹ College of General Aviation and Flight, Nanjing University of Aeronautics and Astronautics, Nanjing, Jiangsu 210016, China
 ² College of Astronautics, Nanjing University of Aeronautics and Astronautics, Nanjing, Jiangsu 210016, China
 ³ Department of Aerodynamics, Nanjing University of Aeronautics and Astronautics, Nanjing, Jiangsu 210016, China

Received 9 June 2021; Accepted (in revised version) 23 January 2022

Abstract. In this paper, the local domain-free discretization (DFD) method is extended to large eddy simulation (LES) of fluid-structure interaction and the vortex-induced vibration (VIV) of an elastically mounted rigid circular cylinder, which is held in the middle of a straight channel, is numerically investigated. The wall model based on the simplified turbulent boundary layer equations is employed to alleviate the requirement of mesh resolution in the near-wall region. The ability of the method for fluid-structure interaction is demonstrated by simulating flows over a circular cylinder undergoing VIV. The cylinder is neutrally buoyant with a reduced mass $m^* = 11$ and has a low damping ratio $\zeta = 0.001$. The numerical experiment of the VIV of a cylinder in an unbounded flow shows that the present LES-DFD method is more accurate and reliable than the referenced RANS and DES methods. For the cylinder in the middle of a straight channel, the effect of the channel height ($d^* = d/D$) is investigated. The variations of the response amplitude, vortex-shedding pattern and the length of the induced separation zone in the channel boundary layers with the channel height are presented.

AMS subject classifications: 74F10, 76F65

Key words: Immersed boundary method, domain-free discretization, large-eddy simulation, fluid-structure interaction, vortex-induced vibration.

1 Introduction

For the numerical investigation of fluid-structure interactions (FSI), transient re-meshing strategies, such as grid deformation/regeneration techniques, are usually required in

*Corresponding author. *Email:* chzhou@nuaa.edu.cn (C. Zhou)

http://www.global-sci.org/aamm

1456

©2022 Global Science Press

boundary-conforming methods, which are time-consuming and increase the algorithmic complexity. Transient re-meshing strategies may work well in the framework of Reynolds-Averaged-Navier-Stokes (RANS), but typically lack accuracy when being coupled with eddy-resolved methodologies. The immersed boundary (IB) method, which can solve moving-boundary problems on a fixed mesh, is an alternative approach.

The IB method was firstly proposed by Peskin [1] to investigate FSI in the cardiovascular circulation. So far, many amendments have been proposed with the aim of improving the stability and the applicability of this method [2,3]. The fundamentals and the recent applications to simulations of complex fluid-structure-interaction (FSI) problems are reviewed by Sotiropoulos and Yang [4] and Huang and Tian [5]. In [6,7], Shu et al. proposed an IB method, named the domain-free discretization method (DFD), to solve partial differential equations (PDEs) on irregular domains. In the DFD method, the discrete form of a PDE at an interior node in the immediate vicinity of the IB may involve some exterior nodes. In the original DFD method, the functional values at an exterior dependent node are evaluated along the whole mesh line, so it is not applicable for complex domains. To make the method more general, a local DFD method was developed by Zhou et al. [8], in which the functional values at the exterior dependent nodes are obtained by using a proper local extrapolation along the direction normal to the wall and in conjunction with the boundary conditions. The local DFD method has been successfully applied to simulate various inviscid or laminar flows [8,9]. Recently, it was extended to RANS simulation [10] and large eddy simulation (LES) [11] of turbulent flows by introducing the wall modeling techniques to alleviate the requirement of mesh resolution for the boundary layer.

Vortex-induced vibration (VIV) is a popular topic in the FSI field. This topic elicited the attention of researchers after the dramatic collapse of the Tacoma Narrows Bridge in 1940. The VIV phenomenon involves complicated physical mechanisms and cannot be despised in design of longer and slender structures, such as skyscrapers, bridges, and chimneys.

Most previous VIV studies are focused on the paradigm of a freely vibrating, elastically mounted rigid cylinder placed in a uniform and unbounded cross-flow. In the review of Williamson and Govardhan [12], the importance of several dimensionless variables are highlighted, including the mass ratio (m^*), the damping ratio (ζ), and the reduced velocity (U_r). Khalak and Williamson [13] reported that the amplitude response of a one-degree-of-freedom (one-DOF) VIV system can be categorized into two types, according to its mass-damping ratio ($m^*\zeta$). With a lower $m^*\zeta$, there are three distinct branches in the response curve with the variation of reduced velocity. The three branches are termed as the "initial", "upper" and "lower" branches. With a higher $m^*\zeta$, the upper branch does not exist.

To date, only a few studies have been dedicated to identifying the effect of a single plane wall in the vicinity of the elastically mounted cylinder. In this case, the cylinder is immersed in a semi-infinite flow and the dynamics of VIV is much more complex. The gap ratio, $S^* = S/D$, defined as the distance between the cylinder bottom and the

wall boundary in the equilibrium condition and normalized by the cylinder diameter D, is one of the dominant parameters of interest. Chung [14] numerically investigated the VIV of a cylinder in a semi-infinite flow under laminar vortex shedding conditions and observed that the maximum amplitude of vibration decreases as S^* decreases. In the work of Daneshvar and Morton [15], the VIV of a circular cylinder with a small m^* is investigated experimentally under turbulent conditions and it was revealed that, for $S^* < 3$, the amplitude of vibration decreases monotonically until $S^* \approx 0.5$, where the cylinder begins to periodically impact the wall.

