CHAPTER 1
INTRODUCTION

1.1 Overview

Most of the real life fluid flow problems are dominated by viscous effects and are associated with flow separation. This phenomenon is one of the most complex aspects of fluid flow. Separated flow behind two and three dimensional geometries leads to the formation of the wake region which encompasses complex phenomena like recirculation, mass and momentum transfer across shear layers and vortex shedding. Beyond a certain Reynolds number, the wake becomes asymmetric giving rise to alternate shedding of vortices from top and bottom sides of the particular geometry forming the well known 'Von-Karman vortex street'. The vortex shedding process repeats itself with a certain frequency which depends on freestream Reynolds number and geometry of the bluff body. In case of three-dimensional geometries, horse shoe vortices may emanate from the tips of the body.

Vortex shedding process leads to increase in negative base pressure and hence increase in pressure drag. This feature is utilized for reduction of velocity in aerospace vehicles like earth to orbit transfer vehicle and in aircrafts by using spoilers. Vortex shedding process is associated with time variant pressure and shear stress distribution on body surface. This leads to time variant loading on the body with a certain frequency which can be expressed as 'Strouhal Number'. When this frequency matches with the natural frequency of a structure, it can lead to a catastrophe. This phenomenon led to incidents like collapse of Tacoma suspension bridge in the State of Washington. Hence, estimation of this dynamic loading is important in the design of high rise towers, submerged structures, suspension bridges etc. Excessive mass and momentum transfer in the wake of bluff bodies leads to augmented heat transfer when bluff body shaped protuberances are placed on or near the surface of heat exchangers. This phenomenon is exploited for effective cooling in electronic devices. The above studies are also useful in the design of automobile shapes producing minimum aerodynamic drag.

Flow separation may broadly be classified into two types - bubble type separation and free shear layer type separation. In bubble type separation, the surface of separation encloses a fluid that is not a part of the main stream but is carried along with the body. The main stream does not wet the separated portion of the body surface. This type of separation is seen in the two-dimensional flow on bluff bodies having rapid closure. For streamlined shape, this type of separation is observed near the trailing edges of airfoils and wings at moderate to high angles of incidence.

In free shear layer type of separation, the space on either side of the surface of separation is filled by mainstream fluid, i.e., the main stream generally wets the body surface. This type of separation is generally encountered in case of the flow around three-dimensional bodies at different incidences, such as slender wings having sharp leading edges or bodies of revolution. In slender wings, the upper and lower surface boundary layers separate from the sharp leading edges to form free shear layers which grow in space in the form of spiral vortices above the wing and the inboard of the leading edges.

Attempt has been made over the last few decades to predict the various characteristics of bubble type separated flows. Some of these methods as described by Jacob (1967), Bhatley and Mcwhirter (1972), Woodward et al. (1974) and others are based on potential flow modelling while other methods as proposed by Baliga and Patankar (1983), Schneider and Raw (1987), Rodi et al. (1989), Masson et al. (1994) and others are based on Navier-Stokes equations.

The difficulty in modelling the bubble-type separation with the help of potential flow models stems from the fact that the various features of separated flow that essentially characterize a viscous wake are relatively little understood and hence difficult to simulate in potential flow models. These include the criteria of the separation point, base pressure, shape and extension of the wake and the features of the viscous wake involving the dissipation and diffusion of vorticity as it is convected downstream and also the cancellation of vorticity of opposite shear layers as their inner edges converge towards each other. Other complications arise because such flows are frequently unsteady with periodicity associated with the asymmetric formation of wake. Despite these problems, the bubble type separation has been studied using potential flow models by several early researchers like Jacob (1967), Bhatley and Mcwhirter (1972), Woodward et al. (1974) and others. The modelling practices involve proper accounting of the viscous effects by considering the effect of flow separation from the body surface. If the location of the separation point is supplied either from experiment, flow visualization or viscous flow calculations, the various aerodynamic parameters and the wake geometry can be predicted to a certain degree of accuracy. In spite of all such limitations, considerable success has been achieved with such potential flow modelling. A very significant advantage of potential flow modelling is its relatively simple formulation and less computing time in comparison with the Navier-Stokes solvers.

