Computers & Fluids 102 (2014) 170–181

Contents lists available at ScienceDirect

Computers & Fluids

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

The flow structures of a transversely rotating sphere at high rotation rates

J. Dobson [a,b], A. Ooi [a], E.K.W. Poon [a,*]

[a] The University of Melbourne, Victoria 3010, Australia

[b] Imperial College of Science, Technology and Medicine, London SW7 2A2, United Kingdom

ARTICLE INFO

Article history:
Received 12 April 2013
Received in revised form 27 March 2014
Accepted 2 July 2014
Available online 9 July 2014

Keywords:
Laminar flow
Transverse rotation
Sphere

ABSTRACT

This paper studies the flow structures surrounding a transversely rotating sphere for the previously unreported conditions of flow Reynolds numbers of $Re = 100$–300 and a rotation rate in the range $\Omega^* = 1.25$–3.00, where Reynolds number is based on sphere diameter and the free stream velocity and rotation rate Ω^* is defined as the maximum sphere surface velocity normalised by free stream velocity. The flow remains steady at $Re = 100$. At $Re = 250$ and 300, the flow can be classified into 2 flow regimes at these rotation rates. Flow at lower rotation rates is dominated by the well known Kelvin–Helmholtz type instability vortex formation mechanism which produces regular, coherent vortices. As rotation is increased above $\Omega^* = 1.50$, the frequency of vortex formation decreases by an approximate factor of two. Linear stability analysis shows that this change in frequency can be attributed to significant changes in the geometry of free shear layers near the surface of the sphere. As rotation rate is increased further ($\Omega^* > 2.00$), the flow enters the 'separatrix' regime. The separatrix divides the free stream flow and the surface-driven boundary layer that engulfs the sphere surface as a result of the rapid sphere rotation. The surface-driven boundary layer leads to better surface pressure recovery and lowers the time-averaged C_D. On the other hand, time-averaged C_L increases with Ω^* in the 'separatrix' regime.

1. Introduction

For many decades researchers have sought to understand and predict the forces which arise when a fluid passes over a solid rotating sphere. This information is of interest because it provides a basic model for a number of engineering systems, in particular systems involving particle laden flows. In a number of particle laden flows, for example in chemical processing pipelines, geophysical flows or combustion processes, the particle (modelled as a sphere) undergoes rotation due to interactions with the wall and the collisions between particles, as well as the presence of shear gradients in the flow field. This rotation has an impact on the forces the particle experiences as well as having the effect of enhancing turbulence in the surrounding fluid [1]. This in turn can have an important impact on the dispersion of particles within the fluid and mixing of the surrounding fluid, demonstrating the importance of considering rotation effects.

The influence of rotation on the motion of a sphere was first recorded by Sir Isaac Newton [2]. It is Benjamin Robins and Heinrich Magnus who are credited with the formal description of the effect and the transverse force arising from rotation is a such known as the Magnus–Robins effect. In order to make use of this effect in engineering applications, researchers then began exploring methods for predicting the forces the sphere would experience based on given flow conditions. One of the early contributors to such work were Rubinow and Keller [3]. Using the Stokes and Oseen expansion, they found that the lift force F_L could be approximated from $F_L = \pi r^3 \rho \Omega U_\infty$ in the limit of very low Reynolds numbers ($Re = \rho U_\infty D/\mu \ll 1$), where D stands for the sphere diameter, r is the radius, ρ represents the fluid density, Ω is the angular velocity of the sphere, U_∞ is the free stream velocity and μ is the dynamic viscosity of the fluid. Although useful in a small range of conditions, the Rubinow and Keller correlation was of no use to the majority of engineering applications and a set of correlations with wider applicability were sought. Researchers such as Tsuji et al. [4] found that analysis of experimental data could lead to correlations which were applicable over a much wider range of conditions. Tsuji et al. defined their lift force in terms of the non-dimensional lift coefficient C_L and found that $C_L = 0.4 \pm 0.1\Omega^*$, where the non-dimensional rotation rate is given by $\Omega^* = \Omega D/2U_\infty$. This expression was said to be applicable to non-

* Corresponding author.
E-mail address: epoon@unimelb.edu.au (E.K.W. Poon).

http://dx.doi.org/10.1016/j.compfluid.2014.07.001
0045-7930/

dimensional rotation rates $\Omega^* \leqslant 0.70$ and Reynolds numbers in the range $Re = 550$–1600 which made it more applicable to real engineering applications. These results were obtained through use of a trajectographic technique. The experimental correlations obtained in this study were then followed by a number of other experimental studies which examined flow behaviour under different sets of flow variables. Some studies [5] focused on high Reynolds number flows, while others tended to focus on flow conditions where the Reynolds number is low but $\Omega^* \rightarrow 6.00$ [6]. Loth [7] produced a correlation for lift coefficient which accounts for both the Reynolds number and rotation rate and is given by $C_L = \Omega^*(1-(0.675+0.15(1+\tanh[0.28(\Omega^*-2)]))\times\tanh(0.18Re^{0.5}))$.

