SYNOPSIS

Thesis Title : CFD STUDIES OF FLUID- STRUCTURE INTERACTION EFFECTS ON RECTANGULAR SECTIONS

Student Name: Keerthana. M

Roll Number : 32EE12J41006

Batch : 2009- 2011

Thesis Supervisor : Dr.P.Harikrishna, Senior Principal Scientist, CSIR- SERC.

The wind flow characteristics around buildings/ structures have to be well understood in order to accurately evaluate the wind induced aerodynamics forces/ pressures acting on them. Boundary layer wind tunnel (BLWT) studies have provided a gamut of information regarding the flow and flow induced pressures which have been used as guidelines in the codes of practice and standards and it is currently the reliable tool for the evaluation of aerodynamics characteristics of building. Recently, advancements in computational fluid dynamics (CFD) and numerical modeling techniques with the aid of high performance computing facilities have lead to simulation of the flow field around buildings/ structures to get temporal and spatial variation of velocity and pressure distributions. Turbulence modeling is a key challenge in applying CFD for wind engineering applications. So far there is no universal turbulence model that suits all types of flow problems. The present study focuses on fluid-structure interaction effects on 2-D rectangular sections using CFD in order to validate the popular turbulence models which are used in practice.

Interaction of movable / deformable structure with the surrounding flow field is a coupled problem and is of practical significance in specific cases as structural motion in fluid flow induces additional aerodynamics forces. Aero elastics instabilities like galloping and flutter occur in certain types of lightweight, flexible structures with low damping like suspension and cable stayed bridge decks. Quasi-steady assumption helps in assessing the aero elastic stability of a structure by conducting wind tunnel studies on rigid sectional models at different angles of wing incidence.

In a similar way, smooth flows around 2-D rectangular sections at various angles of wind incidence will be useful to assess the aero elastic stability. Unsteady 2-D flow around rectangular sections with side ratios of 2 and 5 have been numerically simulated for (0° to 10°) using two of the popular RANS turbulence models, namely Realizable k-e (RKE) and Shear Stress Transport k-e model (SST). The numerically evaluated aerodynamics force coefficients have been used to assess the stability of the section against transverse galloping based on Den Hartog's criteria. Further, lock-in behavior of rectangular section with side ratio of 2 subjected to forced oscillation (sinusoidal) of amplitude 0.1 D (where D is the dimension of rectangular section normal to the flow) over a range of reduced velocities has also been studied with the dynamics meshing capabilities.