Numerical investigation of hydrodynamic influence on long oscillating plates in a viscous fluid

A N Nuriev¹, A M Kamalutdinov², A G Egorov² and O N Zaitseva³

¹Lobachevsky State University of Nizhni Novgorod 18, 23 Gagarina pr., Nizhnii Novgorod, 603950, Russian Federation;
²Kazan Federal University, 18, Kremlyovskaya St., Kazan, Tatarstan 420008, Russian Federation;
³Kazan National Research Technological University, 68 Karl Marx st., Kazan 420015, Russian Federation

E-mail: nuriev_an@mail.ru

Abstract. In the paper an investigation of the hydrodynamic influence on oscillating thin plates in a viscous fluid is presented. The evolution of drag and inertial forces acting on plates in different flow regimes for different vibration amplitudes is considered. It is shown that in regimes with intense vortex formation the shape of the edges of the plates has a strong influence on the resistance. The drag force acting on plates with different shapes of the edges may differ by more than 14% at the same oscillation parameter values.

1. Introduction

The first studies of the interaction of oscillating plates with a quiescent viscous fluid were carried in the 60s-70s of the last century in connection with the study of the wave loads on offshore structures. The hydrodynamic experiments [1, 2] which were conducted at this period of time and the models of vortex interaction [3] which were based on the results of these experiments allowed to determine the basic parameters of the problem, to estimate the hydrodynamic forces and to describe the structure of the flows around oscillating plates for the region of large oscillation amplitudes. At the same period the linearized Stokes theory for thin plates was formulated [4] for the case of small oscillation amplitudes. For a long time these results formed the basis for estimation of the influence of the fluid on oscillation process. However, the formation of new applied fields, such as atomic microscopy and robotics, development of nano-electronic devices and piezoelectric micro-fans, the creation of new methods of measurement the damping properties of the materials [5-7], stimulated the further progress in the studies of this problem.

Nowadays the problem of interaction of oscillating plates with a viscous fluid is often discussed in the context of the problem of determination of the aerodynamic damping of free or forced oscillations of elongated beams. In the assumption that the length of the beams substantially greater than their width and thickness, the interaction of the beams with surrounded air can be considered in the framework of quasi-two-dimensional flow hypothesis, according to which the aerodynamic forces acting on each cross-section of the beam can be considered as a result of the planar flow past a section. This approach formed the basis of numerous experimental and numerical studies [5-11] conducted in the last decade.
In spite of a large number of investigations of the problem of the flow past an oscillating plate, a comparison of their results (including experiments and numerical investigations) reveals a wide range of variations in the estimates of the hydrodynamic influence in the region of moderate and large amplitudes of the vibration. One hypothesis explaining the reasons for such differences [12] is the incomplete correspondence of the geometric parameters of the samples used in various experiments and numerical studies, in particular, different processing of the edges of the plates. To test this hypothesis in the paper a numerical investigation of flow past oscillating plates with different shape of edges is carried out.

2. Problem statement
Let us consider plane flows that arise near long thin plates, when they oscillate in a viscous incompressible fluid according to the harmonic law

\[ s = A \sin \omega t, \]

where \( s \) is the displacement, \( A, \omega \) are the amplitude and the frequency of oscillation respectively. The geometrical characteristics of the cross-section of the investigated plates are shown in Fig. 1, aspect ratio \( h/b \) in the study was taken equal to 1/10.

![Figure 1. Geometrical characteristics of the plates: rectangular plate (I), plate with chamfered edges (II).](image)

After normalization of the spatial coordinates, time and velocity by \( b, b/U_0, -U_0 \) respectively (where \( U_0 \) is the amplitude of the oscillation velocity), the system of the governing equations of fluid motion in Cartesian coordinates are written as

\[
\begin{align*}
\frac{\partial U}{\partial t} + U \cdot \nabla U &= -\nabla p + \left(\beta KC\right)^{-1} \Delta U, \\
\nabla \cdot U &= 0,
\end{align*}
\]

(1)

where \( U = U(u, v) \) is the non-dimensional velocity, \( p \) is the non-dimensional pressure, \( \beta = b^2 \omega / (2\pi\nu) \) is the non-dimensional frequency, \( KC = 2U_0 \pi / b\omega = 2\pi A / b \) is the non-dimensional amplitude, \( \nu \) is the kinematic viscosity of fluid.