Most of the numerical investigations of the VIV of a circular cylinder in the unbounded or semi-infinite flows are restricted to low Reynolds numbers ($Re=10-10^3$) [14, 16,17], in which many mechanisms were illustrated and explored. For the cylinder VIV problems at higher Reynolds numbers in the range of $10^4 - 10^6$, most of the current numerical simulations are conducted by the two-dimensional RANS simulation and the resolution of flow features is not sufficient. In the work of Pan et al. [18] for the VIV of an elastically mounted rigid circular cylinder, a two-dimensional RANS code equipped with the SST $k-\omega$ turbulence model is used. Although vortex mode and transition in the branches agree well with the referenced experimental results, there are some discrepancies in the result of the maximum amplitude in the upper branch. More recently, khan et al. [19] conducted the RANS simulation of the VIV of a cylinder and the maximum amplitude is underestimated considerably. The challenge of capturing the maximum oscillation amplitude has been discussed in [20, 21]. The influence of the spanwise turbulence in the wake is omitted when doing a two-dimensional simulation. As shown in the earlier work of Rosetti et al. [22], RANS models do not provide sufficient resolution for flows over a cylinder, especially for high Reynolds numbers, due to their intrinsic properties of isotropic eddy viscosities and homogeneous Reynolds stresses.

With the continuous progress of computer technology, the trend of numerical investigation of the VIV of circular cylinder is diverted to LES. Jus et al. [23] investigated the feasibility and accuracy of LES of the cylinder VIV and showed the ability of LES to reproduce the mechanism responsible for energy exchanges involved in the interaction between flow and moving boundary of the cylinder. In the work of Pastrana et al. [24], the VIV of a low-mass-ratio two-DOF circular cylinder at subcritical Reynolds numbers was investigated by LES. The results of the maximum amplitude and frequency of motion in both directions are in good agreement with the referenced experimental data.

In this work, the DFD method is extended to LES of FSI and the VIV of a cylinder held in the middle of a straight channel is numerically investigated. For the reminder of the paper, the content is arranged as follows. In Section 2, the governing equations and numerical approximations are briefly presented. The treatment of the immersed boundary and the wall modeling technique in LES are discussed in Section 3. Section 4 presents the algorithms for coupling of FSI. In Section 5, VIV of a cylinder in an unbounded flow is simulated to verify the accuracy and efficiency of the LES-DFD method for FSI problems. The effects of the channel boundaries on the VIV characteristics of the cylinder are also investigated in this section. Finally, summary and conclusion are given in Section 6.

2 Governing equations and numerical approximations

An incompressible fluid with constant density and viscosity is considered in this paper. Employing a subgrid-scale (SGS) model, the filtered N-S equations in non-dimensional form can be written as

$$\mathbf{I}^{m} \cdot \frac{\partial \mathbf{w}}{\partial t} + \frac{\partial \mathbf{f}_{i}}{\partial x_{i}} = \frac{\partial \mathbf{g}_{i}}{\partial x_{i}}, \tag{2.1}$$

where \mathbf{w} , \mathbf{f}_i and \mathbf{g}_i are the vectors of filtered flow variables, convective fluxes and viscous fluxes, respectively,

$$w = \begin{bmatrix} \bar{p} \\ \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \end{bmatrix}, \quad f_i = \begin{bmatrix} \bar{u}_i \\ \bar{u}_1 \bar{u}_i + \bar{p} \delta_{1i} \\ \bar{u}_2 \bar{u}_i + \bar{p} \delta_{2i} \\ \bar{u}_3 \bar{u}_i + \bar{p} \delta_{3i} \end{bmatrix}, \quad g_i = \frac{1}{Re} (1 + \nu_t / \nu) \begin{bmatrix} 0 \\ 2\bar{S}_{i1} \\ 2\bar{S}_{i2} \\ 2\bar{S}_{i3} \end{bmatrix}, \quad (2.2)$$

and $\mathbf{I}^m = diag(0,1,1,1)$ is the modified identity matrix annihilating the temporal derivative of pressure from the continuity equation. In Eq. (2.2), \bar{p} and \bar{u}_i are filtered pressure and velocity, δ_{ij} the Kronecker's delta, ν the molecular viscosity, ν_t the eddy viscosity, \bar{S}_{ij} the strain-rate tensor,

$$\bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right).$$
(2.3)

The eddy viscosity v_t in Eq. (2.2) is modelled to be

$$\nu_t = C_s \bar{\Delta}^2 \left| \bar{S} \right|, \tag{2.4}$$

where $|S| = \sqrt{2\bar{S}_{ij}\bar{S}_{ij}}$ is the magnitude of the strain-rate tensor, C_s the coefficient and $\bar{\Delta}$ the filter width. C_s is determined dynamically by the Lagrangian averaging procedure proposed by Meveneau et al. [25] and $\bar{\Delta}$ is related to the grid spacing by the cube root of the cell volume for the tetrahedral mesh used in this work.

