Journal of Fluids and Structures 68 (2017) 15–31

Contents lists available at ScienceDirect

Journal of Fluids and Structures

journal homepage: www.elsevier.com/locate/jfs

Numerical study of fluid flow past a rotating elliptic cylinder

Sandeep N. Naik, S. Vengadesan*, K. Arul Prakash

Indian Institute of Technology Madras, Chennai, Tamil Nadu, India, 600036

ARTICLE INFO

Keywords:
Elliptic cylinder
Rotating
IBM
Vortex shedding
Hovering vortex

ABSTRACT

Numerical solutions are presented for fluid flow past a rotating elliptic cylinder for different non-dimensional rotational rates (0.5, 1.0 and 1.5) and various axis ratios (0.1, 0.3, 0.5, 0.7 and 1.0). The two-dimensional, incompressible Navier–Stokes equations are solved using Immersed Boundary method on a non-body conforming Cartesian grid. Reynolds number based on the perimeter of the elliptic cylinder is 100. Periodic flow after an impulsive start of the elliptic cylinder is studied. Flow patterns and variation of aerodynamic force coefficients are reported for different set of parameters and it is observed to be very different from that of rotating circular cylinder. As the cylinder rotates, the trailing edge of the cylinder aids the flow while the leading edge deters the flow. This has significant effect on the wake region and vortex shedding pattern. Complex interactions between cylinder and shed vortices result in many interesting phenomenon, such as formation of hovering vortex. Parameters affecting the formation of hovering vortex and its underlying mechanism are analyzed. Aerodynamic force coefficients and vortex shedding frequencies are presented and it is observed that force coefficients have multiple amplitudes and frequencies.

1. Introduction

Flow past bluff bodies have been the interest of many fluid dynamicists over the years. In particular, studies on flow past circular cylinders (stationary, rotating and oscillating) and aerofoils at different angles of attack (AOA) have dominated the broad spheres of both experiments and computations. To some extent, this is because of the ease with which one can set up equipment in experiments or define the geometry and boundary conditions in numerical simulations. A comprehensive review on flow past cylinders and associated flow structures is given in Zdravkovich (1996) and Williamson (1996). These bluff body flows command the attention of engineers because of its applications in flow control and drag reduction. This seemingly simple problem has rich physics involved in it. Recent studies have focused on three-dimensional instabilities for flow past rotating cylinder (Pralits et al., 2010; Radi et al., 2013; Rao et al., 2013; Pralits et al., 2013).

There have been few studies that discuss the effects of axis ratio on flow past rotating cylinder. The problem of rotating elliptic cylinders can act as a base case to study the auto-rotation phenomenon and associated aerodynamics. Mccutchen (1977) and Alexander (2004) have reported that plants and trees take advantage of this auto-rotation and increase the dispersion area of their seeds. Smith (1971) experimentally investigated the autorotation phenomenon in wings of different shapes and axis ratios about the spanwise axis. Aerodynamic forces were calculated for a range of *Re* (25000–250000). Fluid flow patterns were very different from that of static wings and the stalling angle was very large. This was attributed to the delay in growth of the boundary layer and acceleration of the fluid on the upper (suction) surface of the wing when the angle of attack is rapidly increased. Freely falling wings were also studied and it was found that the forces acting on the wings were comparable to those observed in the fixed axis tests and it

* Corresponding author.
E-mail address: vengades@iitm.ac.in (S. Vengadesan).

http://dx.doi.org/10.1016/j.jfluidstructs.2016.09.011
Received 8 March 2016; Received in revised form 31 August 2016; Accepted 30 September 2016

appeared that the flow patterns were similar. Lugt and Ohring (1977) were one of the first to study the flow past rotating elliptic cylinder. They presented 2D numerical results for a rotating thin elliptic cylinder in a viscous fluid at rest and in a parallel stream. Simulations were carried out at Re = 400 and 2000 for rotation in stationary fluid and at Re = 200 for rotation in parallel stream. They found that a pair of vortex is shed in every half cycle of rotation. They reported that retreating edge vortex was stronger than the advancing edge vortex. They attributed this to the retreating edge vortex remaining much longer on the airfoil surface and the influence of the vortices shed in previous cycles. In their study, maximum drag occurred just before the plate is normal to the flow, and maximum lift just after the plate reaches the horizontal position. Iversen (1979) studied the variation of the autorotation characteristics with changes in the Reynolds number, aspect ratio, thickness ratio and moment of inertia of a flat plate based on the data gathered from previous investigations. He came up with correlations concerning forces on the elliptic cylinders. Lugt (1980) studied autorotation of an elliptic cylinder about an axis fixed perpendicular to the flow. He concluded that proper interplay between vortex shedding and vortex-edge interaction is necessary to sustain autorotation of the cylinder.

