SCRAMJET PERFORMANCE

Griffin Y. Anderson
Charles R. McClintock
John P. Weidner
NASA Langley Research Center

ABSTRACT

Fundamental scramjet propulsion research, conducted by NACA/NASA since the 1940’s, has been centered mainly at the Langley Research Center located in Hampton, Virginia. Since the 1960’s, this effort has been focused on the development of airframe-integrated engine concepts with high-performance potential at speeds approaching orbital velocity. Thus, this research effort is conducted in order to support the incorporation of air-breathing propulsion technology within future orbital transportation systems. This chapter presents an overview of the NASA research effort with special emphasis on the problems of engine-airframe integration, ground simulation of high-speed combustor flows, fuel injector/combustor design issues, and performance projection at high hypersonic speeds.

INTRODUCTION

Interest at NASA (then NACA) in technology relevant to high-speed propulsion and scramjets in particular extends back into the late 1940’s and early 1950’s. Various theoretical (refs. 1 and 2) and experimental (refs. 3 and 4) studies were reported, including an early comprehensive ideal-gas cycle-analysis assessment of scramjet performance potential in 1958 (ref. 5). The period from the mid-1950’s to mid-1960’s can be appropriately thought of as "Aerospace Plane I," where a substantial investment was made, primarily by the U.S. Air Force, in system studies and technology relevant to air-breathing orbital propulsion. With the creation of NASA in 1958 and the subsequent refocusing of U.S. aerospace resources on expendable rocket-powered propulsion for lunar exploration, the termination and/or limitation of most hypersonic air-breathing research inevitably resulted. The NASA Hypersonic Research Engine (HRE) Program was a notable exception and resulted in the demonstration of high levels of dual-mode scramjet engine ground-based performance for Mach 5-7 conditions, and also successfully demonstrated fabrication techniques, aerodynamic performance and robust survival of regeneratively cooled engine structures at Mach 7 conditions (ref. 6). While completing the HRE Program, NASA restructured a modest research effort at the Langley Research Center to address issues relevant to airframe-integrated scramjet propulsion systems (ref. 7). The culmination of this effort was the demonstration of an adequate fixed-geometry engine performance level capable of accelerating a candidate conceptual manned-research airplane through the Mach 4-7 speed range. This successful demonstration of a ground-base technology, extending into the hypersonic speed range, became part of the supporting data for embarking on the National Aero-Space Plane (NASP) Program in the 1980’s. This latter effort marked the first major allocation of significant resources applied to hypersonic air-breathing flight technology since the 1960’s. The NASP Program was quite sharply focused on a single-stage-to-orbit (SSTO) concept; and thus, from an historical perspective, NASP may be appropriately considered as "Aerospace Plane II." Thus, in the 1980’s and 1990’s, the Aerospace Plane II effort has centered around the development of an advanced-space-launch system which could reduce the cost and increase the operational flexibility of inserting, on demand, small high-value payloads into orbit, thereby making space more readily accessible for a variety of missions.

The fundamental differences between the ramjet and scramjet cycle should be examined before proceeding to a more detailed treatment of the scramjet propulsion system operating at hypervelocity speeds. In general, any hypersonic air-breathing vehicle must maintain a flight trajectory within the atmosphere that reflects heating and structural load limits while delivering enough airflow to the propulsion system to maintain an adequate net-thrust level for acceleration. At flight speeds exceeding approximately Mach 3, since the turbine inlet temperature is the primary limiting factor in achieving high specific thrust (classical Brayton cycle), the static temperature developed in the combustor must be kept well below that of the flight stagnation conditions. Concurrently, the compressor is no longer required to achieve adequate cycle-pressure ratio and hence the ramjet becomes the propulsion cycle of choice. To
achieve hypersonic speeds (Mach greater than 5), active regenerative cooling utilizing the heat-sink capacity of fuel is needed; however, typically the performance penalty associated with cooling large internal surface areas is nontrivial, and hence becomes a serious design consideration. As flight speed is further increased, the elevated stagnation temperature and pressure precludes use of the ramjet cycle. Additionally, the thermal management task is further stressed by the increasing aggregate component heat load, and hence cryogenic hydrogen fuel technology is generally considered. As flight speed is still further increased, the supersonic combustion ramjet (scramjet) cycle becomes the engine cycle of choice, since it does not require excessive speed reduction of the high-enthalpy mainstream airflow. Therefore, operation over a wide flight-speed range dictates an engine capable of operating as both a ramjet and as a scramjet, and hence typically requires variable geometry.

Fundamental differences between these two cycles are highlighted in figure 1. The normal or strong shock system required in the operation of a subsonic-combustion ramjet, located just downstream of the inlet throat, is stabilized by backpressure generated by choking the engine flow in the exhaust nozzle. To maximize ramjet cycle performance: the entrance-flow Mach number must be in the range of Mach 1.4 to 1.8 at the beginning of the shock system, the inlet contraction ratio must be relatively large, and boundary-layer bleed may be needed to prevent inlet unstart and/or to stabilize the shock system. The net result is a higher inlet total-pressure loss compared with that generated by the scramjet cycle, which avoids the subsonic-normal-shock conditions despite operating at a lower inlet contraction ratio. The major advantage of the ramjet cycle is comparatively small Raleigh losses associated with the combustion process. This trade-off between inlet-pressure recovery and combustion losses governs the comparison of fuel-specific impulse between these propulsion cycles. Fuel-specific impulse data, presented in figure 2, is one performance discriminator between the two cycles, and typically indicates a benefit for operating the ramjet into the Mach 6 or 7 regime; however, the internal-duct-static pressure is a conflicting discriminator (illustrated in fig. 3) with the latter being constrained by overall system and engine weight. For the data presented, the ramjet duct pressure within the subsonic diffuser was calculated assuming an inlet-kinetic-energy efficiency of 0.95, and is seen to increase rapidly as a function of Mach number. Additionally, accelerator-type vehicles, such as those envisioned for space launch applications, tend to optimize at high flight-dynamic pressures typically on the order of 2000 psf or more, further dictating high values for the internal-duct-static pressure. Rather than degrading ramjet performance through reduced pressure recovery, transitioning engine mode operation to a scramjet cycle would effectively reduce internal-engine-static pressure by maintaining supersonic flow throughout the engine. A dual-mode engine, operating as a ramjet at low Mach numbers, which smoothly transitions to a scramjet cycle as flight speed is increased, would satisfy these general criteria. Compromises in ramjet performance of the dual-mode engine could be potentially offset by a shorter diffuser and use of a thermal choke rather than a mechanical choke at the nozzle throat. The development and demonstration of such an engine was the objective of the NASA Hypersonic Research Engine (HRE) Project started in the 1960's (ref. 6 and 8). In ground-facility tests, the HRE successfully demonstrated dual-mode ramjet and scramjet cycle operation at Mach 5, 6, and 7 and, in addition, demonstrated flight-weight actively cooled structures at Mach 7 flight conditions. More contemporary examples of dual-mode scramjet engines are described in Chapter 1 of this volume.