With the advent of high speed digital computers, attempt has been made to develop Navier-Stokes solvers since Navier-Stokes equations provide a complete description of the flow. If the various initial and boundary conditions are properly described and if the equations are solved accurately, all possible flow phenomena and their effects will be automatically predicted; this includes both types of separated flow phenomena discussed earlier, vortex formation, heat transfer and so on.

A number of numerical methods have been developed over the years for solving the incompressible Navier-Stokes equations. These methods vary in grid-layout, solution algorithms, pressure calculation techniques and other parameters. One of the earliest and most widely used methods is the Marker and Cell (MAC) method due to Harlow and Welch (1965). In this method, pressure is obtained by solving the pressure Poisson equation. Later, several pressure correction schemes like Semi-Implicit Method for Pressure Linked Equations (SIMPLE) due to Patankar and Spalding (1972), and a solution algorithm (SOLA) method due to Hirt et al. (1975) emerged. These methods make use of staggered grid arrangements to avoid spurious pressure oscillations.

For the above methods, use of rectangular grids is well suited to simple body geometries like square or rectangular cylinders. However, when the curvature of the body surface is involved, the application of rectangular grids severely impairs the accuracy of the solution. To circumvent this problem, attempt is generally made to use curvilinear body-fitted grid and the solution is usually carried out in the transformed plane. However, the accuracy of such a practice is questionable because of the introduction of Christoffel symbols (because of the use of contravariant or covariant velocity components) in the transformed momentum equations (Patankar, 1980). These symbols are generally functions of geometrical parameters and their presence in the momentum equations make them non-conservative.

In recent years, the use of non-staggered (collocated) grid is becoming increasingly popular because of its easy applicability to body-fitted coordinate systems and the implementation of boundary conditions. This approach remedies the problem faced by the staggered grid practices (i.e. the non-conservativeness of the governing equations because of the introduction of Christoffel symbols due to the use of covariant or contravariant velocity components) by retaining the Cartesian form of the velocity components and solving the problems in the physical plane. However, with this type of grid arrangement, problem arises from the development of spurious pressure modes and odd-even decoupling when the flow variables at the cell faces are approximated by linear interpolation of the adjoining nodal values. Special emphasis has to be given to obtain the cell face variables to ensure proper pressure-velocity coupling for all the cells.

In this context, the Physical Interpolation Approach (PIA) of Rhie and Chow (1983) is worth mentioning. In this approach, the cell face velocities are obtained by solving the momentum equations at each of the cell faces. This ensures the pressure-velocity coupling between the adjoining cells and prevents the odd-even pressure oscillations. However, this way of calculating the cell face velocities needs matrix inversion because of the use of implicit scheme and hence requires long computing time. Later, Deng et al. (1994) developed an explicit formulae called Consistent Physical Interpolation (CPI) for the calculation of cell face velocities and used these values with an implicit scheme for the solution of momentum equations at the cell centres. This method does not require any matrix inversion for calculating the cell face velocities and is therefore relatively less expensive than Rhie and Chow (1983) method. This method also avoids the spurious pressure modes associated with the non-staggered grids.

In the present investigation, bubble type separated flow problems have been studied at low Reynolds number by developing a two-dimensional incompressible Navier-Stokes solver.

1.2 Motivation of the present work

Review of the existing numerical methods for solving Navier-Stokes equations suggests that there is still some scope for improving the existing schemes and to evolve better ones in future. In addition, it is to be mentioned that not all aspects of the complex flow field characteristics of unconfined flow past two- and three-dimensional arbitrary geometries have been studied by researchers to the best of the author’s knowledge. Flow past three cylinders in various configurations and flow past multiple cylinders in close proximity to a plane wall as well as flow past airfoils at low Reynolds numbers are considered as the source of motivation for the present research work due to the complex flow pattern and non-linear behaviour of flow parameters involved in them.

1.3 Objectives and scope of the present thesis

The objective of the present research work is to introduce and establish the ‘CFRUNS’ scheme for solution of two-dimensional incompressible Navier-Stokes equations. It is also aimed to achieve a better understanding of the complex flow characteristics associated with flow past multiple cylinders as well as flow past a few airfoil sections at a low Reynolds number of 1000.