Applying the Galerkin finite-element approach proposed by Mavriplis and Jameson [26] and the concept of a lumped mass matrix, the semi-discrete form of Eq. (2.1) at a node *P* can be obtained,

$$\mathbf{I}^{m} \cdot \Omega_{P} \left(\frac{\partial \mathbf{w}}{\partial t} \right)_{P} = -\sum_{e=1}^{n_{e}} \frac{\mathbf{F}^{A} + \mathbf{F}^{B} + \mathbf{F}^{C}}{3} \cdot \Delta \mathbf{S}_{ABC} + \sum_{e=1}^{n_{e}} \frac{4}{3} \mathbf{G}^{e} \cdot \Delta \mathbf{S}_{ABC}, \qquad (2.5)$$

where **F** and **G** represent the inviscid and viscous flux tensor respectively, n_e the number of the tetrahedrons sharing the vertex P, and Ω_P the volume sum of these tetrahedrons. As shown in Fig. 1, ΔS_{ABC} in Eq. (2.5) is the directed (outward normal) area of the triangular face opposite to P, \mathbf{F}^A , \mathbf{F}^B and \mathbf{F}^C are the inviscid flux tensors at vertex A, B and Crespectively, and \mathbf{G}^e is the constant viscous flux tensor over tetrahedron *PABC*.



Figure 1: Directed area of influence-domain-boundary face of a tetrahedron.

For temporal discretization, a dual-time-stepping scheme [27] is employed. The method of artificial compressibility [27] is employed to reduce the disparity in speeds of sound wave and convective wave. The Galerkin finite-element approximation is equivalent to the central difference. To prevent odd-even decoupling, an artificial dissipation operator [27] is adopted.

3 IB treatment by local DFD method and wall-modeling in DFD-LES

An immersed boundary method, named the local domain-free discretization (DFD) method [9,11], is employed to deal with the moving solid boundaries in LES of VIV problems. In the local DFD method, a partial differential equation is discretized at all mesh nodes inside the solution domain, but the discrete form at an interior node in the immediate vicinity of the immersed boundary involves some exterior nodes. This method has been extended to LES by Pu et al. [11], in which the wall modeling technique is adopted to alleviate the requirement of mesh resolution for the turbulent boundary layer. Here, with reference to Fig. 2, the evaluation of flow variables at an exterior dependent node is described briefly.

First, a reference point inside the solution domain is defined to calculate the flow variables at a given exterior dependent node. As depicted in Fig. 2, the point *F* on the wall-normal line with a distance δ from the wall-normal intersection *W* is defined as the reference point for the exterior dependent node *D*. The constant distance is calculated as

$$\delta = \max_{i=1,N_D} \delta_i,$$

where N_D is the total number of the exterior dependent nodes and δ_i is the distance between *W* and *E*. Point *E* is the interception between the normal line and the nearestto-wall triangular face whose vertices are all interior nodes. The flow variables at *F* can be obtained via linear interpolation over its host tetrahedron, i.e., *ABCH* in Fig. 2.



Figure 2: Immersed boundary model. D: exterior dependent node for A, B, C; F: reference point; W: wall-normal intersection.

Solving a simplified momentum equation in the wall-normal direction and using the non-penetration condition at point *W*, the pressure at *D* can be obtained,

$$p_D = p_F + |FD| \left(\frac{d\tilde{V}}{dt}\right)_W,\tag{3.1}$$

where \tilde{V} is the normal velocity of the body motion and p_F the pressure at F. According to the non-penetration condition at W, the linear extrapolation gives the normal velocity at D,

$$\tilde{v}_D = \frac{|FD|\tilde{V}_W - |WD|\tilde{v}_F}{|FW|},\tag{3.2}$$

with \tilde{V}_W the normal velocity of the body motion at W and \tilde{v}_F the normal velocity at the reference point F. To calculate the tangential velocities at the exterior dependent node D, the wall shear stresses yielded by the wall modeling technique are enforced at the immersed boundary. A modified local coordinate system [28,29] is employed to simplify the solution of the wall model. This local coordinate is composed of the streamwise direction η , the wall-normal direction n, and the binormal direction ζ which is perpendicular to the $\eta - n$ plane. The streamwise direction η is approximated to be the projection of the velocity direction at the reference point onto the surface. With this local system, the wall model only needs to be solved in the η - direction.

The wall model adopted in this work is based on the simplified TBLEs [30], which can be written as

$$\frac{\partial}{\partial n} \left[(v + v_t) \frac{\partial \tilde{u}}{\partial n} \right] = 0, \tag{3.3}$$

where \tilde{u} is the velocity in tangential direction η . Imposing the no-slip condition at W and

integrating Eq. (3.3) from F to W, the wall shear stress is obtained,

$$\tau_w = \mu \frac{\partial \tilde{u}}{\partial n} \Big|_{y=0} = \frac{\tilde{u}_F - \tilde{U}_W}{\int_0^{\delta} \frac{dy}{v + v_t}},$$
(3.4)

where \tilde{U}_W is the tangential velocity of body motion at W and \tilde{u}_F the tangential velocity of fluid at the reference point F. With the approximation of τ_w ,

$$\tau_w = \mu \frac{\partial \tilde{u}}{\partial n} \bigg|_{y=0} \approx (v_F + v_{t,F}) \frac{\tilde{u}_F - \tilde{u}_D}{|FD|}, \tag{3.5}$$

the tangential velocity at the exterior dependent node D can be obtained,

$$\tilde{u}_D = \tilde{u}_F - \frac{|FD|}{v_F + v_{t,F}} \tau_w, \tag{3.6}$$

where v_F and v(t,F) are the molecular and eddy viscosities at *F*, respectively.