As speed increases, the specific impulse of the scramjet degrades to finally approach that of a rocket (fig. 2), at which point (or perhaps even earlier) the air-breathing engine has no advantage and the vehicle would transition to rocket propulsion and accelerate out of the atmosphere. Performance of the air-breathing vehicle is thus improved over a rocket powered vehicle by extending the useful operation of the scramjet engine to as high a flight speed as possible. Extending operation of a hypersonic vehicle to hypervelocity speeds puts extreme demands on the propulsion system. This system must survive not only a very severe thermal environment, but must also operate at a high level of efficiency while producing enough net thrust to adequately accelerate the vehicle. Additionally, at higher speeds component efficiency becomes much more critical, a consequence of the increasing quantity of energy resident in the airflow processed by the engine (fig. 4). For example, at extreme hypervelocity speeds, the level of energy contained within the engine airflow becomes much larger than the energy added by burning hydrogen fuel; therefore, the net thrust (the difference between gross thrust and ram drag) becomes only a small fraction of the processed-engine-airflow stream thrust. Specifically, at Mach 6 the energy associated with heat release of a stoichiometric hydrogen fuel-to-air mixture is about twice that of the kinetic energy contained in the air stream, whereas at Mach 16 this value is only about one-fourth. At the same time, additional factors contribute to increased performance sensitivity: the high inlet contraction
ratio (required to compress the air at high altitudes), high temperatures (to be contained within the engine) and high fluid flowpath velocities. Consequently, component-efficiency sensitivity rapidly increases with increasing vehicle speed throughout the hypersonic flight regime. The following sections of this chapter examine loss mechanisms and design challenges of the hypersonic-air-breathing-propulsion system.

**CYCLE CONSIDERATIONS**

The overall cycle performance of the scramjet propulsion system is a result of individual component performances and interactions amongst the various components. Starting at the front of the engine, the forebody/inlet configuration must efficiently compress the air to a suitable pressure level to allow for both ignition and robust combustion upon fuel addition. Additionally, the inlet influences cycle performance in several ways: the degree of flow uniformity delivered across the width of the vehicle intake, the level of compression accomplished via the external and internal portion of the inlet, combustor-entrance flow-field distortions caused by shocks and expansions, and total pressure losses resulting from shock waves and shock/viscous interactions. An additional impact of inlet efficiency on cycle performance at high flight Mach numbers is the associated static temperature rise that accompanies the inlet compression process. This effect is a potentially limiting scramjet-performance factor, since excessive chemical dissociation significantly reduces the combusted-fuel-air-mixture heat release. As is illustrated in figure 5, to achieve a given static pressure level utilizing a highly efficiency inlet, comprised mainly of isentropic compression and weak shock waves, results in a much lower static temperature than that generated by a comparatively less efficient inlet employing stronger shock waves. By entering the combustor at a lower static temperature (for a given static pressure), a correspondingly lower combustion temperature results, and hence less chemical dissociation is generated. Ideally, a subsequent equilibrium-nozzle-expansion process allows the dissociated species to recombine and thereby recover the dissociation energy. However, typical hypersonic-vehicle designs require rapid area expansion in the initial nozzle contour, thereby generating a correspondingly rapid decrease in static pressure, usually resulting in a "frozen" chemical process. Thus, the dissociation energy is "lost" from the propulsion cycle and, consequently, the propulsive-system-net-thrust production is significantly degraded.

Having achieved the desired fluid-dynamic and thermodynamic state, the inlet air stream is mixed with fuel to yield heat release upon combustion. At high flight speeds, in reasonable combustor lengths, achieving effective fuel distribution (matching the local inlet airflow profile) is exceedingly difficult utilizing only wall injection. Thus, consideration of intrusive injectors (struts and the like) is often required, in spite of serious design concerns, in particular injector drag and mechanical survival. Hence, an obvious design trade exists, resulting in an optimal balancing between intrusive injector losses and a reduced combustor length scale. Typically the injector/combustor design goal for an accelerator-class engine is the total oxygen consumption of the main air stream within a reasonable length (i.e., reasonable weight) while simultaneously utilizing a near stoichiometric fuel-to-air mixture ratio. Balancing the overall engine-cooling requirement with the available fuel-cooling capacity is another engine design trade influenced by many choices in the engine cycle selection. Classical limiting processes such as constant-area or constant-pressure heat addition are often used, for simplicity, to represent the combustor analytically. The actual three-dimensional process is much more complex, and, as pointed out by Ferri (ref. 9), the overall cycle process is best considered in terms of the peak engine pressure generated, rather than the pressure generated by only the inlet process. Thus, an increasing combustor area as a function of axial distance may be appropriate to control peak pressure (and heat transfer) as the fuel mixes and burns, even at some penalty in reduced thrust performance (ref. 7). Again, a complex trade-off between engine design and operating parameters is required to achieve high levels of overall system performance.

For a typical high-speed combustor exit-flow expansion process, inclusive of both the internal nozzle and the vehicle aftbody, the optimum exhaust area is approximately 1.5 to 2.0 times the freestream engine flowpath capture area. Since the exhaust flow is always highly underexpanded, due to the losses incurred during the compression and combustion processes, an optimization of the nozzle/aftbody surfaces, inclusive of the truncation of the nozzle contours, is required to simultaneously achieve: a high net thrust value, proper vehicle trim characteristics and low system weight. Again, there are numerous complex tradeoffs required to optimize overall system performance, particularly amongst the amount and rate of nozzle expansion and the aft-end vehicle length scale. Hence, in summation, throughout the entire flight regime nozzle/aftbody performance effects are very important; therefore, a comprehensive study,
addressing these issues, is one the most complex and challenging design considerations involved in achieving high overall system performance. The following sections address in detail each component process of the scramjet engine and the associated performance issues.

