

The Biharmonic Approach for Unsteady Flow Past an Impulsively Started Circular Cylinder

Jiten C Kalita^{1,*} and Shuvam Sen²

¹ *Department of Mathematics, Indian Institute of Technology Guwahati, PIN 781039, India.*

² *Department of Mathematical Sciences, Tezpur University, PIN 784028, India.*

Received 20 April 2011; Accepted (in revised version) 12 December 2011

Available online 17 April 2012

Abstract. In this paper, a newly developed second order temporally and spatially accurate finite difference scheme for biharmonic semi linear equations has been employed in simulating the time evolution of viscous flows past an impulsively started circular cylinder for Reynolds number (Re) up to 9,500. The robustness of the scheme and the effectiveness of the formulation can be gauged by the fact that it very accurately captures complex flow structures such as the von Kármán vortex street through streakline simulation and the α and β -phenomena in the range $3,000 \leq Re \leq 9,500$ among others. The main focus here is the application of the technique which enables the use of the discretized version of a single semi linear biharmonic equation in order to efficiently simulate different fluid structures associated with flows around a bluff body. We compare our results, both qualitatively and quantitatively, with established numerical and more so with experimental results. Excellent comparison is obtained in all the cases.

AMS subject classifications: 35Q30, 65M06, 76D05, 76D25

Key words: Biharmonic, non-rectangular, N-S equations, transient, circular cylinder, von Kármán vortex street.

1 Introduction

Flow over a bluff body is a common phenomenon which occurs when fluid flows over an obstacle or along with the movement of a natural or artificial body. Common examples are the flows past an airplane, a submarine, an automobile, or wind blowing past a high-rise building. Although bluff bodies exist in many different shapes, the circular cylinder is considered to be the representative of a two dimensional bluff body. As such, the flow

*Corresponding author. *Email addresses:* jiten@iitg.ernet.in (J. C. Kalita), shuvam@tezu.ernet.in (S. Sen)

around a circular cylinder has been the subject of intense research in the last century and numerous theoretical, numerical and experimental investigations have been reported in the literature [1–18]. The time development of an incompressible viscous flow induced by an impulsively started circular cylinder is now a classical problem in fluid mechanics. It displays almost all the fluid mechanical phenomena for incompressible viscous flows in the simplest of geometric settings.

In the context of numerical studies on this problem, with the advent of Computational Fluid Dynamics (CFD), more and more computational methods for simulating fluid flows are coming into the fore [10–16, 19–25] which has led to the better understanding of the characteristics of the flow. Numerical simulation could now invade areas hitherto unexplored by experimentalists where it is possible to analyze all aspects of the flow at each stage of its development. A quick look at these works reveals that there exists only a few studies where a single numerical scheme has been employed to tackle the flow throughout the whole range of $10^0 \leq Re \leq 10^4$. The ranges of Reynolds numbers which came under the purview of these works, varied from one study to the other. Although investigations on vortex shedding has been quite popular with the experimentalists ever since Roshko [6] first measured the period of von Kármán vortex shedding behind a bluff body, the simulation and characteristic study of streak lines seem to have failed to attract the attention of numerical analysts. One of the objectives of the present study is also to address these two issues apart from capturing other flow characteristics for this problem.

Fluid flow problems governed by Navier-Stokes (N-S) equations can be solved by using a variety of numerical methods. As is well established, these methods can broadly be classified as finite difference, finite volume or finite element approach. Amongst these, finite difference (FD) method is the most popular approach that has been used quite frequently in CFD because of its easiness in implementation. In the FD set up, approximation of a higher order derivative generally requires more points and as such is associated with non-compact stencils. Such schemes, which are used for higher order differential equations on non-compact stencils, require additional conditions in order to tackle the difficulty of flow computation at the boundary. Contrary to these, a compact finite difference scheme [26–28] which utilizes grid points located only directly adjacent to the node, computes the flow with information solely from the nearest neighbours and are gaining popularity *via-a-vis* wide-molecule schemes [29–32].

Over the years, the CFD community has seen the extensive use of both the primitive variable and stream function-vorticity (ψ - ω) formulation to compute incompressible viscous flows governed by the N-S equations. Both these formulations have their relative advantages and disadvantages over each other: while the primitive variable formulation has been traditionally difficult because of the presence of the pressure term in the governing equations, a typical difficulty with the ψ - ω formulation is that the vorticity ω is not prescribed on the boundaries. Due to these facts, the biharmonic pure stream function form of the N-S equations, which eliminates the need to compute both pressure and vorticity, is emerging as an attractive alternative [33–36]. Besides, this approach has the advantage of requiring to solve only a single fourth order PDE instead of a system of