CHAPTER I

MOTIVATIONS, CONCERNS, AND BACKGROUND

I.1 Introduction

During the 1950s and 1960s, the computation of hypersonic flowfields was an area of active research, but it subsequently languished. Recently it has seen a resurgence on a national scale due to the proposed development of several vehicles that would travel in the atmosphere at hypersonic speeds. To design these craft, their aerothermodynamics must be predicted. However, ground-based testing facilities such as shock tunnels, ballistic ranges, and arc-jets cannot produce the combination of free-stream velocity, density and model geometry to exactly mimic many flight conditions. Thus computational methods are required that accurately model the physics of a hypersonic flowfield. These flows are non-ideal, in that they are vibrationally and electronically excited, chemically reacting, and ionizing, all of which must be included in the description of the gas. Therefore it is not possible to directly apply standard computational fluid dynamics methods to these flowfields.

This dissertation represents an attempt to make such a progression and involves the solution of a set of equations that is more complex and encompasses more of the relevant physics than any prior work. Previous researchers have reported techniques that treat one or several of these hypersonic phenomena for specific flowfields, but a method that is capable of predicting the hypersonic flow over an arbitrary body has not been developed. However, with the greater accessibility of supercomputers and the development of more efficient numerical algorithms make it possible to extend this research.

In this chapter we will briefly discuss the features of the proposed hypersonic vehicles and the nature of their flowfields. We also consider the influence of the non-ideal physical phenomena on the vehicles' aerothermodynamics. The previous research in the field is briefly summarized and its relationship to the present work are highlighted.

I.2 Motivations

The field of hypersonic aerodynamics has seen a rebirth in recent years because of the proposed development of several hypersonic vehicles. The design of these craft will depend on the prediction of the severe aerothermal loading that they will experience in their flight through the atmosphere. Three of the proposed configurations, the AOTV, the TAV, and the second generation Space Shuttle are discussed below[1].

The Aero-assisted Orbital Transfer Vehicle (AOTV) would act as a space-ferry which would transport payloads to and from near-earth orbit and geo-synchronous or other orbits. On return from outer-space it would fly into the atmosphere, slow down to low-earth orbital speeds using aerodynamic drag, and then maneuver to rendezvous with the Space Shuttle. Along this trajectory, which must be accurately predicted, the vehicle would experience large aerothermal loads including radiative heating. The correct analysis of both forms of heat transfer is required for the design of the minimum weight heat shield.

The Trans-Atmospheric Vehicle (TAV) or National Aero-Space Plane (NASP) would be a high lift over drag transport that would takeoff from a runway, fly into low earth orbit, and finally land, all under its own power. It would be subjected to long periods of moderate thermal loading, making the total heat pulse significant. The drag of the vehicle is strongly dependent on the state of the boundary layer and the point where turbulent transition occurs. And the design of the propulsion system requires an accurate knowledge of turbulent mixing and combustion at supersonic speeds.

The second-generation Space Shuttle would replace the current shuttles and would be a manned reusable booster for lifting large payloads from earth to low earth orbit. This vehicle also would have significant aerothermal loading during re-entry and its aerodynamics must be known for its stability and control.

These three vehicles are similar in that their trajectories would take them through the atmosphere at hypervelocities. However, they differ from previous hypersonic vehicles such as the current Space Shuttle and Apollo because their missions require them to be as efficient as possible and completely reusable. Any extra weight devoted to the heat shield or an inefficient aerodynamic design reduces the payload and may cause the vehicle to lose cost-effectiveness or even the ability to fly. Thus a method that may be used to predict

[1] See also Howe (1985) and Anderson (1984) for further details.

the aerothermodynamics of these hypersonic vehicles is a necessary design tool.