Flow Nonuniformities and Cycle Performance

Interpreting multidimensional fluid-dynamic phenomena within a one-dimensional context is frequently required to assess integrated engine-vehicle performance and is particularly acute for off-axis engine design and analysis. Yet presently, no singular design methodology is widely applied by the industrial or scientific community. Historically, integral boundary-layer techniques (ref. 10) have provided much of the focus for this analytic reduction process, with the major emphasis directed towards the characterization of the mass, momentum and energy defects within the distorted region of the flow. Other specialized fluid-dynamic implementations have been explored, with particularly notable successes in the design of diffusers (ref. 11) and ramjet isolators (refs. 12 and 13). Additionally, the concept of applying integral-distortion techniques to engine cycle performance analysis is not novel (ref. 14), and is required to address the inherent limitations of purely one-dimensional cycle performance analysis techniques (ref. 15).

A simple bounded-axial-frictionless-adiabatic-constant-area flow field transitioning from supersonic to subsonic flow serves to illustrate this point. Given that at the inflow plane, the following are uniquely specified: total-enthalpy flux, mass-flow rate, cross-sectional area and stream thrust, then only two one-dimensional fluid states are compatible with the specific inflow conditions, i.e., the subsonic and supersonic solutions associated with the normal shock states. Hence, a purely one-dimensional methodology is only relevant to the two end states of this simplified ramjet engine isolator process, and therefore excludes all other pressure rises associated with the separated nonuniform fluid-flow fields typically observed during experimentation (ref. 16). Commonly, the one-dimensional engineering approach to performance analysis is to "match" within the control volume the integrated values for the mass, momentum and energy fluxes, while concurrently employing an equation of state consistent with an equilibrium thermodynamic chemistry assumption. Unfortunately this approach, as does each purely one-dimensional methodology, creates an unknown biasing, since the flow-field nonuniformities and angularities are absorbed within the definition of the associated pseudo-thermodynamic states. Individual component thrust and drag accounting can be tautologically satisfied; however, no unique cycle or process efficiency can be deduced, due to the fact that the one-dimensional methodology characterizes numerous multidimensional flow fields. Historically, this entire matter has not been aggressively pursued, in any Mach number regime, due to the expected minimal associated performance effects; however, with the maturing of hypersonic engine design technology, flow nonuniformity effects, inclusive of angularity effects, must be reassessed and hence, incorporated into cycle-performance analysis in a more complete and comprehensive manner.

To help illustrate the usefulness of a distortion based methodology, a time-dependent spatially one-dimensional integral formulation of the fluid-equation set is presented, in flux-conserving form (ref. 17) with arbitrary source terms:

\[
\begin{pmatrix}
\frac{\partial}{\partial t} \rho A \eta \eta^* \\
\rho u A \eta_A \eta_A^* \\
\rho A \eta \left( T - \frac{p}{{\rho} \eta^*} \right)
\end{pmatrix}
+ \frac{\partial}{\partial x}
\begin{pmatrix}
\rho u A \eta_A \eta_A^* \\
\rho A \eta_A \eta_A^* \left( T_F + \frac{p}{\rho} \eta^2 \right) \\
\rho u A \eta_A \eta_A^* \left( T_T - \frac{p}{\rho} \eta^2 \right)
\end{pmatrix}
= \begin{pmatrix} S_1 \\ S_2 \\ S_3 \end{pmatrix}
\]

The time-dependent forms quantify the temporal variations of the fluid mass, momentum and total-energy within the control volume, whereas the spatial forms quantify the mass, stream thrust and total-enthalpy fluxes defined by the corresponding control surface. This formulation utilizes numerous distortion parameters to quantify the spatial nonuniformities and each is analogous to the defect parameters utilized by integral boundary-layer methodologies. The justification for this added complexity
is demonstrated via the application of this equation set to the prior example of the simplified ramjet isolator problem, i.e., the bounded-axial-frictionless-adiabatic-constant-area flow field transitioning from supersonic to subsonic flow. Modeling the separated zone pressure variation versus length as a function of local dynamic pressure (ref. 12), and assuming that the mass and momentum defects are identical, yields a subset of equations that have a closed-form solution (ref. 18):

\[
\xi = \gamma \frac{C_s^2}{C_b} \left[ \frac{\frac{P}{(C_b - \bar{P})(C_b - 1)}}{1 - \frac{C_s + \bar{P}}{C_b - \bar{P}}} \right] - \frac{\gamma - 1}{2} \ln \left| \frac{C_b - \bar{P}}{(C_b - 1)\bar{P}} \right|,
\]

where, \(C_s\) and \(C_b\) characterize the flow-field total enthalpy and stream thrust, and \(\xi\) is proportional to the axial position associated with the corresponding pressure ratio, \(\bar{P}\). This resulting distortion-based solution is in good agreement with the existing ramjet isolator Mach number 2.23 data, summarized in reference 12, detailing a pressure ratio factor of 3.8 achieved in 7.80 hydraulic diameters versus a prediction of 7.77 derived from this distortion methodology. Hence, not only is the distortion methodology consistent with the limiting value, i.e., the experimentally obtained normal shock recovery value, but also it is consistent with the inherent spatial scales of the problem. This enhanced ability to assess and predict flow-field evolution is required to realistically assess high-speed engine cycle performance, and it is recommended that a more complete cycle methodology, including distortion parameters of the form defined in the prior text, be adopted by the community at large.

INLET

Sidewall Compression Concepts

Inlet research at Langley Research Center over the last three decades has emphasized three-dimensional designs that have the potential to optimize both aerodynamic and structural performance. The airframe-integrated propulsion system may be made up of a group of rectangular propulsion modules attached to the bottom surface of the vehicle, as illustrated in figure 6. The forebody serves as a portion of the external inlet, and the aftbody completes the nozzle expansion process. The inlet illustrated in figure 6 is one of a class of three-dimensional inlets that utilize lateral compression from the inlet sidewalls to complete the longitudinal compression initiated by the vehicle forebody. The bottom surface (cowl) leading edge is typically located aft near the inlet throat to allow air to spill at low speeds, thus making possible fixed-geometry designs that can operate over a wide speed range (see ref. 19).

