Vortex Flow Past a Sphere in a Constant-Diameter Pipe

L. Grinis, U. Tzadka

SCE, Beer-Sheva, Israel

ABSTRACT

This paper describes an experimental and numerical study of concentrated vortex flow past a sphere in a constant-diameter pipe. We study the response of a sphere elastically-mounted in a pipe to a fluid flow through the pipe. The fluid flows through the gap between the pipe wall and the sphere's surface, displacing the sphere. As a result the sphere is able to move with two degrees of freedom in the radial direction of the pipe. We observe the rotation of the sphere around the longitudinal axis of the pipe. This rotation is induced by a vortex shedding from the sphere. We will discuss computational simulation and experimental investigations of this strong sphere rotation and the interaction of forces between pipe and sphere. The angular frequency of the sphere rotation and force interaction with the pipe wall has been measured as a function of the wide range Reynolds Number (10⁴ - 10⁶). This study demonstrates the possibility of using this phenomenon for industrial applications.

KEY WORDS: Sphere; pipe; vortex; shedding; rotation, CFD.

INTRODUCTION

In many of the technological processes used in today's industry, aqueous saturated salt solutions have to be piped from one place to another. During this operation, sedimentation and rust on the inner surface of the pipes may occur. Methods used to remove these sediments are chemical, mechanical, aerodynamic or hydrodynamic. Moreover, the commonly used mechanical cleaning methods involve brushes or rubber balls, pushed inside the pipes by water or compressed air to scrape the sediment off the surface, or require hammering the pipes from the outside. These methods are not always adequate to remove all kinds of sediments. In addition, these methods often require an outside source of power to drive the scraping or hammering device. These methods are thus time consuming and not always effective. In this study we consider an application of vortex-induced vibration for various industrial applications.

Vortex-induced vibration of structures is of practical interest in many branches of engineering. It has also been noted (Martner, Joubert, and Chong, 2003; Cyril, 1995) in cases of elastically-mounted spheres vibrating in pipes with a fluid flow, where one of the problems is fluid-structure interactions. Jauvtis, Govardhan and Williamson (2001) proposed studying the behavior of the fluid flow interaction with the sphere for different conditions. sphere for different conditions. It is known (Govardhan and Williamson, 2000; Hover, Tvedt and Triantafyllou, 2001) that the process of fluid flow around a bluff body (sphere, cylinder) is accompanied by a periodic vortex trail (Karman vortex street), which induces vibrations. It is also known that the resulting forces act on the bluff body in a direction transversely to that of the flow. The growth and movement of these vortices create a fluctuating lift-and-drag force on the body. The problem of vortex-induced vibration of spheres, regarding fluid flow, has been thoroughly investigated, and several reviews (Jonson and Patel, 1999; Kim and Choly, 2002) discuss this problem. In this work, we observed the phenomenon of the vibration and rotation of the sphere in the flow around the X – longitudinal axis of the inner surface of the pipe. This phenomenon has been investigated by a simplified theoretical model, by computational simulation and tested experimentally. Applications of computational fluid dynamics (CFD) to the industry continue to grow as this advanced technology takes advantage of the increasing speed of computers. In the last two decades, different areas of flow modeling including grid generation techniques, solution algorithms, turbulence modeling, and computer hardware capabilities have witnessed tremendous development. In view of these developments, computational fluid dynamics can offer a cost-effective solution to many engineering problems. Various researchers used turbulence modeling to simulate flow around axisymmetric bodies. In this study we use Fluent (fluid dynamics computer simulation software) to model the flow around a sphere in the tube, when the flow is turbulent. Prediction of flows that exhibit massive separation remains one of the principal challenges of CFD. The main interest of the present study is to calculate the turbulent flow over a sphere at high Reynolds numbers.

The nature of the flow around a sphere in tube changes as the Reynolds number of the flow increases. In general, the higher Reynolds number, the more complex the flow. The aims of this study are: the experimental investigation of the stability and instability of the vibration of the sphere in fluid flow inside a pipe; derivation of a simplified analytical expression for the stability of the vibrational motion of the sphere rotating in a pipe under conditions of fluid flow; validation of the mathematical model and