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 engineering applications. In many areas of engineering, circular cylinders form the basic component of structures and machinery. For example; nuclear power plant cooling systems, offshore structures, heat exchangers, cooling towers, submarines, transmission cables, 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, 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-Kármán 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. Under certain circumstances this 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 study. 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 two-dimensional finite volume Navier-Stokes solver is developed to investigate viscous flow past two-dimensional geometries. 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 unstructured grid solver has been named as CFRUNS. This solver is applied to simulate unconfined flow past two side by side cylinders by varying the center to center distance between the cylinders. The flow problems were analysed by keeping the cylinders stationary, rotating and rotationally oscillating about their respective axes. Streamlines, vorticity and λ_2 contours are plotted to visualize instantaneous flow field. POD is used to extract the coherent structures in the flow, Fast Fourier Transform (FFT) is used to obtain the frequency spectrum of the flow and the time variation of the flow is studied through the associated stress fields. Important 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 and square cylinder are primarily used for validating the present CFRUNS solver. Reasonably good comparison is obtained between the present results and results available from literature.

Keywords: finite volume method, incompressible Navier-Stokes solver, unstructured triangular mesh, circular cylinder, side-by-side arrangement, Strouhal number, lift, drag, POD.