For the numerical solution of the problem it is convenient to rewrite governing equations in a moving coordinate system associated with the plate. In this case, to retain the governing system in the form (1) in the new non-inertial coordinate system, a new pressure is determined:

\[ p = \bar{p} + 2\pi KC^{-1} \sin \left(2\pi t / KC\right) x. \]

Here the first term \( \bar{p} \) is the pressure in the fixed coordinate system and the second is the contribution of inertial components.
On the surface of the plate the no-slip conditions are fulfilled:

\[ u = v = 0. \]

At the infinity the changes in velocity are determined by the following law:

\[ u = \cos(2\pi / KC), \quad v = 0. \]

The calculation of hydrodynamic forces acting on the plate from the fluid in the presented dimensionless formulation is carried out according to the formula:

\[ F = \int_S pnds - \int_S \vec{\sigma} \cdot nds, \]

where \( \vec{\sigma} \) is a viscous stress tensor, \( S \) is the surface of the plate, \( n \) is the surface normal vector.

The resulting force vector \( F \) can be decomposed into a vertical component \( F_y \) (lift force) and the horizontal \( F_x \), which consists of drag and inertia forces. Inertia component arises due to the acceleration of the fluid and consists of two parts: the inertia force of the added mass arising due to the local acceleration near the plate, and Froude-Krylov force which is related to the pressure gradient created in the fluid for the simulation of the oscillatory flow.

To approximate the influence of inertial and viscous components of the horizontal force Morison's formula [13] is used:

\[ F_x = \pi C_M \frac{du_x}{dt} + C_D |u_x| u_x, \]

where \( u_x \) is the fluid velocity at infinity, \( C_M \) is the coefficient of inertia forces and \( C_D \) is the drag coefficient.

3. Numerical scheme

The numerical solution is carried out using the OpenFOAM software package [14] based on the numerical scheme [15]. The flow is modelled in a rectangular domain with a plate in the center of it. The size of domain is \( 60 \times 40 \). The sides of the computational domain are parallel to the axes \( (Ox, Oy) \) of the selected Cartesian coordinate system. Oscillations of the flow occur along the \( Ox \) axis.

Two types of block meshes are used to discretize the computational domain. The first type is structured meshes. The improvement of the resolution near the plate on these meshes is achieved by linear grading of the cell size in the normal direction to the sides of the plate. This provides a smooth change of cell size in the computational domain. On grids of the second type, in addition to the mesh grading, the local mesh refinement by cutting cells near the plate in 4 is used. That leads to a violation of the regularity condition and to sharp change of the size of the cells in the joining zone of the refined and basic meshes. However, this allows to significantly increase the resolution of mesh without multiple increasing of the total number of cells. The maximum number of cells used in calculations is \( 3 \times 10^5 \).

The discretization of the governing equations is carried out using the finite volume method (FVM) in the Cartesian coordinate system. The discrete values of the velocity components and pressure are localized in the centroids of the cells. To calculate volume integrals over the control volume the general Gaussian procedure is used. The pressure gradient approximates by the linear interpolation. In the diffusion terms, when the discretization of the Laplace operator is carried out, the normal velocity gradients on the surface of the cells are approximated by a symmetric second-order scheme with non-orthogonality correction (in the case of plates with truncated edges).

