Abstract

Flow past two and three dimensional bodies of different geometries has been under intense research since such geometries are encountered in many of the real life problems and industrial applications. In many areas of engineering, circular cylinders form the basic component of structures and machinery. For example; heat exchange tubes, cooling systems for nuclear power plants, offshore structures, cooling towers, transmission cables, submarines, pipelines and structural components of bridges under the effect of wind, etc. Separated flow behind such bodies leads to the formation of 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 forming the well known 'Von-Karman vortex street'. The surface pressure changes each time a vortex is shed from the cylinder. Therefore the cylinder is subjected to fluctuating drag and lift in addition to the mean force components. Such fluctuating forces may cause structural vibrations, acoustic noise and resonance, and in some cases can trigger structural failure or enhance undesirable flow mixing in the wake. Therefore it is very important to have a thorough understanding of the flow phenomena in order to appropriately control vortex shedding in practical engineering environments.

Two-dimensional analysis of flow past any arbitrary body geometry allows a broad spectrum of parameters to be considered and provides a baseline for more detailed studies. Depending on the particular application, three-dimensional flow effects may be significant, but the increased difficulty of the analysis would limit the scope of detailed parametric study. In the present research work, an incompressible unsteady viscous two-dimensional finite volume Navier-Stokes solver is developed to investigate flow past two-dimensional geometries. Both bluff geometries like circular and square cylinder as well as streamlined geometries like airfoils have been considered for the investigation. In this solver, the full Navier-Stokes equations have been solved numerically in the physical plane itself without using any transformation to the computational plane. A "Consistent Flux Reconstruction" (CFR) technique has been implemented on a collocated unstructured mesh comprising of triangular cells. The present CFR <u>**uns**</u>tructured grid solver would be called as 'CFRUNS'. This solver is applied to simulate unconfined flows around bluff bodies like single and multiple circular cylinders in side-by-side, tandem and staggered arrangements and also streamlined bodies like NACA0012 and Eppler-340 airfoil at different angles of attack. Flow past single and two-cylinders in close proximity to a plane wall are also calculated. Streamlines and vorticity contours are plotted to visualize instantaneous flow field. Some parameters characterizing the flow such as lift and drag coefficients and Strouhal number are also computed and quantitatively compared with results of other researchers. The results obtained for single circular cylinder and NACA0002 airfoil are primarily used for validating the present CFRUNS solver. Reasonably good comparison is obtained between the present results and results available from literature.

Keywords: incompressible Navier-Stokes solver, finite volume method, unstructured triangular mesh, circular cylinder, side-by-side arrangement, tandem arrangement, staggered arrangement, wake-induced flutter, Strouhal number, gap ratio, vortex-wrapping, wall shear layer, wake instability, drag polar, lift, drag, airfoil.