INTRODUCTION

The development of numerical procedure for solving Euler and Navier-Stokes equations for high-speed compressible flows is presented here. Overviews of supersonic and hypersonic flows over different geometries, shock diffraction and shock-vortex interaction, shock-shock interaction and hypersonic flows with air chemistry are provided in this chapter. Motivations of the present work, objective and scope of the present thesis are also highlighted here. The last section describes the layout of the thesis.

1.1 Development of numerical solvers

The study of high-speed compressible flows has been a fertile field of investigation for several decades. The basic characteristics of this kind of flows are the presence of shock waves, expansion waves and their interactions. In addition, the shock-shock interaction, shock wave diffraction and high temperature effects in hypersonic flows make the problem more complex. Sometimes it becomes a difficult task to do the ground-test since the test times are extremely brief. The high-total enthalpies associated with the hypervelocity flow field are also major concerns in conducting the experimentation. An alternative tool is to accomplish the task by developing suitable numerical simulation with proper modeling.

Development of Euler and Navier-Stokes solvers, particularly for high-speed applications, is a challenging task. The present thesis aims to achieve a certain degree of maturity in high speed solver development before trying to solve challenging research problems involved in high speed vehicle design. This is an important step since experimental data are relatively scarce in the hypersonic flow regime and the numerical analyst has to use his experience and insight while solving a problem where few or no supporting data exist. Such insight can only be developed by careful solver development and its application to problems where supporting data - be it numerical or experimental - are available. The author has attempted such an exercise in the current work.

An important aspect of this work has been the combination of a low dissipation scheme, the *AUSM+*, with an unstructured grid based solution procedure. The low dissipation of *AUSM+* makes it suitable for capturing sharp flow gradients, separation bubbles and high heat fluxes and also unsteady phenomena and the unstructured grid provides the much needed flexibility to handle complicated geometry with ease.

The first part of the present work aims to develop finite volume numerical methodologies to solve the Euler and Navier-Stokes equations for high-speed compressible flow problems. The inviscid terms are modeled using *AUSM+* flux splitting and *MUSCL* algorithms. For the viscous diffusive terms, a symmetric averaging approximation is used.

1.2 Overview of applications of the developed numerical methods

High-speed compressible flows (supersonic to hypersonic) around bluff bodies like wedge, cone and conical/cylindrical bodies with hemispherical nose have been a topic of intense research for last several years due to their significance in engineering application. In recent time, this research has gained more momentum as a result of ever increasing computational power and advent of modern experimental techniques. The present analysis focuses on the supersonic flow past a wedge considering and without considering viscous effects at a free stream of Mach number 3. The development of oblique shock, expansion fans and the wake is discussed in details. The appearance of tail shock, recirculation zone and mixed flow regime in the wake region make the flow quite complex and interesting study. The high speed compressible flow over a blunt slab with semicircular nose is considered to show the development of strong bow shock waves and the change of flow properties across the bow shock. Three different free streams with Mach numbers of 3, 6 and 10 are selected. The variations of flow properties along the stagnation streamline and over the surface of the body are discussed. A perfect diatomic gas model is considered for numerical calculation of supersonic flows over the bluff bodies.

Reflection and diffraction of shock waves, shock-vortex interaction, shock-shock and shock-expansion interactions are important fundamental problems in high speed compressible flow and studies on these problems are important from academic as well as application point of view. These problems form the building blocks of all high speed flow applications. Shock diffraction problems have special significance in the prediction of blast wave interaction with structures. Vorticity production and shock-vortex interaction are normal phenomena in shock diffraction. Identification and evolution of the vortices, shock-vortex interaction, vorticity production and rate of circulation production are important features of the flow which demand proper attention. The shock-vortex interaction is usually referred as a two-stage phenomenon. The main core vortex develops in the wake region of the flow in the initial stage. The shape of this core vortex is initially circular and with the progresses of time it becomes elliptical. At the end of the first stage the vortexlets (Kelvin-Helmholtz vortices) start to develop and start a spiral journey towards the main vortex core. The violent mixing of these KH vortices with the core vortex is an important feature in shock diffraction phenomena. A complex shock structure evolves as the shock interacts with the core vortex and the vortexlets. The interaction of the shock with the vortexlets makes up the second stage of the interaction process. The rapidly changing flow pattern creates an intense acoustic field. The complex structure of the flow field and the consequent rapid and large changes in the flow, both in time and space, make the proper resolution of the flow extremely difficult. The shock diffraction over a 90^0 convex corner and a wedge (Schardin's problem) is studied with a high-resolution structured Navier-Stokes solver to capture the flow features. Vorticity and circulation production rates and acoustic source field are computed.