Numerous performance characteristics of this inlet class are varied by altering the sweep angle of the sidewall-compression leading edges (fig. 7). This latter feature is particularly desirable from a structural and heating perspective. When flow is compressed by a swept-sidewall leading edge, a flow component is generated in the direction of the sidewall sweep, turning the flow down towards the cowl in the case of the aft-swept leading edge. Flow is allowed to spill ahead of the cowl leading edge, making the inlet easier to start, that is, to pass the normal shock and establish supersonic flow. The downward component of flow is reduced at higher speeds to reduce spillage, resulting in a significant increase in airflow captured at Mach 6 (ref. 19). However, the downward turning of the flow by the swept leading edge and spillage causes a shock wave from the cowl leading edge, and this increases distortion ahead of the combustor. Reversing the sweep of the sidewall leading edge causes flow to be turned upward ahead of the cowl and reduces spillage and flow distortion. Inherently this design makes the inlet harder to start and consequently variable-geometry inlet features may be required. Starting characteristics of this class of inlet, having fixed geometries, have been studied (ref. 20), and some of those results are shown in figure 8 for inlets with aft sweep. Note that starting characteristics are a function of aspect ratio, and that inlets having a low aspect ratio can be started with a fairly high contraction ratio. Some experience has also been gained with inlets having a forward sweep (ref. 21), and these inlets have proven to be much harder to start without variable geometry. Recent work has examined other sidewall shapes that would retain the desired starting characteristics of aft swept inlets while improving mass capture, pressure recovery, and throat flow distortion (ref. 22).
The interaction of shock waves with boundary layers formed within the inlet and ingested from the vehicle forebody are a first-order concern in the design of the hypersonic inlet. A particularly troublesome shock wave/viscous interaction that exists during the sidewall compression inlet process is similar to that of the interaction of a fin generated shock with a thick boundary layer, which has been studied by several researchers (refs. 23 and 24). In this interaction (fig. 9), the incoming boundary layer separates, for all but the weakest interactions, and rolls up into a strong vortex near the base of the compression surface. A number of studies have been conducted to examine the crossing-shock-wave interaction formed by two fins, again similar to the compression system generated by the surfaces of a sidewall compression inlet (refs. 25, 26, and 27). These investigations find a strong vortex-vortex interaction occurring along the flow-field centerline next to the body surface to create a large core of low total pressure air. Computational studies of this flow field (ref. 28) illustrate this low-energy core (fig. 10) which penetrates deep into the inviscid flow. Also illustrated in this figure is that, given the viscous nature of the flow, a full 3-D Navier Stokes solution is required to capture this flow field. Recent research has examined the consequences of this complex flow on inlet performance, as illustrated by computational efforts (ref. 22) to examine the influence of sidewall shape on the vortex-vortex interaction and resulting inlet performance. The problems associated with this interaction would be expected to be reduced at higher hypersonic speeds, as is typical of other shock-viscous interactions. However, there has been little work to study this type of interaction at hypersonic speeds (an exception is the work by Frank Lu in references 29 and 30), so that its influence on performance at higher speeds is largely unknown.

A significant consequence of the low-energy air core at low speeds is the resultant effect on performance during low-speed ramjet operation. When operating as a ramjet, a strong shock system progresses forward toward the inlet throat, a consequence of the downstream-subsonic-combustion process. The low-energy core of air acts as a path for communication between the downstream high pressure and the upstream supersonic flow within the inlet and may cause the inlet to unstall prematurely. Thus, the maximum value of pressure that can be sustained downstream of the inlet throat, without unstalling the inlet, is a measure of the ramjet-mode-performance potential. This problem of a supersonic viscous flow coupled with a downstream subsonic high-pressure region is very difficult to study computationally, but has been studied experimentally in reference 31. In this study, backpressure was applied with a throttling device to simulate a subsonic combustion process, and backpressure was increased during the test until the inlet was forced to unstall. The objective was to alter the body side geometry that generates the thick boundary layer in order to energize, or reduce the effect of the vortex interaction region, thereby allowing a higher backpressure to be obtained before the inlet unstalled. Some results of this study, given in figure 11, indicate a benefit of adding a compression ramp to the body side surface, with the level of maximum backpressure increasing corresponding with increasing ramp angle. These results are encouraging, yet more work is required in order to understand the complex flow mechanisms and to determine how to optimize inlet-geometry performance.

Interactive Inlet Design

In general, computationally based interactive and/or automated design algorithms relevant to ramjet and/or scramjet inlet configurations, addressing realistic three-dimensional viscous gas dynamics, are prohibitively costly to implement due to the large computational resource requirements. By necessity this limitation, long recognized by the design community, led to numerous uniquely ingenious inlet design solutions and insights, a prominent early example being the nearly equivalent-strength multiple-shock solution of Oswatitsch (ref. 32). Yet, given the historical trend of increasingly more powerful computational hardware, numerous researchers in associated and parallel disciplines continue to address the inherent technical limiting factors of interactive analytic design, with notable progress being achieved in nonlinear optimization algorithms (refs. 33 and 34) and wall function methodologies (refs. 35 and 36). The former addresses the generation of an optimal solution, without invoking the typical iterative methodologies of constrained optimization (ref. 37), whereas the latter addresses circumventing the inherent stability and accuracy restrictions of numerically generated solutions to the equations governing viscous fluid phenomena. Albeit that these techniques are still theoretically incomplete, each contributes to the ultimate goal of producing a useful interactive inlet design tool. Also to this end, Korte et al. (refs. 38 and 22) utilized a Parabolized Navier-Stokes fluid-dynamic spatial-marching algorithm to examine design issues relevant to laminar two-dimensional flight-type scramjet inlet configurations, as well as turbulent three-dimensional scramjet inlet geometry, and concluded that optimal inlet designs must invoke contoured nonplanar wall shapes.
In summation, clearly the relevant constraints and the exact measures of performance criteria to be optimized are design specific, and hence complicate the generalization of any encompassing procedure. Yet, the need to ultimately enhance the relevant technology remains and, therefore, future research efforts are envisioned.