For more details, one can refer to the work of Pu et al. [11].

4 Algorithms for coupling of FSI

The local DFD method coupling with the wall modeling technique for LES of turbulent flows has been validated via numerical experiments for various turbulent flows involving stationary or moving boundaries [11]. To validate the performance of present LES-DFD method for simulating FSI problems, the one-DOF motion of an elastically mounted cylinder freely oscillating in the cross-flow direction is considered in this work. The motion of a rigid circular cylinder can be represented by the following non-dimensional equation,

$$\frac{d^2Y}{dt^2} + \frac{4\pi\zeta}{U_r}\frac{dY}{dt} + \frac{4\pi^2}{U_r^2}Y = \frac{2C_y}{\pi m^*}.$$
(4.1)

In this equation, Y = y/D denotes non-dimensional displacement of the cylinder, $\zeta = c/(2\sqrt{Km})$ the damping ratio, m^* the reduced mass (mass ratio), $C_y = 2F_y/(\rho D U_{\infty}^2)$ the lift coefficient and $U_r = U_{\infty}/(f_n D)$ the reduced velocity, with f_n the damped natural frequency of the structure, U_{∞} the free-stream velocity, D the diameter of cylinder, ρ the fluid density, m, c, and K the mass, damping coefficient and the stiffness coefficient of the spring oscillator, respectively.

The fluid and structure dynamics are coupled together by the following no-slip conditions:

$$\bar{u}_2 = \frac{dY}{dt}$$
 at Γ , (4.2)

where Γ represents the fluid/structure interface. Eq. (4.1) is a second-order ordinary differential equation. Numerically, this equation can be solved by transforming it into

two first-order ordinary differential equations as follows

$$\frac{dY}{dt} = \phi, \tag{4.3a}$$

$$\frac{d\phi}{dt} + \frac{4\pi\zeta}{U_r}\phi + \frac{4\pi^2}{U_r^2}Y = \frac{2C_y}{\pi m^*}.$$
(4.3b)

The algorithm for loose and strong coupling of fluid-structure interaction (LC-FSI and SC-FSI) presented by Borazjani et al. [31] is adopted. The LC-FSI is very attractive from the viewpoint of computational cost since it requires the solution of equations of fluid motion only once at each time step. However, LC-FSI is known to experience numerical instability because of its explicit nature [32]. In the proceeding VIV simulations, LC-FSI algorithm is not always stable. When the LC-FSI algorithm is unstable, it is replaced by the SC-FSI one. The SC-FSI implementation is substantially more expensive than LC-FSI since SC-FSI requires an iterative solution of the structure equations and the fluid equations need to be fully converged within each iteration.

5 Numerical experiments

In this work, all the numerical tests are performed at $Re = (U_{\infty}D)/v = 10^4$, $m^* = 11$ and $\zeta = 0.001$. These parameters are chosen to match those in the experimental study of Hover et al. [33]. The tetrahedral meshes used in our simulation are obtained by dividing the hexahedral cells of Cartesian meshes. To alleviate the mesh resolution in the near-wall region, the wall model based on the simplified TBLEs [30] is adopted. The application of wall modeling technique dictates that the near-wall mesh reslotion must be smaller than the boundary thickness. Successive refinement of the three dimensional LES mesh will lead to a dramatic increase of node number. So, in this work, the test of grid independent or grid convergence with a global mesh refinement is very difficult to be conducted due to the extremely large computational cost. In addition, the solution of implicitly filtered LES is sensitive to the numerical grid used [34]. For some LES solution, grid refinement causes the agreement with direct numerical simulation or experimental data to deteriorate [34].

5.1 VIV of an elastically-mounted circular cylinder in unbounded flows

The purpose of this section is to verify the accuracy and efficiency of the present LES-DFD method for VIV simulation. Numerical simulation of the unbounded flows around an elastically-mounted circular cylinder is conducted. Cylinder oscillation is constrained in the *y*-direction by a spring-damper system with the stiffness coefficient *K* and damping coefficient *c*, as shown in Fig. 3. The reduced velocity U_r ranges from 3 to 9. The variation of U_r is achieved by altering the frequency ratio ω_t (ratio of damped natural frequency of structure to fixed-cylinder vortex-shedding frequency) while maintaining the velocity of the freestream. Hover et al. [33] indicated that cylinder vibration is characterized by



Figure 3: Flow around a circular cylinder with one degree of freedom.



Figure 4: Sketch map of the computational domain.

the frequency ratio ω_t . The algorithm for LC-FSI is always applicable for high m^* [35,36]. So, the algorithm for LC-FSI is adopted, except for the case $U_r = 5.84$. Our numerical experiment shows that when LC-FSI is adopted for $U_r = 5.84$, the response amplitude is much under-predicted and this may be due to the complexity of the fluid dynamics at this reduced velocity. Therefore, the SC-FSI algorithm is adopted for $U_r = 5.84$.