High-speed supersonic and hypersonic flows over double-wedge geometry constitute a class of important fundamental problem in high-speed flow. The flow field is characterized by different types of shock-shock interactions or asymmetric shock wave reflection phenomena. Different type of shock-shock interactions are termed as Type-VI, Type-V, Type-IV, etc. as classified by Edney (1968). Depending on the strength or type of interaction the flow field may also contain number of subsonic pockets. The parameters that control the type of shock-shock interaction include the stream Mach number, two wedge angles and ratio of the face lengths. The complex shock structure introduces large changes/jumps in the flow properties in quick succession within the interaction zone. The rapid changes in entropy across a number of shocks make the flow rotational. Due to the complex nature of shock interaction and rapidly varying flow field, high-speed flow past double-wedge poses a serious challenge to accurate numerical prediction. In the present study, Type-VI and Type-V shock interactions at free stream Mach number of 9 are studied numerically using an unstructured Euler solution. Transition from Type-VI to Type-V interaction and both six-shock and seven-shock Type-V interactions are captured by changing the second wedge angle while keeping the other parameters fixed.

The temperature in high Mach number flow usually reaches quite high and chemical reactions among the species of air can occur. In such a situation, the air cannot be reasonably assumed a perfect gas and the effects of the chemical reactions on flow dynamics need to be reasonably incorporated. The chemically reacting flow in comparison with perfect gas flow brings down the temperature significantly while the density is raised. This has considerable influence on the overall flow. Since the residence time in high-speed flow is small, the chemical reactions are usually in non-equilibrium. Due to the complexity of incorporating non-equilibrium chemistry, often the chemical reactions are assumed in equilibrium. There are many ways to incorporate equilibrium air chemistry effects and can easily be included in a perfect gas solver. The effective γ approach is found quite attracting in this context. An iterative method is required to compute the concentration of the species involved and pressure at each point. The effective γ can be obtained from an appropriate polynomial curve fit. In the present work the equilibrium air chemistry effects are incorporated in the developed unstructured Euler solver using an equivalent γ method. Five species air with three chemical reactions model is considered. Flow over a blunt slab with semicircular nose is simulated and the results are compared with the results from perfect gas model. The shock off distance reduces significantly and the temperature in the flow field also reduces drastically.

1.3 Motivation of the present work

High-speed compressible flows encompass a wide range of practical applications. The complexity of the flow topologies and nonlinear behavior of the flow parameters in high-speed flow over bluff bodies that include diffraction of shock wave over different geometries, shock-vortex interactions and shock-shock interaction phenomena are the source of motivation for the present research work.

1.4 Objectives and scope of the present thesis

The objective of the present work is to develop two-dimensional finite volume Unstructured Euler (UnsEuler) solver with and without air chemistry, Unstructured Navier Stokes (UnsNS) solver and Structured Navier-Stokes (StrNS) solver suitable for high-speed compressible flows using the *AUSM+* flux splitting method with appropriate reconstruction and limiter. It is also aimed to achieve a better understanding of the shock diffraction and complex shock-shock interaction problems.

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

1. Development of the finite volume solvers UnsEuler and UnsNS implemented on triangular cells and StrNS on quadrilateral cells.
2. Application of the above solvers to various two-dimensional problems:
    - Inviscid and viscous flow past a wedge to investigate oblique shock development, supersonic flow separation and wake.
    - Investigation of flow over a blunt slab with semicircular nose.
    - Analysis of shock diffraction problem on different geometries.
    - Study of shock interaction problem in flow over double wedge.
    - Investigation of flow over blunt slab considering equilibrium air chemistry.

1.5 Layout of the thesis

The thesis discusses the development of numerical solvers based on finite volume method using structured and unstructured meshes for different high-speed compressible flow problems. Chapter 1 introduces the development of numerical solvers; the overview of the investigations undertaken for different applications in this thesis and motivation of the present work followed by the objectives and scope of the present thesis.

Chapter 2 briefly reviews the literature related to the solution of compressible Euler and Navier-Stokes equations for high-speed applications. Brief review of works relevant to high-speed flow over different bluff bodies, shock reflection and diffraction, shock-vortex and shock-shock interactions and hypersonic reacting flow problems is also included.

In Chapter 3 different flux difference and flux vector splitting methods are discussed. The development of the present finite volume numerical solvers is presented. The developed unstructured and structured solvers are validated and some grid refinement study is presented in this chapter.

High-speed inviscid and viscous flows over wedge and slab with semi-circular nose are studied in Chapter 4. The shock diffraction and shock-vortex interaction in Schardin problem are studied in Chapter 5 using the developed structured Navier-Stokes solver.

Chapter 6 deals with the shock-shock interaction problem in hypersonic inviscid double-wedge flows. Chapter 7 presents an account of inviscid equilibrium reacting flow over a blunt slab. Concluding remarks drawn from the present study and some recommendations for future work are provided in Chapter 8.