**Inlet/Isolator Interactions**

The inlet is coupled aerodynamically to the combustor through both the viscous flow that originates within the inlet, and shock and expansion waves that continue into the combustor. At lower speeds, when the engine is operating in a subsonic combustion ramjet mode, performance is most sensitive to the aerodynamic coupling between these two components. Before moving to the combustor, we will examine performance characteristics of a constant-area "isolator" section intended to aerodynamically isolate the supersonic inlet from the downstream subsonic combustion. Ideally, the downstream pressure rise would approach that which would exist behind a normal shock wave at the inlet throat. Isolator research is a subject of a chapter in this text. The work described is based on tests conducted in a constant-area duct with a uniform inflow profile. The flow field within the constant-area duct represents conditions at the inlet throat, and the length of duct that is affected by the pressure rise is a measure of the length of isolator required to support a given pressure rise. While this is an accurate method to develop isolator performance maps, the method assumes that the flow from the inlet is uniform, or that inlet flow uniformity does not substantially affect isolator performance.

Tests have been conducted at Langley Research Center in a Mach 4 blowdown tunnel to address inlet/isolator performance when representative inlet flow fields are included. The model used for these tests is described in figure 12, and consists of an inlet, isolator, diverging section, and downstream choke mechanism. The inlet has a removable foreplate to vary the thickness of the incoming boundary layer, and a rotating cowl to start the inlet and to change the inlet contraction ratio. Two cowl lengths were provided to change the flow field within the inlet and the resulting flow profile entering the isolator. Note that the long cowl produces a shock system that nearly cancels on the bodieside shoulder to minimize distortion, while the short cowl produces a shock system that reflects from the cowl side opposite the shoulder to produce a much higher level of distortion. The test procedure was to start the tunnel and inlet with the cowl deflected down. Once the inlet is started, the inlet cowl is rotated to a predetermined position, and the flowmeter choking device is slowly closed until the inlet unstarts. Data channels are scanned continuously during this process, which enables static pressure to be plotted through the model at successive times, as illustrated in figure 13. Note the large pressure fluctuations within the isolator before backpressure is applied, presumably due to unstable shock/boundary-layer interaction and separation, and the increase in downstream pressure up to the point of inlet unstart.

Tests were conducted over a range of inlet cowl rotation angles for the two inlet cowl angles illustrated in figure 12 with and without the forward boundary-layer generating plate. Maximum backpressure achieved before inlet unstart is summarized in figure 14 for an isolator length that is 4.7 times the inlet throat height. Maximum backpressure is plotted against inlet convergence angle, which is the difference between the body ramp angle and the cowl angle. Backpressure generally increases with convergence angle, and is much higher for the longer cowl. Several interesting points can be observed from this figure. First, a higher inlet convergence angle is achievable with a thinner incoming boundary layer, and the maximum convergence achieved is independent of cowl length. This latter observation suggests that inlet unstart caused by increasing convergence angle is not due to overcompressing the flow, but is caused by the strength of the shock/boundary-layer interaction ahead of the throat. It is not surprising that the long cowl produces higher pressure, since backpressure is a product of the pressure rise within the isolator and the inlet pressure ratio. Because of its larger area contraction ratio, the pressure ratio generated by the longer cowl is greater than that generated by the short cowl. Also note that all of the data converge at low inlet convergence angles where there is only a small contraction ratio from the cowl lip to the inlet throat.

Data from figure 14 are given in a different form in figure 15, where only the isolator pressure rise is plotted against inlet contraction ratio, thereby separating isolator performance from inlet performance. The Mach 4 flow is compressed by the 11° forebody to Mach 3.2, and is then further compressed by the inlet internal contraction from the cowl lip to the throat. Comparisons are made with data from the constant-area isolator study reported in another chapter in this text by relating the local
Mach number within the isolator to the contraction ratio required to compress the flow from Mach 3.2 to that Mach number that exists within the isolator but ahead of the pressure rise. Thus for all cases, for a given contraction ratio, the Mach number is about the same and is increased to approach Mach 3.2 as inlet contraction ratio approaches unity. Note that as throat Mach number increases with reduced contraction ratio, the isolator pressure rise from the cases considered increases and tend to converge. Thus, when the inlet is operating at a low contraction ratio, very little pressure ratio is being generated by the inlet, the throat Mach number is fairly high, and isolator performance approaches that measured in a constant-area duct. At higher inlet contraction ratios where inlet throat Mach number is significantly reduced, the effect of inlet configuration and resulting distortion into the isolator becomes obvious, particularly when a thick bodyside boundary layer that is more consistent with flight geometries considered. At this incoming Mach number with maximum inlet contraction ratio, the isolator behind the short cowl, which produces a shock pattern that causes a very high distortion level, yields only about half the pressure rise found for an isolator operating on the flow from a constant-area duct without distortion.

To summarize, boundary-layer thickness, inlet contraction ratio, throat Mach number, and the nature of the compressive shock pattern within the inlet all contribute to a level of distortion ahead of an isolator that has been shown to have a direct impact on isolator performance. This discussion of inlet/combustor isolators has emphasized the low Mach number portion of flight where subsonic combustion is expected, a large pressure rise is associated with the subsonic diffusion and combustion process, and the isolator section is absolutely necessary to contain that high pressure. At higher speeds, where supersonic combustion is initiated, some isolation from the inlet may still be required as a result of a close proximity of fuel injectors and resulting pockets of subsonic flow to the inlet throat. At high hypersonic speeds, a high supersonic Mach number will exist at the inlet throat so that an isolator section will not be required. Thus, the challenge in the design of a dual mode scramjet capable of operation over a wide speed range is to provide a constant-area isolator section at low speeds without penalizing scramjet performance at high hypersonic speeds.

**COMBUSTION AND COMBUSTORS**

**Introduction**

The next section highlights efforts to extend and improve scramjet combustor technology. The demonstrated operating range of scramjet combustors has been increased from a status approaching flight Mach number less than 10 in the 1975-85 time frame to Mach number 15 to 20 today. Scramjet combustor technology quality, in terms of the level of detail addressed and the accuracy of predictions, has also been dramatically improved during this time frame with the development and application of laser diagnostic measurement techniques and three-dimensional reacting flow computational fluid dynamics (CFD). This improvement was driven by the small thrust margin at hypervelocity (see figure 2). Methods have been implemented to quantify the effect of losses on overall vehicle performance. Today, optimized combustor designs are based, as a minimum, on engine thrust potential. Integrated design tools are now available which are capable of tracking all combustor losses and their impact on vehicle performance over the vehicle operating range. Combustor designers can account for losses due to fuel mixing, nonequilibrium chemical kinetics, injector drag, combustor flow distortion, combustor wall shear and heat flux, overall thermal balance, engine weight, etc. The large gains in scramjet combustor technology obtained over the past decade has resulted from careful application of experimental, analytical, and numerical methods.