Nair and Sengupta (1997) studied two-dimensional flow past rotating elliptic cylinder numerically. They solved the N–S equation in Vorticity–Stream function formulation using a higher order finite difference method and presented early time solutions for Re = 200 and 1000 at two different rotation rates, α = 1 and 4. Axis ratio of the cylinder was 0.1. They concluded that, for higher Re, vorticity is confined to a small region surrounding the body and this leads to a strong interaction between the vortices and cylinder. They suggested that a long time study would be useful to understand the interaction of shed vortices in the wake. Hocking (1974) studied the development of Stewartson layers on cylinders having high curvature. He took a thin elliptic cylinder in a rotating frame as an example and found the modified form of such layers. Lua et al. (2010) reported Digital Particle Image Velocimetry (DPIV) measurements for a two-dimensional elliptic airfoil rotating about its own axis of symmetry in a fluid at rest and in a parallel free stream. Reynolds number based on the chord length of the cylinder was 200 and 1000. A range of $\alpha = 0.417, 0.834, 1.67$ and 2.5 were considered. For the case of stationary fluid, Re based on the tangential velocity of the cylinder ranged from 400 to 2000. For a rotating airfoil in stationary fluid, they observed two distinct types of vortex shedding depending on Re. Shed vortices are either diffused or rotated with the cylinder which leads to the vortex suction effect. For low Re, the shed vortices diffuse quickly and the cylinder rotates in its own wake of diffused vorticity. For high Re, the shed vortices are stronger and even after few rotations, the vortices remain attached to the tip of the cylinder and form tongue like structures. In a parallel stream they observed that the rotation rate plays a significant effect on the flow topology. Similar to Lugt and Ohring (1977) two pair of vortices were shed for each rotation of the cylinder. Based on the rotation rates, different extent of vortex-edge interaction was observed. The edge of the cylinder stretched the vortices in the upstream direction as it rotated. With the increase in rotation rate, this lead to the formation of what Lua et al. (2010) have called 'hovering vortex'. They reported that the initial starting configurations of the airfoil affects the flow field during initial phase of rotation only and have no effect on the overall flow topology at later times.

Recently, Ruifeng (2015) presented 3D numerical results for flow past rotating wing. He used an open source CFD solver Popinet (2003) for simulating the flow. The parameters used in the study were same as that of Lua et al. (2010) i.e. Re = 200 and $\alpha = 0.417, 0.834$ and 1.67. In addition to that, effects of changing the axis ratio, $AR = 0.5, 0.25$ and 0.167 on the spanwise vorticity were studied. He observed that for large axis ratio streamwise vortices are as strong as spanwise vortices; and for small axis ratio, the streamwise vortices are interconnected with the main vortex rings, resulting in a complicated wake structure. He observed that, as rotation speed increased, the high-pressure zone on the leeward side of the wing turns into a low-pressure region at small angle of attack. He attributed this to the change of vortex shedding pattern where at high rotation rate, vortices cluster near the wing creating a low-pressure zone. These clustering vortices eventually lead to a hovering vortex as previously observed by Lua et al. (2010). The aerodynamic moment and forces, all increased with increasing rotation rates.

Most of the studies on flow past a rotating elliptic cylinders are restricted to initial flow development after the cylinder is started from rest. The changes in aerodynamic forces due to the combination of varying AR and α are not studied in detail. The effect of hovering vortex on the flow features and aerodynamic forces remain unanswered. Our present study aims at addressing few of these questions by studying the vortex dynamics and the corresponding changes in force coefficients. In this work, we have performed 2D simulations of incompressible fluid flow past a rotating elliptic cylinder and studied the effects of axis ratio, defined as the ratio of minor axis to the major axis ($AR = b/a$), and rotational rates on the flow features. Since the force coefficients are calculated on the surface of the cylinder, the perimeter (l_p) of the elliptic cylinder is kept constant (= π) in all the cases and Re is defined based on l_p and free stream velocity. This ensures that all the cylinder have equal surface area and we can compare the aerodynamic forces across AR. Another important point to note is that the Re defined based on perimeter (as characteristic length) is π times more than the Re based on the largest chord of the cylinder (i.e. diameter for circular cylinder). Since all the simulations are performed at Re = 100, if we recast the Re based on the largest chord of the cylinder, then the Reynolds number is approximately 30. From literature (Pralits et al., 2010; Radi et al., 2013; Rao et al., 2013; Pralits et al., 2013), we note that this is less than the critical Reynolds number (based on diameter of cylinder) above which three-dimensionality in flow features is observed. Akoury et al. (2008) observed that for rotating circular cylinder, the critical Re above which 3D effects become dominant increases with rotation rate. Therefore, in the present study, three-dimensionality in flow features will be marginal.

2. Governing equations and boundary conditions

The unsteady incompressible viscous flow past a body whose boundary is given by Γ, immersed in a fluid domain Ω, is governed by the Navier–Stokes equations. The non-dimensional equations are given below: