ABSTRACT

Characteristics of the uniform flow around an oscillating circular disk have been investigated. The circular disk is forced to oscillate sinusoidally along its axis, and a uniform flow is introduced in the direction within the plane of the disk. The incompressible Navier-Stokes equations are solved with direct numerical simulations based on the finite volume method (FVM) using the open source CFD (Computational Fluid Dynamics) code OpenFOAM. Deforming mesh technique is adopted to simulate the oscillation motion of the disk. The thickness ratio (thickness/diameter) of the disk is 0.1. The hydrodynamic force component in the axis direction of the disk is written in a Morison’s equation-like form, and then the coefficients of added mass and damping are calculated using the Fourier analysis. The added mass of the disk decreases as the velocity of the in-plane flow increases. However, the existence of the in-plane flow with a relatively large velocity significantly increases the damping of the disk.

KEY WORDS: DNS; oscillatory flow; heave plate; vortex shedding

INTRODUCTION

Heave plates installed in Spar platforms have been demonstrated effect components to minimize the heave motion response of the platform due to incident wave forces. In the past two decades, investigations of the hydrodynamic performance of heave plates have been conducted both experimentally and numerically, see, e.g., Prislin et al. (1998), He (2003), Tao & Thiagarajan (2003a,b). These studies were mainly focused on the geometric dependence of the hydrodynamic characteristics, e.g., thickness ratio (Thiagarajan et al., 2002), porosity (Tao & Dray, 2007), edge taper angles of the heave plate (Shen et al., 2012) and the distances between multiple heave plates (Magee et al., 2000). However, the effect of the current which is widely existing in oceans on the hydrodynamic performance of heave plates has not been well addressed yet.

In this study, numerical simulations are carried out for the steady currents around a heave plate with the technique of CFD (computational fluid dynamics). We consider a circular disk with the thickness of 0.1D, where D is the diameter of the disk. Keulegan-Carpenter number (KC) and frequency number (β) are used to characterize the oscillation motion of the disk, and they are defined as

\[ KC = \frac{U_{max} T}{D} \]

\[ \beta = \frac{D^2}{\nu T} \]

where \( U_{max} \) is the maximum speed of the oscillation, \( T \) is the period of oscillation, \( a \) is the amplitude of oscillation, \( \nu \) is the kinematic viscosity of the fluid.

The dimensionless velocity for the current, i.e., the reduced velocity, is given as

\[ V_r = \frac{U_c}{\omega D} \]

where \( U_c \) is the magnitude of current velocity and \( \omega \) is the angular frequency of oscillation, taken as \( \omega = 2\pi/T \).

The Reynolds numbers for the oscillation motion (\( Re_o \)) and current (\( Re_c \)) are defined respectively as

\[ Re_o = \frac{U_{max} D}{\nu} = (KC)\beta \]

\[ Re_c = \frac{U_c D}{\nu} \]

In this study, we mainly focus on the effects of current on the hydrodynamic characteristics of heave plate, therefore, the cases with same \( KC \) and \( \beta \) numbers but different \( V_r \) numbers are...
considered, i.e., \( KC = 1 \), \( \beta = 200 \) and \( 0 \leq V < 0.25 \). Hence, \( Re_o = 200 \) and \( Re_c < 315 \).

**MATHEMATICAL FORMULATIONS AND NUMERICAL METHODS**

The Navier-Stokes (N-S) equations for an incompressible viscous fluid are written as

\[
\frac{\partial u_i}{\partial x_i} = 0 \quad (6)
\]

\[
\frac{\partial u_i}{\partial t} + \frac{\partial u_i}{\partial x_j} u_j = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} \quad (7)
\]

where \( x_i \) denotes the coordinate in \( i^{th} \) direction, \( i = 1–3 \); \( u_i \) denotes the corresponding velocity component; \( p \) is the pressure and \( \rho \) is the density of the fluid. The coordinates \( x_1, x_2 \) and \( x_3 \) are also referred as \( x, y \) and \( z \), respectively.

The N-S equations are discretized using the finite volume method (FVM) based on the open source CFD code OpenFOAM. OpenFOAM is mainly applied for solving problems in continuum mechanics. It is built based on the tensorial approach and object oriented techniques (Weller et al., 1998). The PIMPLE algorithm which merges PISO (Pressure Implicit with Splitting of Operators) and SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithms is used in the present study. The spatial schemes for interpolation, gradient, Laplacian and divergence are linear, Gauss linear, Gauss linear corrected and Gauss linear schemes, respectively. All these schemes are in second order. The second order Crank-Nicolson scheme is used for the time integration. Further details of these schemes are given in OpenFOAM (2009).