The status of supersonic combustor technology in the 1985 time frame is summarized by Northam and Anderson (ref. 39). At that time, NASA combustor technology was focused on a Mach 4-10 research aircraft. Scramjet combustor design tools in this era were based largely on experimental studies, and designs were evaluated both in direct-connect combustor and in free-jet engine experiments (see ref. 39). Performance levels for both all supersonic and dual mode (mixed supersonic and subsonic combustion) were established. Some of the critical problems encountered in this "low" supersonic combustion speed regime were addressed, including fuel mixing, ignition, flameholding, mode transition, and combustor-inlet interactions. The scramjet concepts investigated were shown capable of thermally balanced operation with sufficient thrust and efficiency to accelerate a research aircraft.
Hypersonic Combustion Physics

Single-stage-to-orbit air-breathing concepts, such as NASP, provide new challenges to combustor designers. As flight speed increases, the combustor environment becomes truly hypersonic (ref. 40). At such speeds, the kinetic energy of the free-stream air entering the scramjet propulsion cycle is large compared to the energy released by reaction of the oxygen content of air with hydrogen fuel. Thus, the effects of reaction at Mach 25 speeds, where heat release from combustion may be 10 percent of the total enthalpy of the working fluid, will be small compared to Mach 8 flight where the air kinetic energy and potential combustion heat release are roughly equal. Hypersonic combustion, perhaps, corresponds closely to the gradual diffusive mixing and burning process described by Ferri (ref. 41). Flow deflections due to heat release are small—a few degrees at most—and flow boundaries are conceived as contoured to control the pressure rise at the location of the flame and eliminate the possibility of strong shock formation.

To the contrary, at speeds of Mach 8 and below, combustion in ducted flows can generate large local pressure rise, flow deflection and separation. The behavior of this "upstream interaction" has been studied extensively by Billig (ref. 42) and is characteristic of supersonic combustion in constant-area channels below flight speeds of about Mach 8. In such flows involving local separation and a bulk Mach number near one, local wall static pressure is representative of the pressure across the entire flow at a given axial station. Therefore, a one-dimensional approximation to the flow can provide a reasonable description of the flow behavior. In hypersonic combustion, however, local Mach number remains high, Mach angles are quite shallow, and significant variations in static pressure are likely to occur across the combustor flow field at a given axial station. Typically, a fully three-dimensional representation of the flow field will be required; and as pointed out by Stalker (ref. 40), special care must be taken to properly relate the implications of one-dimensional calculations to fully three-dimensional experimental data in a meaningful way.

So hypersonic combustion flows differ from supersonic combustion flows in that the flow remains hypersonic throughout in a bulk sense, the effects of heat release are smaller, and the pressure field is fully three dimensional. Common features of hypersonic and supersonic combustion flows include: real gas effects, nonadiabatic wall boundaries, finite-strength shock waves, dissimilar gas injection, turbulent mixing, finite-rate chemical reaction, flow separation, etc. Also, because aerodynamic, fluid mixing, and chemical rate processes are all expected to be important, experimental simulation requires near full-size hardware and duplication of flight conditions. Simulation requirements are discussed in more detail in the next section of this paper.

Hypersonic combustion raises some additional uncertainty and concerns. First, the effects of (extreme) compressibility on turbulence generation and mixing are not well known or understood. Second, at about Mach 12 the velocity of the injected (hydrogen) fuel stream equals the velocity of the combustor air stream, and at higher flight speeds the air velocity exceeds the fuel velocity. The behavior of fuel-air mixing under these conditions is also not well known. In fact, compared to flight at Mach 8 and below, there are very little data on which to base confidence in our understanding of or ability to model hypersonic combustion flows.

Simulation Requirements

The primitive variables available which describe the flow in a hypersonic combustor are:

P pressure (or rho density)
T temperature
u velocity
L model length
nuj gas composition
In principle, the composition can be manipulated in any fashion that results in duplication of the flight values of certain well-known dimensionless groups, i.e., simulation parameters. The first-order simulation parameters are:

M  Mach number
Re  Reynolds number
St  Stanton number
D₁  Damkohler's first number
D₂  Damkohler's second number
GW  wall enthalpy ratio

To these may be added certain second-order parameters such as:

Pr  Prandtl number
Sc  Schmidt number

The physical interpretation of the first three fluid dynamics and heat-transfer parameters is also well known, i.e., the Mach number represents the ratio of kinetic to thermal energy, the Reynolds number the ratio of inertial to viscous forces, and the Stanton number the ratio of heat flux to inviscid energy flux. However, the interpretation of the last three is less well known as they are more specific to hypersonic reacting flows. These three parameters are the ratios of flow transit time through the combustor to chemical reaction time (D₁), the ratio of heat added by reaction to the stagnation enthalpy of the inviscid flow (D₂) and the ratio of enthalpy at the wall temperature to stagnation enthalpy of the inviscid flow (GW). The second-order parameters represent gas properties, e.g., the ratio of viscosity to thermal conductivity and the ratio of viscosity to diffusivity. However, to the extent that these parameters reflect turbulence characteristics, they are more dependent on the first-order simulation parameters than on molecular properties of the gas. The first-order parameters can, in turn, be related to the primitive variable as follows:

\[ M \sim \frac{u}{\sqrt{T}} \]  \hspace{1cm} (1)

\[ Re \sim \frac{\rho u L}{\sqrt{T}} \sim \rho LM \]  \hspace{1cm} (2)

\[ St \sim \frac{q_w}{\rho u H} \]  \hspace{1cm} (3)

\[ D_1 \sim \frac{L}{u t_c} \]  \hspace{1cm} (4)
\[ D_2 \sim \frac{\eta_c \Delta h_c}{c_p T + \frac{u^2}{2}} \]

\[ GW \sim \frac{c_p T_w}{c_p T + \frac{u^2}{2}} \]

where \( t_c \) is a characteristic combustion time, \( h_c \) is a combustion efficiency, \( \Delta h_c \) is the heat of combustion, \( c_p \) is a characteristic specific heat and \( T_w \) is the temperature of the combustor wall. The overall reaction time for a typical combustion a process is generally proportional to both a function of pressure (or density) with an exponential dependency of approximately 1.75 and a function of temperature with an exponential dependency of unity. In certain restricted conditions wherein only binary (two-body) reactions occur, the combustion time is linear in density, leading to a direct relationship between Reynolds number and Damkohler's first number:

\[ D_1 \sim \rho L / u \exp (-T) \]

\[ \sim Re \frac{\sqrt{T}}{u^2} \exp (-T) \]

Hence, if velocity and temperature were to be duplicated, then simulation of Reynolds number would also satisfy the requirements for simulation of binary reaction time, and vice versa. However, even in the simplest of situations, it is virtually impossible to manipulate the temperature, velocity and model length in a fashion that will simultaneously preserve the values of Mach number, Reynolds number, and Damkohler's numbers.