The sizes of the computational domain are 30D and 20D in the streamwise and vertical directions, respectively. The two-dimensional sketch map of the computational domain is illustrated in Fig. 4. The background mesh is stretched in the *x* and *y* directions to make the mesh fine enough within the region where the cylinder will pass. In the wall-resolved LES/body-fitted computation for flows over a stationary cylinder at Re = 3900 [37, 38], the height of the first cell at the cylinder surface is about $3 \times 10^{(-3)}D$. In this work, wall-modeling technique is adopted at the immersed solid surface, so the mesh spacings in the near-wall region are set to be $\Delta x = \Delta y = 0.01D$, which is much larger than those in the wall-resolved LES computation. The maximum distance to the wall for all the reference points, $y^+|_{max}$ is first calculated at each time step and then y^+ is defined



Figure 5: Comparison of the response amplitude (a) and phase difference (b) with referenced experimental and numerical results.

to be

$$y^+ = \frac{1}{N_t} \sum_{i=1,N_t} y^+ |_{\max}$$

with N_t the total number of the averaging times. For $U_r = 3.78, 4.4, 5.26, 5.84, 6.56$ and 8.77, y^+ is 16.7, 17.9, 20.3, 20.0, 17.3 and 16.8, respectively. Due to the coarse mesh resolution, the size of the mesh used by wall-modeled LES can be reduced greatly. The total number of the computational nodes of the numerical simulations in this section ranges from 1.6×10^6 to 3.4×10^6 for different U_r . Meyer et al. [39] conducted a LES of the turbulent flow (Re = 3900) over a stationary cylinder with an immersed boundary method and the total number of computational nodes was about 7 million, even if a local mesh refinement was employed. The spanwise size of the domain is set to be 4D and the constant mesh spacing in spanwise direction is $\Delta z = 0.1D$. Periodicity is imposed in the spanwise direction. Based on the linearized characteristics approach [27], approximate non-reflecting far-field boundary conditions are constructed to improve the accuracy and rate of convergence. The size of non-dimensional time step is taken to be 0.003.

As shown in Fig. 5, the non-dimensional amplitudes A^* and the phase differences between lift and displacement for various reduced velocities are compared with those of the two-dimensional RANS simulation [19], three-dimensional DES (detached eddy simulation) [40] and experimental investigations [33]. The maximum response amplitude of $A^*=0.99$ is presently predicted at $U_r=5.84$ in the upper branch. As shown in Fig. 5(a), compared to the results of the referenced two-dimensional RANS or three-dimensional DES, the amplitudes in the lock-in region predicted by the present LES-DFD method agree much better with the experimental data. As shown in Fig. 5(b), compared to the referenced DES results, the phase differences predicted by the present LES-DFD method



Figure 6: Isosurface of Q = 0.3 and the contours of the spanwise vorticity on the isosurface.

agree better with the experimental data. The variation of phase differences has a distinct shift from 0 to 180°. Fig. 6 shows the instantaneous three-dimensional vortex shedding pattern in the wake of the cylinder for $U_r = 5.84$ represented by the isosurface of Q = 0.3, and the contours of the spanwise vorticity defined as

$$\omega_z = \left(\frac{\partial u_3}{\partial x_2} - \frac{\partial u_2}{\partial x_3}\right) \left(\frac{U}{D}\right)^{-1}$$

are plotted on the isosurface.

The corresponding power spectra density (PSD) for *Y* and *C_y* are shown in Fig. 7. As the natural frequency of the structure f_n approaches the vortex-shedding frequency *f*, the lock-in phenomenon or synchronization occurs. This behavior can be observed in Fig. 7, when the frequency ratios are at $\omega_t = 1.0$ and 0.8. It can also be seen from Fig. 7 that there is only a single frequency component in the displacement response while there are multiple harmonics in the lift force. When displacement frequencies of the cylinder are close to natural frequency of the structure, the PSD of lift coefficient shows multiple frequencies, which has also been referred in the DES investigation of Nguyen et al. [40].

5.2 VIV of a circular cylinder in channel flows: effects of the channel height

In this section, the VIV of a circular cylinder in channel flows is considered and the configuration is illustrated in Fig. 8. We aim to investigate the effects of the non-dimensional channel height $d^* = d/D$ on the cylinder response at a selected reduced velocity $U_r = 6.56$, which is in the upper branch for unbounded flow. The Reynolds number, reduced mass and damping ratio are the same as those in Section 5.1 and d^* are set to be 1.6, 2.6, 3.4, 4 and 6. Without considering the case of cylinder hitting the walls and being bounced



(c) PSD of displacement (left) and lift coefficient (right), $\omega_t = 0.8 (U_r = 6.56)$



(d) PSD of displacement (left) and lift coefficient (right), $\omega_t = 0.6 (U_r = 8.77)$

Figure 7: Power spectral density of non-dimensional displacement and lift coefficient at various frequency ratio $(m^* = 11, \zeta = 0.001, Re = 10000)$.