For the interpolation of variables in convective terms the hybrid Spalding scheme [16] is used (an analogue of the "Streamline upwind" scheme that is widely used in finite element method). It represents the combination of linear and upwind interpolation. Linear interpolation is used in an area where the cell Reynolds number (or Peclet number) is lower than 2 (\( \text{Re}_h < 2 \)). The results of [17,18] show, that the hybrid scheme provides good matching of numerical results with experimental data for a wide range of Reynolds numbers for the considerate class of problems.
The implicit Euler scheme is used for the time discretization. The time step in all calculations is chosen from the condition that the maximum Courant number does not exceed 0.1.

The solution of discretized problem is carried out using the PISO method. To solve the system of equations for the pressure, the preconditioned conjugate gradient (PCG) method with the geometric-algebraic multigrid (GAMG) preconditioner was applied. In the GAMG realization we used the Gauss-Seidel method with one and two pre- and postrelaxations for smoothing, respectively, and the faceAreaPair algorithm for agglomeration of the mesh cells. The systems of equations for the velocity components were solved by the bi-conjugate gradients (PBiCG) method with the preconditioner based on an incomplete LU factorization. All calculations were distributed according to the MPI technology and the method of decomposition of the calculation domain.

4. Results
Numerical simulation of flows around the plates of type I and II (see Fig 1) were carried out at $\beta = 50, 0.1 < KC < 10$. The results of the measurement of hydrodynamic coefficients ($C_D, C_M$) are shown in Fig. 2. As can be seen in the range of moderate and large amplitudes, the coefficients $C_D$ and $C_M$ for different plates have (as was assumed in the [12]) visible differences.

![Figure 2. $C_D$ and $C_M$ via $KC$ for $\beta = 50$ for plates of types I and II.](image)

The average relative variation of the $C_M$ coefficient depending on the shape of the edges is 10%. These variations are associated mainly with the the Froude-Krylov force which is different for plates of I and II type. For the analysis of drag coefficient $C_D$ let us consider its evolution in different flow regimes.

In the range $5 \leq KC \leq 10$ the values of drag coefficient for rectangular plates are on average 14% higher than $C_D$ for plates with chamfered edges. The behavior of the $C_D$ in both cases is well described by the asymptotic proposed in the work [12]:

$$C_D = c \left(\frac{KC}{2\pi}\right)^{-0.58} \quad (2)$$

For a rectangular plate, the constant in (2) is $c = 5.1$. The obtained dependence for this plate is close to the data [KC]. For the plate of II type, the value of the constant is $c = 4.412$. It coincides with the value which defined in the experimental study [12] for plates with chamfered edges. To analyze
the causes of the marked differences in drag force measurements we will compare the flow structure around the plates and measure the pressure distribution over the surface of the plates.

In the range $5 \leq KC \leq 10$, the flow around the plates has a periodic vortex-shedding pattern (see Fig. 3). On each half-cycle, a pair of vortices is detached from the plate. Detachments occur alternately from opposite angles of the plate. At the initial moment (after detachment), the pairs of vortices move at an angle of $\sim 45^\circ$ relative to the axis of oscillation. The structure of the observed flow and the region of its localization well coincides with the experimental data presented in [1].

Comparison of the flow structure around the plates of different cross-section, as well as the pressure distribution on the plates are shown in Fig. 3. Flow visualization is obtained by coloring the fluid near the plate.

As can be seen in Fig. 3 the structure of the flows around the plates of different type is in general very similar. In particular the structure, size and position of the vortex pairs which are generated near plates are identical. However, in the vicinity of the edges of the plates there are still visible differences in flow associated with changes in the flow separation points. For the plate with chamfered edges the separation occurs at the vertex of an angle, for plates with a rectangular cross section, the separation occurs at the apex of the right angle from the side of the incoming stream. This factor effects on the pressure distribution on the plate surface (see Fig. 3). For the plate with chamfered edges, the difference between the pressure distribution on the right and left sides of the plate in the vicinity of the edges is less than for the rectangular plate. This leads to a decrease of the aerodynamic drag for the plate of II type.

![Flow structure and pressure distribution (c) in the vicinity of the plates](image)

**Figure 3.** Flow structure and pressure distribution (c) in the vicinity of the plates for $\beta = 55, KC = 7$: rectangular plate (a), plate with chamfered edges (b). Visualization performed by ink.