Therefore, it is clear that, in general, in hypersonic combustion experiments it is necessary to duplicate the primitive variables, including model length and gas composition, to ensure a faithful representation of the coupled chemical and flow processes. This automatically satisfies all of the simulation parameter requirements, with the possible exception of wall temperature and wall reactivity simulation.

**Experimental Simulation**

As pointed out above, combustion heat release in air produces about the same energy increment as the kinetic energy of flight at Mach 8. Thus, the experimental simulation of supersonic combustion flow conditions for propulsion studies in ground facilities frequently utilizes so-called direct-combustion heating with oxygen replenishment or vitiation heating as a means of generating a test environment. Essentially, the test is conducted in (fuel-lean) combustion products where the amount of free oxygen is adjusted to represent the free-oxygen content of air. Of course, the remaining nonoxygen content of the test gas is not principally nitrogen as in air but will consist of some carbon dioxide and/or water vapor, etc., depending on the choice of fuel and oxidizer. The National Aero-Space Plane (NASP) "Engine Test Facility" (ETF) at Aerojet is an example of a Mach 8 vitiated propulsion facility based on a storable propellant (nitrogen tetroxide/monomethylhydrazine) rocket gas generator (ref. 44). The NASA Langley 8-Foot High-Temperature Tunnel (8' HTT) is an example of a Mach 7 air/methane/oxygen direct-combustion heated propulsion facility (ref. 45).

Contamination of the test media with combustion products is, of course, a concern because the contaminants are not inert and do not have the same thermodynamic properties as the nitrogen they replaced. Suffice it to say that interpretation of the test data from vitiated facilities (and from any facility, in fact) requires detailed consideration of the actual test media as it affects the results in comparison with
free flight in air. As the speed of the ground-test simulation increases, in addition to the turbulence (velocity fluctuation) intensity of the test flow, other (turbulence) factors like temperature and oxygen-concentration fluctuation levels, chemical contamination, the degree of dissociation of contaminants and of molecular oxygen, etc., all may become issues which must be addressed.

Other sources of energy such as storage heaters or electric-arc heaters can also provide high-temperature test conditions. Contamination (with the addition of contamination from either dust particles or NO\textsubscript{X} formation, respectively) is still an issue which must be addressed. To their credit, arc heaters have the potential of reaching energy levels corresponding to much higher flight speeds than combustion-heated facilities and, in fact, have been used successfully to simulate the orbital reentry flow environment. However, for propulsion flow simulation, conventional arc heaters lack the ability to operate at stagnation pressures adequate for hypersonic combustion simulation. Typically, arc heaters can achieve operating pressures of 20 to 50 atmospheres where pressures of 2000 to 5000 atmospheres and higher are desired.

The enthalpy requirements for hypersonic combustion simulation are summarized in figure 16. The sensible total enthalpy of flight is shown plotted against Mach number. The right-hand-most curve shows the free-stream Mach number along the flight path. The forebody flow field and inlet compression processes reduce the local Mach number and raise the flow static pressure along a nearly constant total enthalpy path, as indicated by the arrow. The shaded band represents the range of Mach number and total enthalpy representative of combustor entrance conditions. The local static temperature, stagnation temperature, and stagnation pressure representative of the combustor entrance condition are shown at the points indicated for flight Mach numbers of 10, 15, 20, and 25. The static pressure is typically 0.5 to 1 atm. Note that the static temperature remains in the range of 3000\degree to 4000\degree R while the stagnation temperature quickly reaches levels where significant dissociation will occur. Also note that the stagnation pressure rises exponentially with Mach number from achievable levels (100 atm) at Mach 10 to extreme levels exceeding a million psi at Mach 25. The scale at the far right shows a level of oxygen dissociation and expected in a facility which adds energy to the test gas at rest (such as an arc heater or a reflected-shock tunnel) as a function of stagnation enthalpy. Pulse facilities, which generate high pressure, hypervelocity flows for a short time, represent the only presently proven means to approach simulation of these hypersonic combustion flow conditions on the ground.

Figure 17 shows the total enthalpy simulation capability of selected pulse facilities on the same coordinates used in figure 16 (data from ref. 46). Aerodynamic simulation of Mach number and Reynolds number is achieved in the Calspan reflected-shock tunnels with ambient temperature or moderately heated (600\degree F) helium as a driver gas, but duplication is limited to a flight Mach number of about 10 for a test duration of 1 msec. The reflected-shock tunnels T4 and T5 take advantage of the higher temperature achieved by free-piston compression of helium driver gas to achieve energy approaching orbital velocity. The Ames 16-inch shock tunnel is intermediate in energy simulation capability with a hydrogen combustion-heated driver. The expansion tube (HYPULSE), which is essentially two shock tubes in tandem, can achieve Mach 16+ energy with an ambient temperature helium driver.

Only one type of pulse facility, namely a free-piston-driven expansion tube, has the potential capability to duplicate both total enthalpy and total pressure, as well as velocity and gas composition, above about Mach 16 (ref. 47). This capability has thus far only been demonstrated in a pilot-scale (1-1/2-inch inside diameter) facility (ref. 48), however, addition of a free-piston driver to the 6-inch inside diameter NASA expansion tube (HYPULSE) located in Ronkonkoma, Long Island, New York, and operated by GASL is planned, and should increase the operating envelope as indicated in figure 17.