To achieve this goal, the scope of the research work is outlined as follows:

1. Development of incompressible finite volume laminar Navier-Stokes solver based on two-dimensional (2-D) unstructured grid comprising of triangular cells (this solver is referred to as 'CFRUNS').
2. Application of the above solver for studying various two-dimensional flow problems like:
    - Flow past a single circular cylinder.
    - Flow past two and three circular cylinders in side-by-side or tandem arrangements.
    - Flow past two and three circular cylinders of equal and unequal diameters in staggered arrangements.
    - Flow past single- and multi-cylinders in the vicinity of a plane wall.
    - Flow past streamlined bodies like NACA 0002, NACA 0012 and Eppler 340 airfoils.

1.4 Organization of the thesis

The overview and motivation of the present work followed by the objectives and scope of the present thesis are discussed in this chapter. The remaining parts of the thesis are organized in the following fashion.

Chapter 2 gives a brief review of literature related to potential flow models as well as viscous flow models involving incompressible Navier-Stokes equations relevant to the study of separated flow problems. Some papers reporting experimental results on such problems are also reviewed.

In **Chapter 3**, a generalized incompressible Navier-Stokes solver using body-fitted curvilinear grid has been developed, based on finite volume approach, for solving unconfined flow past arbitrary two-dimensional body geometries at zero and non-zero incidences. The full incompressible Navier-Stokes equations have been solved numerically in the physical plane itself without using any transformation to the computational plane. A fully explicit scheme is adopted for the calculation of both cell face velocities and flow variables at the cell centers. For calculation of cell face velocities "Consistent Flux Reconstruction" (CFR) technique (Roy and

Bandyopadhyay, 2006a) has been formulated on unstructured grid which is based on solving the momentum equations at each cell face explicitly. The present CFR **uns**tructured grid solver would henceforth be referred as ‘CFRUNS’. The flux reconstruction cell on the face of the control volume is placed centrally between the two cells that share that face. The reconstruction volumes are chosen judiciously so that computational effort is still reduced as compared to the original method. A discrete pressure Poisson equation is obtained by substituting the reconstructed cell face velocity expressions in the discrete continuity equation. The pressure Poisson equation is solved iteratively using Gauss-Siedel method. The reconstructed cell face flux is substituted in both continuity and momentum equations. For updating the flow variables at the cell centers, the momentum equations are solved in an explicit manner in contrast to implicit scheme of Deng et al (1994).

The spatial discretization scheme used for ‘CFRUNS’ solver is second-order accurate. The spatial accuracy of the scheme is verified by solving Couette flow which has exact solution. Conservation of kinetic energy for collocated finite volume method in the present solver is also discussed. The errors are expected to be small. Validation of ‘CFRUNS’ solver has been carried out by comparing the present results for flow past single circular cylinder and NACA 0002 airfoil at different incidences with the experimental and numerical results available in the literature. The comparison is found to be satisfactory. In chapters 4-7, the ‘CFRUNS’ solver has been applied to solve a variety of problems involving various cylinder configurations and airfoils.

In **Chapter 4,** the solver has been applied to unconfined flow past two and three circular cylinders arranged in side-by-side and tandem arrangements. The two-cylinder flow bears similarity to flow over more-than-two cylinders in many aspects such as synchronization and merging of wakes, deflection and flip-flopping of the gap flows, and narrow-wide wake structures. However, at the same time, two cylinder flow may not be entirely representative of flows over more number of cylinders because they exhibit certain disparate behaviors as explained by Sumner et al. (1999) and Zhang and Zhou (2001). Therefore, numerical simulations are performed over three circular cylinders in side-by-side and tandem arrangements with various transverse and longitudinal gaps at Reynolds number (*Re)* = 100 and 200. The cylinder diameter is taken as the length scale for calculating Reynolds number. The time averaged lift and drag coefficients, vorticity contours, streamlines and Strouhal numbers have been presented for different cases. In order to get better understanding of the wake interference around multiple cylinders, the results obtained for flow past a single cylinder is taken as a reference for analysis. The results obtained for single or multi-cylinder cases have been validated with results from existing literature.

In **Chapter 5,** flow past two and three cylinders in staggered arrangements has been investigated. In this chapter, the main intention is to analyze the flow past two side-by-side cylinders under the influence of an upstream cylinder. The size of the upstream cylinder (can be called as control cylinder) can change as well as its axial location with respect to the two side-by-side cylinders. Simulations are carried out at Reynolds numbers 100 and 200 for three cases of multi-cylinder arrangements. It is observed that the nature of flow depends strongly on the arrangement of cylinders and the Reynolds number. A downstream cylinder located in the unsteady wake of an upstream cylinder experiences very large unsteady forces that may lead to wake-induced flutter. The Strouhal number based on the dominant frequency obtained from time history of lift coefficient attains the same value for all the cylinders for a given cylinder arrangement.