Apart from a few isolated examples [1], experimental studies provide information on the force on the sphere but cannot elaborate on the structures of the flow. Numerical simulations offer a solution to this problem and by the early 1990s numerical solutions of the governing equations could produce revealing information on the flow field [8]. Subsequent studies solved for the full Navier–Stokes equations using a number of different numerical techniques such as the marker and cell method [9], immersed boundary methods [10] and spectral collocation methods [11–16]. The study by Giacobello et al. [11] found that rotation significantly altered the structure of the flow field compared with the stationary case, resulting in significant changes in the hydrodynamic forces. In particular, the study revealed the existence of three different flow regimes at $Re = 300$ and rotations in the range $\Omega^* = 0 - 1.00$. The latter of these regimes, a shear layer instability type vortex formation mechanism occurring for $\Omega^* \geqslant 0.80$ was previously unreported in the literature. Kim [10] has expanded the understanding of this topic to rotations of $\Omega^* = 1.20$ and has also found shear layer instability mechanisms at this rotation rate. The study by Poon et al. [17,18] revealed that the orientation of the rotation axis also has a significant effect on the flow structure and hydrodynamic forces. In particular, the shear layer instability mechanism reported by Giacobello et al. [11] was found at $Re = 300$ not only for transverse rotation but also for a limited number of oblique rotation axis angles.

Given these interesting flow developments at $Re = 300$ the aim of the present study is to expand the understanding of flow behaviour to conditions of higher rotation, in the range $\Omega^* = 1.50$–3.00. Such flows are still industrially relevant, with studies such as that by Bluemink et al. [19] showing particles in industrial flow conditions rotating at $\Omega^* = 2.00$. This study will also only examine the case of rotation about an axis transverse to the free stream flow, since this is found to be the most common rotation axis orientation [20]. Beginning with a discussion of the problem geometry and the numerical formulation in Section 2, the study then goes on to discuss and categorise in Section 3.1 the flow physics in different flow regimes. A subsequent discussion of force coefficients is presented in Section 3.2 and in the final analysis, Section 4, conclusions are drawn regarding behaviour.

2. Problem definition and solution methodology

2.1. Problem definition

The basic geometry of the problem as illustrated in Fig. 1, where a solid sphere is shown in the coordinate system used by the computer simulations. The purpose of the study is to solve the discrete form of the incompressible unsteady Navier–Stokes equations for a domain defined by a spherical polar coordinate system. This requires solution of Eqs. (1) and (2) in terms of the primitive variables:

$$\frac{d\boldsymbol{u}}{dt} + \boldsymbol{u}\cdot\nabla\boldsymbol{u} = -\nabla P + \frac{1}{Re}\nabla^2\boldsymbol{u} \tag{1}$$

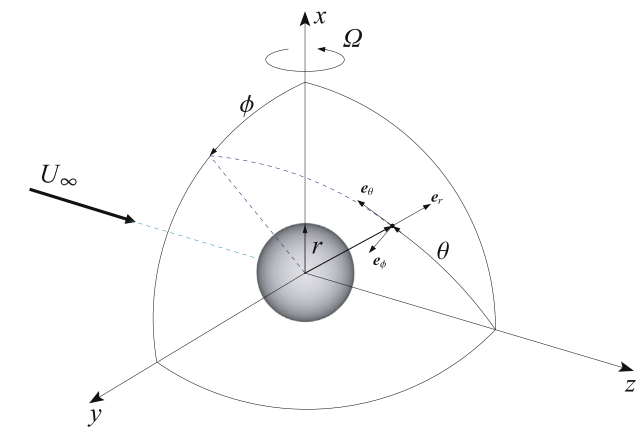

Fig. 1. The spherical and the Cartesian coordinate systems. The free stream flow is aligned with the z-axis and the sphere is rotating in the x-direction.

and

$$\nabla\cdot\boldsymbol{u} = 0. \tag{2}$$

The solution of the Navier–Stokes equations also depends on the boundary conditions imposed, both at the outer surface of the domain and at the inner surface (the surface of the sphere). Sphere rotation has an important influence at this inner surface and the rotation is described in this study in terms of Ω^* where the rotation axis is collinear with the x axis.

Solution of the Navier–Stokes equations yields velocity and pressure information at every computational cell within the domain. In order to make sense of this information the data is redefined in terms of particular variables which represent the collective behaviour of the flow. Three of the key 'collective variables' used in this study are the force coefficients in the three orthogonal directions, C_{Lx}, C_{Ly} and C_D. These three components are

$$C_{Lx} = \frac{2L_x}{\rho U_\infty^2 S}, \tag{3}$$

$$C_{Ly} = \frac{2L_y}{\rho U_\infty^2 S}, \tag{4}$$

$$C_D = \frac{2D_z}{\rho U_\infty^2 S}, \tag{5}$$

where L_x is the lift force in the x direction, L_y is the lift force in the y direction, D_z is the drag force in the z direction and $S = \pi D^2/4$. The other important parameter in this study is the Strouhal number, which is defined as

$$St = \frac{fD}{U_\infty}, \tag{6}$$

where f is the dominant frequency calculated from the energy spectrum of C_{Ly}.

2.2. Solution methodology

In this study the Fourier–Chebyshev spectral collocation method is used with a two-step fractional time step to solve the Navier–Stokes equations. This solver, based on the formulation of Mittal [21], is a popular choice for solving the Navier–Stokes equations as noted by Brown and Minion [22] and a variation of the solver used in this study has also been used in several other studies [11–16]. There are two main aspects to the numerical solution of the Navier–Stokes equations, namely the need to discretise the problem in the spatial domain and the need to discretise the problem in the temporal domain. Temporal discretisation of the