Another and related motivation for the development of a numerical technique for the computation of non-ideal hypersonic flowfields is to explore the accuracy of different physical models of high temperature air and to test the quality of simplifying assumptions. Many excitation and reaction models that are used in high temperature gas dynamic calculations have been calibrated in ground test facilities where the operating temperatures are much below those encountered in flight. Thus they are essentially unproven for the regime of interest and must be tested before they can be used with confidence. A numerical method that solves a model that includes many of the relevant physical phenomena can be used as a test-bed for the existing models and for the development of new or improved modeling approaches. Ideally the numerics would be trustworthy enough so that they could be removed from consideration as a possible source of error and the implications of the particular physical model could be studied by themselves.

I.3 Relevant Physics of a Hypersonic Flow

The vehicles discussed above would fly at hypervelocities where the assumptions of a perfect and inviscid gas used in classical aerodynamics do not apply. The shock layers that envelop these craft are characterized by thick boundary layers. Furthermore, the gas is at high temperature which causes reactions and thermal excitation. Thus the air is not calorically perfect and inviscid. The implications of these non-ideal effects on the design of hypersonic vehicles are discussed in this section.

The thickness of the boundary layers on hypersonic vehicles can be large relative to the thickness of the shock layer. This can be seen by considering that the laminar boundary layer thickness, δ, on a flat plate grows like (Anderson (1984))

$$\delta \sim \mathcal{M}_\infty^2 / \sqrt{Re_x}, \tag{1.3.1}$$

where $\mathcal{M}_\infty$ is the free-stream Mach number and Re_x is the local Reynolds number. Clearly for large $\mathcal{M}_\infty$ the boundary layer can occupy a significant portion of the flowfield and, in some cases, may merge with the bow shock wave itself. The interaction with the inviscid region of the flow influences the stability of the boundary layer and the location of the shock wave. This can have a major affect on the lift, drag, and stability of a hypersonic vehicle.

Not only is the boundary layer thick, but it is also hot because of the conversion of kinetic energy to thermal energy because the gas stagnates at the wall. Also gas that has passed through a strong shock wave will be at an elevated temperature. The temperatures in the boundary layer and some parts of the inviscid region are large enough to cause chemical reaction and thermal excitation of the gas. These processes occur at finite rates, which when coupled with the large convection speeds within the shock layer, result in a state of thermo-chemical nonequilibrium. That is, a reaction or a thermal relaxation process may be initiated at one point in the flow, but by the time it has progressed appreciably, the fluid has been convected to a new location. Maus *et al.* (1984) postulate that this state of nonequilibrium has a significant affect on the pitching moment of the Space Shuttle and would be important for other hypersonic vehicles. The degree of reaction and thermal excitation has a large influence on the temperature distribution in the flowfield, and, as a result, on the heat transfer to the body.

The vibrational and electronic state of the gas and the concentration of highly radiative species have a large influence on the amount of radiation emitted from the shock layer. The degree to which these non-ideal thermal modes are excited determines the importance of radiative heating to the vehicle.

Thus the shock layer around a hypersonic vehicle is characterized by thick, high temperature boundary layers and hot inviscid regions where significant thermal excitation and chemical reactions occur. In many cases the time scales of the reactions and relaxation are similar to the fluid time scales and a state of thermo-chemical nonequilibrium is present. These non-ideal effects must be included in an aerothermodynamic analysis of a hypersonic vehicle because they dominate the gas dynamics and the aerothermal loading. Solution techniques that do not include them will yield erroneous results and an inaccurate description of the flowfield.

I.4 Previous Research

Since the 1950s researchers have made attempts to solve flowfields in which the hypersonic phenomena discussed above are important. Much effort was devoted to the solution of a subset of the equations that govern such a flow. A brief outline of some of this work is given in this section, with particular emphasis on how it is relevant to the current research.