The axis of the disk coincides with the \( z \)-direction and the current is in \( x \)-direction. Therefore, the drag force coefficients in \( x \)- and \( z \)-directions are calculated, respectively, as

\[
C_x = \frac{F_x}{\frac{1}{2} \rho U_c^2 D t_d} \quad (8)
\]

\[
C_z = \frac{F_z}{\frac{1}{8} \rho U_{ref}^2 \pi D^2} \quad (9)
\]

where \( F_x \) and \( F_z \) are the force components in \( x \)- and \( z \)-directions acting on the disk, respectively, and they are directly calculated by integrating the pressure and viscous shear stress over the disk surfaces; \( t_d \) is the thickness of the disk; in the case of steady flow normal to a stationary disk carried out for code validation, the reference speed \( (U_{ref}) \) is taken as the free stream speed \( (U_s) \); and in the oscillating disk case, \( U_{ref} \) is taken as the maximum speed of the disk \( (U_{max}) \). The time-averaged value of \( C_x \) is denoted as \( \overline{C_x} \). The force component acting on the disk in \( y \)-direction is zero due to the symmetry flow configuration.

The axis viscous drag force acting on the disk could be written in a Morison’s equation-like form (Tao & Dray, 2008), as follows

\[
F_{zd}(t) = C_d \frac{1}{2} \rho D^2 \frac{\pi}{4} U(t) |u_z(t)| \quad (10)
\]

where \( C_d \) is the damping coefficient. By using the Fourier analysis, the damping and added mass coefficients could be calculated, respectively, as (Sarpkaya et al., 1981)

\[
C_d = \frac{3\omega}{\rho \pi D^2 U_{max}^2} \int_0^T F(t) \cos(\omega t) dt \quad (11)
\]

\[
C_a = \frac{1}{\pi \rho \forall U_{max}} \int_0^T F(t) \sin(\omega t) dt \quad (12)
\]

where \( \forall \) is the immersed volume of the disk, taken as

\[
\forall = (1/4) \pi D^2 t_d \quad (13)
\]

**COMPUTATIONAL DOMAIN AND BOUNDARY CONDITIONS**

Fig. 1. Cut-away view of the computational domain and the grids.

As shown in Fig. 1, a spherical computational domain is used in this study, and the disk locates at the core of the domain. On the disk surface, no slip boundary condition is prescribed and the outer boundary is set as zero normal gradient velocity boundary.
condition. The pressure at the out boundary is fixed at zero. The computational domain is discretized with hexahedral elements, and the grids near the surface of the disk are refined to resolve the steeper gradient there, see Fig. 1. The oscillation motion of the disk is realized using the deforming mesh technique, in which the grid points between the disk and outer boundary are relocated at each time step in a spring-like way.

**CODE VALIDATION**

Benchmark calculations for the flow normal to a circular disk with thickness $t_d = 0.1D$ at low Reynolds numbers ($Re \leq 100$) are carried out to validate the present numerical approach. The computational domain size and boundary conditions are following the configurations in the DNS (direct numerical simulation) study by Shenoy & Kleinstreuer (2008). However, we use a denser mesh with 599040 elements than that with 242066 elements used by Shenoy & Kleinstreuer (2008). As shown in Fig. 2., the drag force coefficient $C_d$ calculated from the present study agree well with the DNS results by Shenoy & Kleinstreuer (2008) and the experimental results by Roos & Willmarth (1971). Therefore, the present numerical approach is able to give reliable results for the flow around a circular disk.

**CONVERGENCE STUDIES**

Table 1. Results of the grid and time step convergence studies for the case at $KC=1$ and $V_r=0.15$

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Elements</th>
<th>$n_1/D$</th>
<th>Time step</th>
<th>$C_x$</th>
<th>$C_a$</th>
<th>$C_d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>117760</td>
<td>0.0055</td>
<td>T/2500</td>
<td>687</td>
<td>5.50</td>
<td>2.84</td>
</tr>
<tr>
<td>B</td>
<td>397440</td>
<td>0.0036</td>
<td>T/5000</td>
<td>691</td>
<td>5.50</td>
<td>2.87</td>
</tr>
<tr>
<td>C</td>
<td>942080</td>
<td>0.0027</td>
<td>T/7500</td>
<td>694</td>
<td>5.50</td>
<td>2.89</td>
</tr>
</tbody>
</table>

Convergence studies are carried out to assert the effects of spatial and temporal resolutions on the calculated results. The parameters selected for the convergence are $KC=1$ and $V_r=0.15$. The details of the three typical cases with different grid elements and time steps, as well as the hydrodynamic characteristics are shown in Table 1. $n_1$ is the normal wall distance of the first layer grid away from the disk surface. As shown, as the number of elements increases from 117760 to 942080 and the time step decrease from T/2500 to T/7500, the values of $C_x$, $C_a$ and $C_d$ do not change significantly, indicating a good convergence for the grid resolution and time step. The settings with medium mesh size and time step, i.e., Mesh B and time step of T/5000, are selected to simulate the cases with other current velocities.