Figure 8: Side view of a cylinder in a straight channel, which is elastically mounted and subject to VIV in the transverse direction.

back, the minimum value of d^* is chosen to be 1.6. The streamwise size of the computational domain and the size of time step are also the same as those in Section 3. The spanwise size of the computational domain is 4D, which is consistent with the numerical simulations of cylinder VIV in unbounded or semi-infinite flows at similar Reynolds numbers [23, 24, 41]. The wall modeling technique is applied at the cylinder surface as well as the upper and lower walls of the channel. The mesh is stretched in the vertical direction and the mesh resolution in this direction is $\Delta y = 0.01D$ near all the solid walls. The piston condition is applied at the inlet and the convective condition [42] is applied at the exit. The total number of computational nodes ranges from 2.4×10^6 to 4.5×10^6 for different channel heights. The fluid dynamics are much more complex due to the existence of the channel walls, so the SC-FSI algorithm is adopted in all the computations in this subsection.

Fig. 9 displays the time histories of the non-dimensional displacement of the cylinder





Figure 9: Time histories of non-dimensional displacement (Y) at different d^* for the near-wall cylinder undergoing VIV at $U_r = 6.56$.



Figure 10: Variation of response amplitude of the cylinder with d^* .

for different channel heights. The variation of the response amplitude with the height d^* is presented in Fig. 10. It can be seen obviously that the cylinder vibration depends strongly on d^* . The amplitude of vibration increases monotonically with d^* until $d^* \approx 4$, and then the variation of displacement becomes nearly sinusoidal with a constant frequency. In the work of Wang et al. [43], the circular cylinder is held at various heights above a single plane wall and is also subjected to vibrate in the transverse direction. Also for the reduced velocity that is in the upper branch for unbounded flow, Wang et al. reported a variation of the response amplitude similar to that in Fig. 10.

For different values of d^* , the motion of the cylinder exhibits different features. When d^* is small, as shown in Fig. 9 for $d^* = 1.6$, the displacement pattern generally is very complex due to the presence of the two parallel walls. When d^* becomes larger, the displacement patterns become nearly sinusoidal, as shown in Fig. 9 for $d^* = 4$ and 6. The



Figure 11: Power spectra density of displacement (left) and lift coefficient (right) at different d^* .

PSD for displacement and lift coefficient at $d^* = 1.6$ and 4 are shown in Fig. 11. In the case of $d^* = 1.6$, the response of the cylinder exhibits some type of non-linearities. The peaks in the displacement spectrum (the left in Fig. 11) clearly indicate that the response displacement has multiple frequencies. In the case of $d^* = 4$, the displacement spectrum has only one frequency. It can be seen from the spectrum of lift coefficient (the right in Fig. 11) that the vortices are shedding in multiple frequencies, which is different from the VIV of a cylinder in unbounded flow (see Fig. 7(c)).

Fig. 12 shows the patterns of vortex shedding in the instantaneous span-averaged flow field when the cylinder is near or at the equilibrium position. In the twodimensional RANS results of [18, 44, 45], the vortex street in 2S or 2P mode is regular. In the present results shown in Fig. 12, the near-cylinder wake flow has broken into a



Figure 12: Vortex contour and streamlines in the instantaneous span-averaged flow field of the vibrating cylinder with various d^* at Re=10000, $U_r=6.56$, and $\zeta=0.001$.

chaotic state. It can be clearly seen from the vortex patterns that when $d^* = 1.6$, the formation of vortex shedding from the cylinder surface is suppressed by the wall proximity and the separation zone in the boundary layers on the channel walls is not large. The vor-



Figure 13: Partial view of the streamlines at $d^* = 4$.

tex shedding from the cylinder surface when $d^* = 2.6$ and 3.4 are obviously in 2S mode. Even though the predicted response amplitudes at $d^* = 4$ and 6 are close to each other, the vortex shedding patterns are different due to the effect of the channel walls. Moreover, at $d^* = 4$, the channel walls are induced to separate by the vortices shedding from the cylinder surface, while at $d^* = 6$, the effect of the vortices on the plane boundary layers is weak and no separation occurs in the wall boundary layers. However, the boundary layer over the flat walls downstream the cylinder becomes a bit thicker. Fig. 13 is the partial view of the separation zone on the top wall of the channel for $d^* = 4$. For $d^* = 2.6$, 3.4 and 4, the length of the separation zone in the boundary layers on the channel walls decreases as d^* gets larger.

6 Conclusions and summary

In this study, the DFD method is extended to LES of FSI and the VIV of an elastically mounted rigid circular cylinder held in the middle of a straight channel, is numerically investigated. To alleviate the requirement of the near-wall mesh resolution, a wall modeling technique based on the simplified TBLEs is employed.

The accuracy of the present LES-DFD method for FSI is verified by the simulation of flows around an elastically-mounted cylinder in an unbounded flow. It is shown that the present LES-DFD method is more accurate and reliable than the referenced RANS and DES methods. With the wall modeling technique, the LES-DFD method can be used to simulate VIV problems on a relatively coarse mesh.

The effects of the plane channel walls on the VIV of the circular cylinder are examined under a selected reduced velocity $U_r = 6.56$, which is in the upper branch for the unbounded flow. The amplitude of the cylinder vibration increases monotonically with the non-dimensional height of the channel d^* until $d^* \approx 4$. It is also shown that the proximity of the wall leads to some type of non-linearities in the cylinder response as evidenced in the power spectra density of the lift coefficient and displacement. The vortex shedding is suppressed by the channel walls. In addition, the vortices shedding from the cylinder surface force the wall boundary layers to separate. For $d^* = 2.6$, 3.4 and 4, the length of the induced separation zone in the boundary layers over the channel walls decreases as d^* gets larger.