One major thrust of past research was centered around the solution of the flowfield equations for a reacting gas on the stagnation streamline of a blunt body. Fay and Riddell (1958) outlined this technique and developed several useful relations for predicting the stagnation point heat transfer. This method was extended to include the effects of ionization, radiation and even ablation of the surface[2]. An extension of this work to chemically reacting boundary layers was made by Blottner (1964) and thus the flowfield solution could be obtained along the surface of the body. However, this technique requires the specification of the boundary layer edge conditions which, for general body geometries are not known. Maslen (1964) used an inverse method to compute the body geometry that would generate a given shock shape in an inviscid flow, but it was not until 1966 that Moretti and Abbett devised a technique that solved general inviscid flowfields. Their approach was to solve the time-dependent equation set for its asymptotic stead-state. This method is currently used by many researchers and was employed in this work also. In spite of this effort, no one method was developed that could be used for analyzing the complete flowfield of a hypersonic vehicle.

These investigators were instrumental in the genesis of current hypersonic research. For example, they discovered the effect of a catalytic wall on heat transfer, the rudiments of computing radiation from hypersonic shock layers, and the way in which ablative material may be used to protect a re-entry vehicle[3]. A number of vehicle designs were based, at least partially, on these calculations.

However, a number of things have changed since the 1950s. Great advances have been made in computer power and memory. Also a large amount of experience has been amassed on the use of computational fluid dynamics for perfect gas and equilibrium flowfields. The introduction of flux-splitting by Steger and Warming (1979) and the development of efficient numerical methods has given current researchers new tools to make the solution of general hypersonic flowfields possible.

Previous authors in the field of modern computational hypersonic fluid dynamics

[2] See, for example, Howe and Viegas (1963).
[3] See Fay and Riddell (1958) and Goulard (1958)

have taken one of two standard approaches to solving the equation set that governs a nonequilibrium hypersonic flowfield. One approach is to uncouple the equation set from the chemical reaction and thermal excitation rate equations and solve the two equation sets independently[4]. The chemistry influences the fluid variables after each iteration or series of iterations. The uncoupled method has the advantage that it is relatively easy to implement and may be used with an existing Navier-Stokes equations solution algorithm. However, problems have been encountered with maintaining numerical stability and achieving convergence to a steady-state result. The second approach is to solve the entire equation set simultaneously, or in a so-called fully coupled fashion. The advantage of this technique is that the chemistry, which can have a first-order effect on the flowfield, influences the fluid dynamics directly. However, this method tends to be computationally intensive. Eberhardt and Brown (1986) developed a fully-coupled flux-split technique for computing one-dimensional chemically relaxing flows through shock waves. Bussing and Murman (1985) used a method of treating the chemical rate terms implicitly and the fluid mechanics explicitly in their calculation of hydrogen-air combustion. Neither of these methods nor those of other researchers implement sufficient complexity to the thermo-chemical state of the modeled gas to allow them to be used for hypersonic vehicle calculations under severe aerothermal loading.

I.5 The Scope of the Current Work

This dissertation is an attempt to build on the previous work in hypersonic aerodynamics through the application of recent advances in computational techniques. A minimum of assumptions are made in the derivation of the governing equations so that a general flowfield may be considered. The work presented here may be applied to an arbitrary two-dimensional or axisymmetric blunt body with a shock layer that is chemically reacting, vibrationally relaxing, electronically excited, and weakly ionizing. The reactions and relaxation processes are allowed to occur at finite rates and thus a truly thermo-chemical nonequilibrium flowfield may be analyzed, as well as one whose thermo-chemical state is either frozen or in equilibrium. The fully coupled equation set is integrated in time until a steady-state is reached using a numerical method that was developed by MacCormack

[4] See, for example, Gnoffo and McCandless (1986) and Li (1986)

(1985). This technique is completely implicit and uses Gauss-Seidel line-relaxation to reduce the computational cost of the solution.

The work is novel, in that the model of high temperature air includes more of the relevant physics and, consequently, is more accurate than previous work. This is the first time that thermal nonequilibrium and ionization have been computed in a general multi-dimensional flowfield. It is also the first time that a fully implicit, flux-split numerical method has been applied to such a flowfield. We will see that the results support the current physical model of high temperature air and that the numerical method converges to a steady-state solution in roughly the same number of iterations as an implicit perfect gas Navier-Stokes algorithm.

.