**RESULTS AND DISCUSSIONS**

**Flow Observations**

Fig. 3. shows the instantaneous flow structures around the oscillating disk under various in-plane currents, i.e. $V_r=0-0.25$, when the disk has maximum axis velocity. The flow structures are identified by the so-called Q-criterion proposed by Hunt et al. (1988). As shown, the flow is axisymmetric about the axis of the disk in the case in the absence of current, i.e. $V_r=0$, is. And the axisymmetric flow is disturbed by introducing the in-plane current, even it is still symmetric about the center plane ($y=0$). The flow pattern appears to be significantly influenced by the magnitude of the current. The diffusion of vortices generated by the oscillating disk is enhanced as $V_r$ increases. Furthermore, the streamwise distance of the neighboring vortices increases as $V_r$ increases.

**Hydrodynamic Characteristics**

Fig. 4 shows the variations of the time-averaged current drag force coefficient ($\bar{C}_x$) versus $V_r$ at $KC=1$. It appears that $\bar{C}_x$ decreases nonlinearly as $V_r$ increases for the cases at $KC=1$, this is similar to the results for the low Reynolds number flow around other structures, see e.g. Roos and Willmarth (1971) for a circular disk or a sphere, and Sumer and Fredsoe (1997) for a circular cylinder.

Fig. 5 shows the variations of the added mass coefficient ($C_a$) versus $V_r$ for the oscillation at $KC=1$. As shown, as $V_r$ increases, $C_a$ decreases significantly. For a Spar platform, the added mass of heave plates plays an important role in determining the natural heave frequency of the platform, and this frequency increases as added mass decreases. We expect a large added mass and a low natural heave frequency for a Spar to avoid resonance with incident waves. Therefore, the decrease in added mass due to the existence of current is harmful to the hydrodynamic performance of the platform.
Fig. 3. Flow visualizations of the oscillating disk under various current conditions at KC=1.

Fig. 4. Variations of the time-averaged current drag force coefficient ($\overline{C_x}$) versus $V_r$ at KC=1.

Fig. 5. Variations of the added mass coefficient ($C_a$) versus $V_r$ for the oscillation at KC=1.

Fig. 6. Variations of the damping coefficient ($C_d$) for the oscillation versus $V_r$ at KC=1.
Fig. 6 shows the variations of the damping coefficient ($C_d$) versus $V_r$ for the oscillation at $KC=1$. It is evident that, at $KC=1$, the influence of $V_r$ on $C_d$ is relatively weak at low $V_r$ numbers, and $C_d$ in small current condition (e.g., $V_r=0.1$) is slightly lower than that for the case in the absence of current ($V_r=0$). For the cases at $V_r>0.1$, $C_d$ increases significantly as $V_r$ increases.

Overall, the existence of in-plane current appears to influence the hydrodynamic characteristics of an oscillating circular disk. The added mass coefficient ($C_a$) of the disk at larger current velocities is smaller than that for the cases at lower current velocities, however, the damping coefficient ($C_d$) increases as $V_r$ for the cases at $V_r>0.1$.

CONCLUSIONS

Direct numerical simulations have been conducted for the steady in-plane current around an oscillating circular disk at low Reynolds numbers. The effects of the velocity of the current on the flow structures around the disk, as well as on the hydrodynamic characteristics, i.e. current drag force coefficient ($C_x$), added mass coefficient ($C_a$) and damping coefficient ($C_d$), are evaluated. The magnitude of the current significantly influences the flow patterns, and the diffusion of vortices generated by the oscillating disk is enhanced as $V_r$ increases.

The added mass coefficient decreases as $V_r$ increases and the damping coefficient increases as $V_r$ increases.

It should be noted that the present study was initially motivated by the hydrodynamic characteristics of heave plates in Spar platform under current conditions. However, there is still a large difference between the present flow configuration and the real heave plate at sea. The hydrodynamic characteristics of heave plates may also be influenced by the geometric configurations (e.g. plate shape, plate porosity, stiffener and truss structures) and some other flow parameters (e.g. Reynolds number, turbulence intensity and current profiles). Nevertheless, this preliminary study provides some fundamental understandings on the flow features and the hydrodynamic characteristics of heave plate under current conditions. The results presented in this article are of significance for engineering applications and future investigations on similar issues.

ACKNOWLEDGMENT

This work has been supported by National Natural Science Foundation of China (Grant No. 50979057). This support is gratefully acknowledged.

REFERENCES