Chapter 1

Introduction

1.1 Scope of the present work

With rapid advance of computer technology, computational methods have become a powerful tool for solving diverse engineering problems of complex nature. The popularity of computational method is due to the fact that experimental techniques are quite time consuming, costly and expensive. This is particularly true in problems related to wind engineering. In cases where parametric variations need to be done experimental methods are quite laborious and time consuming. Further it is sometimes possible to simulate full scale boundary conditions in numerical method.

Computational methods have been applied in wind engineering to study wind flow pattern around buildings or a group of buildings with a view to understand flow interference effects and its relation to pollution dispersion, pedestrian comfort, ventilation in the building etc. Aerodynamic forces on the buildings have also been predicted through numerical simulation. The main complications of using numerical method arise due to bluff body shape of the structures with sharp corners in contrast with streamline bodies used in aerospace applications. Complicated flow fields around buildings consisting of impingement, flow separation, streamline curvature, reattachment and vortex formation remains the most challenging problem for computational specialists to tackle. Further complications arise due to presence of turbulence. Anisotropic strain rates that develop on the body lead to complicated turbulence characteristics which have put a question mark on kind of modeling to be used for turbulence simulation.

In the framework of computational wind engineering, which deals with the application of CFD methodologies in the classical wind engineering and building aerodynamics problems, numerical solutions [Stathopoulos, 1997] have been developed with the potential to overcome these limitations. However, such solutions have not yet fully demonstrated their ability even for simple building geometries exposed to strong horizontal winds. On the other hand, numerical prediction of fluid flow and their effects on the solid bodies have developed to the point where commercial programs are available for use in many cases. CFD is a good friend of computational wind engineering. There are several areas of improvement. The most significant areas are: (a) Numerical accuracy by using higher order approximations coupled with grid-independence checks; (b) Boundary conditions, which depends on the specific problem under consideration so that they require good physical insight and high level of expertise; (c) Refined turbulence models although adhoc turbulence modifications are unlikely to perform well beyond the specific flow conditions for which they have been made. There is a certainly interactions between these three areas, for instance, more advance turbulence modeling generally require denser meshes for a given level of accuracy.

The evolution of numerical methods, especially finite difference method for solving ordinary and partial differential equations began at the turn of the 20th century. At the same time progress was being made on the development of methods for both elliptic and parabolic problems. Frankel presented the first version of successive over relaxation (SOR) scheme for solving Laplace equation. This provided a significant improvement in the convergence rate. The vision of numerical simulation to accurately model complicated phenomena, perhaps, to the point of replacing expensive experiments has yet to fully realize in fluid dynamics. Although there have been dramatic advances in computer power there have been significant but less such advances in numerical methodologies.

Some of the relevant work in the field of computational fluid dynamics applied to wind engineering is done by Launder [1974] in which vorticity transport equation is solved simultaneously with fluid flow equation, Roache[1976] deals at length with various finite difference method for solving fluid flow problems. This work discusses various stability and convergence criteria. The response of structure to basic wind forces have been extensively covered while analyzing the present state of art in each field in a similar manner by Sachs [1978]. An overall view of the major field in computational fluid dynamics and heat transfer with various techniques is presented by Anderson [1984]. E.H.Mathew [1987] used the primitive variable formulation of the Navier – Stokes equations to predict wind generated disturbances around buildings in elevation for the purpose of calculating natural ventilation flow rates. While A. Paterson [1990] showed that flow is simulated on computer by the solution of Reynolds equation and continuity equation with the help of a k-ε turbulence model. The diagnostic system for assessing the results of numerical error estimation by Murakami (1990) and Ali Quasim's (1992) 2D flow analysis over field facility building located at Texas Tech University are great contributions to the field.

The main contributions of the present work are listed as follows:-

(i) Development of a numerical code based on 2D Navier-Stokes equations with LES turbulence model using vorticity-stream function formulation to study the wind flow behavior over building structures in tandem arrangements. For more realistic and applicable to the actual design of buildings arranged in certain way one should carried out 3D CFD simulations, but due to computer limitation author concentrated on 2D simulations.

(ii) The code has been used to study streamline pattern of the flow field, velocity profiles at various stations–upstream, roof and downstream of the building structures and Cp distribution on the surfaces of the buildings. The following cases have been considered :-

(a) Flow over single prismatic building of different height to width ratio.
(b) Flow over two successive prismatic building combinations of different height to width ratio placed in different sequences.

(iii) Experimental simulation has been performed by conducting experiments in the industrial wind tunnel facility available in the department to obtain surface pressure co-efficient Cp for comparing with the predicted Cp from the code.

(iv) Drag co-efficient and lift co-efficient have been evaluated by integrating Cp distribution on the surface of the buildings.

(v) Using the code flow field studies have been conducted for three buildings in tandem for a few typical cases.

1.2 Documentation Outline

Introduction to the thesis is given in **Chapter 1**, which states the scope of the present work and its purpose and motivation to do this work. Its outlines the importance of the problem. A brief **Review of Literature** is presented in the **Chapter 2**, which concentrates on numerical modeling of wind flow over building structures, introduces the physical complexities of turbulent flow around buildings. The description of the **Experimental Measurement** used in the present study is given in **Chapter 3**. **Chapter 4** deals with **Numerical Simulation Based on LES**, that describes the development of an LES algorithm for 2D flows based on combination of an unsteady stream-function/vorticity method for flow solution coupled to Smagorinsky type sub-grid type models and validation of the code. **Chapter 5** describes the **Results and Discussions** that basically throws light on the effects of arrangement and inter-block spacing on the flow around a cluster of more than one 2D building block. **Chapter 6** sums up the work with **Conclusions**.