Acknowledgements

This work has been supported by National Natural Science Foundation of China under Grant No. 11972191.

References

- [1] C. S. PESKIN, Flow patterns around heart valves: a numerical method, J. Comput. Phys., 10 (1972), pp. 220–252.
- [2] G. IACCARINO AND R. VERZICCO, *Immersed boundary technique for turbulent flow simulations*, Appl. Mech. Rev., 56 (2003), pp. 331–347.
- [3] R. MITTAL AND G. IACCARINO, *Immersed boundary methods*, Ann. Rev. Fluid Mech., 37 (2005), pp. 239–261.
- [4] F. SOTIROPOULOS AND X. L. YANG, *Immersed boundary methods for simulating fluid-structure interaction*, Prog. Aerospace Sci., 65 (2014), pp. 1–21.
- [5] W. X. HUANG AND F. B. TIAN, Recent trends and progress in the immersed boundary method, J. Mech. Eng. Sci., 233 (2019), pp. 7617–7636.
- [6] C. SHU AND L. F. FAN, A new discretization method and its application to solve incompressible Navier-Stokes equations, Comput. Mech., 2001; 27 (2001), pp. 292–301.
- [7] C. SHU AND Y. L. WU, Domain-free discretization method for doubly connected domain and its application to simulate natural convection in eccentric annuli, Comput. Methods Appl. Mech. Eng., 191 (2002), pp. 1827–1841.
- [8] C. H. ZHOU, C. SHU AND Y. Z. WU, Extension of domain-free discretization method to simulate compressible inviscid flows, Int. J. Numer. Methods Fluids, 53 (2007), pp. 176–199.
- [9] C. H. ZHOU AND C. SHU, A local domain-free discretization method for simulation of incompressible flows over moving bodies, Int. J. Numer. Methods Fluids, 66 (2011), pp. 162–182.
- [10] C. H. ZHOU, RANS simulation of high-Re turbulent flows using an immersed boundary method in conjunction with wall modeling, Comput. Fluids, 143 (2017), pp. 73–89.
- [11] T. M. PU, C. H. ZHOU AND J. Q. AI, Extension of the local domain-free discretization method to large eddy simulation of turbulent flows, Int. J. Numer. Methods Fluids, 90 (2019), pp. 456–478.
- [12] C. H. K. WILLIAMSON AND R. GOVARDHAN, Vortex-induced vibrations, Ann. Rev. Fluid Mech., 36 (2004), pp. 413–455.
- [13] A. KHALAK AND C. H. K. WILLIAMSON, Investigation of relative effects of mass and damping in vortex-induced vibration of a circular cylinder, J. Wind Eng. Industrial Aerodyn., 69-71 (1997), pp. 341–350.
- [14] M. H. CHUNG, Transverse vortex-induced vibration of spring-supported circular cylinder translating near a plane wall, Euro. J. Mech. B-Fluids, 55 (2016), pp. 88–103.
- [15] S. DANESHVAR AND C. MORTON, Vortex-induced vibration of a circular cylinder with low mass ration near a plane wall, ASME 2017 Fluids Engineering Division Summer Meeting, 2017; Volume 1C. Waikoloa, Hawaii, USA.
- [16] C. N. JI, Z. XIAO, Y. Z. WANG AND H. K. WANG, Numerical investigation on vortex-induced vibration of an elastically mounted circular cylinder at low Reynolds number using the fictitious domain method, Int. J. Comput. Fluid Dyn., 25 (2011), pp. 207–221.
- [17] M. ZHAO, L. CHENG, H. AN AND L. LU, Three-dimensional numerical simulation of vortexinduced vibration of an elastically mounted rigid circular cylinder in steady current, J. Fluids Structures, 50 (2014), pp. 292–311.