In the zone $1 < KC \leq 4$ the maximum differences between $C_d$ calculated for different plates exceed 15%. In this zone the regime with an asymmetric flow structure is established. The visualization of the flow for the case of the plate with chamfered edges is shown in Fig. 4. The distribution of the colored fluid in the figure shows that the vortex-shedding occurs mainly from one end of the plate. The reasons for the differences of the drag of plates of I and II type in this regime are the same as in the previous regime with a diagonal separation of vortices.
Figure 4. Flow structure in the vicinity of the plate for $\beta = 55, KC = 2.5$. Visualization performed by ink.

At the smallest values of the dimensionless amplitude ($KC \leq 1$) the values of drag coefficient for the two plates (of the I and II type) are almost completely consistent. In this zone stable symmetric flow regimes without intensive vortex formation (see Fig. 5) are localized. The practically complete absence of vortex mechanisms of influence on the plates is the main reason for the complete agreement of the results for different plates in symmetric regimes.

Figure 5. Flow structure in the vicinity of the plate for $\beta = 55, KC = 0.5$. Visualization performed by ink.

5. Conclusion
The comparative analysis of the results of the numerical simulation shows that the shape of the edges of the plate has a significant influence on the aerodynamic drag in flow regimes with intensive vortex formation. For the plate with chamfered edges the difference between the pressure distribution on the right and left sides of the plate in the vicinity of the edges is less than for the rectangular plates. This leads to a decrease the aerodynamic drag of the plate with chamfered edges. In the range of moderate
and large amplitudes of oscillation the values of the drag coefficient for a rectangular plate lie (on the average) 14% higher.

Acknowledgments
The research was supported by RSF (research project No. 15-19-10039).

References
[1] Singh S 1979 Forces on Bodies in Oscillatory Flow (Ph.D. thesis, University of London)
[2] Keulegan G H and Carpenter L H 1958 J Res. Natl. Bur. Stand. 60 (5) 423–40
[3] Graham J M R 1980 J. Fluid Mech. 97 (02) 331–46
[4] Tuck E O 1969 J. Eng. Math. 3 (1) 29–44
[5] Egorov A G, Kamalutdinov A M, Nuriev A N and Paimushin V N 2014 Mech. Compos. Mater. 50 (3) 267–78
[6] Egorov A G, Kamalutdinov A M, Nuriev A N and Paimushin V N 2017 Dokl. Phys. 62 (5) 257–261
[7] Egorov A G, Kamalutdinov A M, Paimushin V N and Firsov V A 2016 J. Appl. Mech. Tech. Ph+. 57 (2) 275–82.
[8] Aureli M and Porri M 2010 Appl. Phys. Lett. 96 (16) 164102
[9] Aureli M, Porri M and Basaran M E 2012 J. Sound. Vib. 331 (7) 1624–54
[10] Tafuni A and Sahin I 2015 J. Fluids Struct. 52 101–17
[11] Jalalisendi M, Panciroli R, Cha Y and Porfiri M A 2014 J. Appl. Phys. 115 (5) 054901.
[12] Egorov A G, Kamalutdinov A M and Nuriev A N 2018 J. Sound. Vib. 421 334–47
[13] Morison J R, Johnson J W and Schaaf S A 1950 J. P. T. 2 (05) 149–54
[14] Greenshields C 2018 OpenFOAM User Guide: CFD Direct, Architects of OpenFOAM
[15] Nuriev A N, Egorov A G and Zaitseva O N 2018 J. Appl. Mech. Tech. Ph+. 59 (3) 451–59
[16] Spalding D B 1972 Int. J. Numer. Meth. Eng. 4 (4) 551–59
[17] Justesen P 1991 J. Fluid Mech. 222 157–96
[18] An H, Cheng L and Zhao M 2009 Ocean Eng. 36 (14) 1089–97