In **Chapter 6,** flow past single and tandem cylindrical bodies in the vicinity of a plane wall is discussed. The behaviour of the vortical wake created by square and circular cylinders placed in a laminar boundary-layer flow is investigated numerically. Calculations are performed at Reynolds numbers 100 and 200. Apart from Reynolds number, the flow field characteristics also depend on distance of the cylinder from the plane wall. To study the effect of gap (G) between the cylinder and the wall, calculations are performed at various gap ratios (g^*). Gap ratio is defined as the ratio of the gap (G) and the characteristic cross-stream dimension of the cylinder. In the case of square cylinder, the characteristic dimension is the length of a side of the cylinder (H) and in the case of a circular cylinder, the dimension is the cylinder diameter (D). The vorticity in the boundary layer formed over the plane wall interacts intensely with the vorticity associated with the shear layer emanating from the separation points on the cylinder surface and produces a complex flow field in the gap and the cylinder wake. An attempt has been made to analyze these complex flow features through numerical flow visualization tools like vorticity contours and streamlines.

In **Chapter 7,** some features of flow past airfoils at very low Reynolds number (*Re* = 1000) have been studied. Such low Reynolds number airfoil flows are practically relevant for small flying machines like micro air vehicles. The airfoil surface pressure distribution, lift and drag coefficient, pressure contours and streamlines are calculated for NACA 0012 and Eppler 340 airfoils. Analysis of the airfoil aerodynamic characteristics and flow separation phenomena is performed at different angles of attack ranging from 0^0 – 16^0 at *Re* = 1000. At low Reynolds number, the boundary layer on the airfoil is expected to be laminar which is sensitive to adverse pressure gradient and consequently it leads to large separation on the suction surface of the airfoil at moderate and high angles of attack. However, no abrupt stall is noticed within the given range of angle of attack.

Concluding remarks drawn from the present study along with future scope of the present work are provided in **Chapter 8**.

The following publications are achieved based on the work presented in the thesis:

1. Harichandan A.B. and Roy A. (2010), Numerical investigation of low Reynolds number flow past two and three circular cylinders using unstructured grid CFR scheme. *International Journal of Heat and Fluid Flow*, Vol. 31, pp. 154-171.

2. Harichandan A.B. and Roy A. (2010), CFR: A finite volume approach for computing incompressible viscous flow, *Journal of Applied Fluid Dynamics*, (accepted).

3. Haldar A., Ghosh. S., Harichandan A.B. and Roy A., Numerical investigation of incompressible low and moderate Reynolds number flow past some reflexed Eppler airfoils. *Aerospace Science and Technology*, (revised).

4. Harichandan A.B. and Roy A., Numerical prediction of incompressible unsteady flow past array of two dimensional circular cylinders placed in staggered arrangements. *Journal of Mechanical Engineering and Science*, (under review).

5. Harichandan A.B. and Roy A., Numerical prediction of incompressible unsteady flow past three circular cylinders of unequal diameters placed in staggered arrangements, *Journal of Applied Fluid Dynamics*, (under review).

6. Harichandan A.B. and Roy A., Numerical investigation of flow past two circular cylinders in tandem in the vicinity of a fixed wall. *Journal of Fluids and Structures, (under review).*

7. Harichandan A.B. and Roy A., Analysis of flows past airfoils at very low Reynolds numbers. *Journal of Aerospace Engineering*, (under review).

8. Harichandan A.B. and Roy A. (2010), Computation of incompressible flow past array of circular cylinders using an unstructured grid finite volume approach. Proceedings of the 20th National and 9th *International ISHMT-ASME Heat and Mass Transfer Conference*, January 4-6, IIT Bombay.

9. Harichandan A.B. and Roy A. (2010), A numerical study of vortex shedding from two circular cylinders in tandem with ground effect. Proceedings of the 37th National and 4th *International Conference on Fluid Mechanics and Fluid Power*, December 16-18, IIT Madras, (accepted).