- [18] Z. Y. PAN, W. C. CUI AND Q. M. MIAO, Numerical simulation of vortex-induced vibration of a circular cylinder at low mass-damping using RANS code, J. Fluids Structures, 23 (2007), pp. 23–37.
- [19] N. B. KHAN, Z. IBRAHIM, L. T. T. NGUYEN, M. F. J. JAVED AND M. JAMEEL, Numerical investigation of the vortex-induced vibration of an elastically mounted circular cylinder at high Reynolds number ($Re = 10^4$) and low mass ratio using the RANS code, PLoS ONE, 12(10) (2017), e0185832.
- [20] Z. KANG, W. NI AND L. SUN, A numerical investigation on capturing the maximum transverse amplitude in vortex induced vibration for low mass ratio, Marine Structures, 52 (2017), pp. 94– 107.
- [21] G. F. ROSETTI AND G. VAZ, On the numerical simulations of captive, driven and freely moving cylinder, J. Fluids Structures, 74 (2017), pp. 492–519.
- [22] G. F. ROSETTI, G. VAZ AND A. L. C. FUJARRA, URANS calculations for smooth circular cylinder flow in a range of Reynolds numbers: solution verification and validation, J. Fluids Eng., 134 (2012), 121103.1-18.
- [23] Y. JUS, E. LONGATTE, J. C. CHASSAING AND P. SAGAUT, Low mass-damping vortex-induced vibrations of a single cylinder at moderate Reynolds number, J. Pressure Vessel Tech., 136 (2014), 051305-1.
- [24] D. PASTRANA, J. C. CAJAS, O. LEHMKUHL, I. RODRIGUEZ AND G. HOUZEAUX, *Large-eddy simulations of the vortex-induced vibration of a low mass ratio two-degree-of-freedom circular cylinder at subcritical Reynolds numbers*, Comput. Fluids, 173 (2018), pp. 118–132.
- [25] C. MENEVEAU, T. S. LUND AND W. H. CABOT, A Lagrangian dynamic subgrid-scale model of turbulence, J. Fluid Mech., 319 (1996), pp. 353–385.
- [26] D. J. MAVRIPLIS AND A. JAMESON, Multigrid solution of the Navier-Stokes equations on triangular meshes, AIAA J., 28 (1990), pp. 1415–1425.
- [27] A. BELOV, L. MARTINELLI AND A. JAMESON, *Three-dimensional unsteady incompressible flow calculations using multigrid*, AIAA-1997-0443, (1997).
- [28] C. H. ZHOU AND C. SHU, A local domain-free discretization method to simulate three-dimensional compressible inviscid flows, Int. J. Numer. Methods Fluids, 61 (2009), pp. 970–986.
- [29] A. DADONE AND B. GROSSMAN, *Ghost-cell method for analysis of inviscid three-dimensional flows on Cartesian-grids*, Comput. Fluids, 36 (2007), pp. 1513–1528.
- [30] M. WANG AND P. MOIN, Dynamic wall modeling for large-eddy simulation of complex turbulent flows, Phys. Fluids, 14 (2002), pp. 2043–2051.
- [31] I. BORAZJANI, L. GE AND F. SOTIROPOULOS, Curvilinear immersed boundary method for simulating fluid structure interaction with complex 3D rigid bodies, J. Comput. Phys., 227 (2008), pp. 7587–7620.
- [32] P. CAUSIN, F. GERBEAU AND F. NOBILE, *Added-mass effect in the design of partitioned algorithm for fluid-structure problems*, Compute. Methods Appl. Mech. Eng., 194 (2005), pp. 4506–4527.
- [33] F. HOVER, S. MILLER AND M. TRIANTAFYLLOU, Vortex-induced vibration of marine cables: experiments using force feedback, J. Fluids Structures, 11 (1997), pp. 307–326.
- [34] T. BOSE, P. MOIN AND D. YOU, *Grid-independent large-eddy simulation using explicit filtering*, Phys. Fluids, 22 (2010), 105103.
- [35] K. SCHULZ, AND Y. KALLINDERIS, Unsteady flow structure interation for incompressible flows using deformable hybrid grids, J. Comput. Phys., 143 (1998), pp. 569–597.
- [36] C. FARHAT, ZEE VAN DER, AND P. GEUZAINE, Provably second-order time-accurate looselycoupled solution algorithm for transient nonlinear computational aeroelasticity, Comput. Methods Appl. Mech. Eng., 195 (2006), pp. 1973–2001.

- [37] J. FRÖHLICH, W. RODI AND PH. KESSLER ET AL., *Large eddy simulation of flow around circular cylinder on structured and unstructured grids*, CNRS DFG Collaborative Research Programe, vol. 66, Vieweg, Braunschweig, (1998), pp. 31–338.
- [38] J. FRANKE AND W. FRANK, Large eddy simulation of the flow past a circular cylinder at $Re_D = 3900$, J. Wind Eng. Industrial Aerodyn., 90 (2002), pp. 1191–1206.
- [39] M. MEYER, A. DEVESA AND S. HICKEL ET AL., A conservative immersed interface method for large-eddy simulation of incompressible flows, J. Comput. Phys., 229 (2010), pp. 6300–6317.
- [40] V. T. NGUYEN AND H. H. NGUYEN, Detached eddy simulations of flow induced vibrations of circular cylinders at high Reynolds numbers, J. Fluids Structures, 63 (2016), pp. 103–119.
- [41] Z. LI, Vortex-induced vibrations with wall proximity, Ph. D. Thesis, B. Eng. (Hons), NUS, 2017.
- [42] P. S. LOWERY AND W. C. REYNOLDS, Numerical simulation of a spatially-developing, forced, plane mixing layer, 1986. Rep. TF-26. Thermosciences Division, Dept. of Mech. Engng, Stanford University.
- [43] X. K. WANG, Z. HAO AND S. K. TAN, Vortex-induced vibrations of a neutrally buoyant circular cylinder near a plane wall, J. Fluids Structures, 39 (2013), pp. 188–204.
- [44] E. GUILMINEAU AND P. QUEUTEY P, Numerical simulation of vortex-induced vibration of a circular cylinder with low mass-damping in a turbulent flow, J. Fluids Structures, 19 (2004), pp. 449–466.
- [45] M. ZHAO AND L. CHENG, Numerical simulation of two-degree-of-freedom vortex-induced vibration of a circular cylinder close to a plane boundary, J. Fluids Structures, 27 (2011), pp. 1097–1110.