The total pressure required for hypersonic combustion simulation is shown in figure 18. The shaded bands show pressure requirements for either free-stream engine simulation or internal combustor flow simulation along with the capabilities of the selected facilities. Blow down facilities with run times on the order of 100's of seconds like the Langley 8' HTT and the NASP ETF's are limited to 2 to 3 kpsi. Higher enthalpy facilities like the Ames 100 MW Arc-Heated Facility have even less pressure capability because of the facility nozzle throat cooling problem. Pulse facilities like the Calspan, Ames 16-Inch and Cal Tech T5 reflected-shock tunnels (ref. 46) can produce and contain significantly higher pressures (up to 30 kpsi) largely because of shorter flow times on the order of milliseconds. Note that at Mach numbers above 12, the total pressure requirement approaches a million psi, and only the expansion tube is capable of producing those pressures. The unique capability of the expansion tube is a result of the fact that the
acceleration process in the expansion tube adds velocity directly to the flow without stagnating it. This means that the facility need not contain the stagnation pressure of the flow that it generates, and this feature is a considerable advantage when duplication of flight conditions is required.

The fundamental difficulty in generating hypersonic flows in a ground facility is related to putting the energy into the proper mode in the test gas that is generated. Figure 19 is an attempt to explain this in more detail (ref. 49). The energy required for flight along a constant dynamic pressure trajectory of 1000 psf is taken as a reference with the generation of a combustor pressure of 0.5 atm and the assumption of an inlet process with a kinetic energy efficiency of 98 percent. The facility process assumes stagnation heating with isentropic expansion from the stagnation conditions to the required combustor entrance condition and chemical freezing of the gas composition at the throat of the facility nozzle. The horizontal axis represents the error in energy in static temperature compared to flight of this particular facility stagnation pressure and total enthalpy, where the energy scale has been made dimensionless with respect to the amount of energy in stoichiometric combustion of hydrogen and air. Similarly, the vertical axis is the error in energy due to composition. That is, if the gas is dissociated at the stagnation energy level, this dissociation persists in expansion through the nozzle, and energy is tied up in NO and atomic oxygen which would not be present in clean air in flight. Various flight Mach numbers are shown by the solid lines, and various total pressure levels are indicated by the dashed lines. Since flight in clean air is taken as the reference, in figure 19 “goodness” is towards the center and bottom of the figure with a small energy error due to a static temperature mismatch and a small energy error in dissociation. Note that as Mach number increases, the amount of energy error in composition increases substantially. Also, as Mach number increases, generally the required stagnation pressure for simulation of flight significantly increases. Figure 20 adds a similar carpet plot of energy error for a facility like the expansion tube which generates velocity directly without heating at stagnation conditions. As can be seen in figure 20, the level of energy in composition error is approximately an order of magnitude smaller for the expansion tube or nonstagnating facility compared to a stagnation heating facility.

Another way to assess facility contamination level is in terms of the amount of molecular oxygen in the test flow compared to the oxygen content of clean air in flight. The performance of a number of facilities is shown in figure 21 as a function of flight Mach number (ref. 50). The hatched band shows the performance typical of reflected-shock tunnels with particular facilities such as T5 and the Ames 16-inch tunnel shown by symbols on the figure. The expansion tube HYPULSE is shown by the circular symbols. As Mach number increases beyond Mach 12 the expansion tube facility shows an increasing advantage over the reflected-shock tunnels in terms of available oxygen to simulate flight in clean air.

The comparatively moderate investment required to generate the desired test conditions in pulse facilities is achieved at the expense of flow duration. Producing adequate flow duration to establish a sensibly steady flow representative of the "real" steady flow becomes an issue which (like the contamination issue) must be addressed in facility design and in the interpretation of data. However, short flow duration has an advantage in high-energy flow tests in that the model cooling requirement can be met by simple heat sink approaches. The short flow duration also makes some types of measurements, like local heat flux, much easier to make accurately in pulse flows than in steady flows. On the other hand, all measurements must have very high frequency response—on the order of $10^6$ Hz—in order to provide meaningful data from test times of $10^{-3}$ sec or less. In this regard, it may be noted that hypersonic tests conducted in a pulse facility almost inevitably involve an unheated (room temperature) model. Hence the wall enthalpy ratio, GW, is typically on the order of 1/10 or less, whereas in flight it might be as high as 2/10. Fortunately in point of fact, the heat-transfer and skin-friction coefficients both become virtually independent of GW for values less than about 3/10.

Of greater or equal concern than duplication of wall temperature is the simulation of wall reactivity. This consists of both the tendency of the wall to catalyze gas phase reactions and the extent to which the gas phase reactions are thermally quenched (or promoted) at the wall. The catalytic efficiency is primarily dependent on certain gross characteristics of the wall materials, i.e., metals tend to be "fully catalytic" and glassy materials (e.g., quartz) tend to be "noncatalytic." Thus, use of a model composed of metal walls with glass windows may present a potential local disruption of wall catalycity effects. However, the extent of the problem is easily determined by using metal blanks in place of windows. The effects of thermal quenching at a model wall temperature that is substantially less than exists in flight is more difficult to determine experimentally, as wall-mounted instrumentation may not function at the true
wall temperature. However, this would appear to be an instance where computational fluid dynamics (CFD) simulations could be trusted to correct the data for wall temperature effects. If necessary, the model could be preheated (e.g., radiatively) and nonintrusive or noncontacting instrumentation (e.g., exit plane surveys) used to assess wall temperature effects.

**Comparison of Combustion Data**

This section describes an experimental assessment of the effects of the level of contamination by dissociation of test gas on a mixing and combustion experiment at Mach 17 flight energy conditions. The intention is only to show the effects of the state of the test gas produced by different facilities on the experimental results and not to imply that one facility type should be pursued to the exclusion of another. In fact, as noted earlier, increased contamination goes along with higher energy (fig. 16). In general, all hypervelocity data need to be assessed in light of the test media involved including the initial composition and finite-rate chemistry which may be important in the flow field. Figure 22 shows the experimental apparatus. Basically the configuration is a circular tube 1-1/2 inches in inside diameter with a sharp leading edge. The tube is about 36 inches long, and an annular injector is located about 7 inches from the entrance of the tube to inject hydrogen into the high-velocity air captured at the entrance of the tube. Hydrogen is supplied to the injector by a fast-acting valve from a Ludwig tube, and the hydrogen flow is initiated such that hydrogen is flowing into the model when the facility flow is established and initiates the combustion simulation.

Two identical models were constructed. One was tested in the NASA HYPULSE facility at Mach 17 flight conditions. The other was tested in the reflected-shock tunnel T4 at the University of Queensland in Brisbane, Australia. Typical flight conditions and the wind tunnel test conditions are shown in Table 1.

<table>
<thead>
<tr>
<th>HYPersonic Combustion Conditions</th>
<th>Mach 17 flight simulation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>HYPULSE</strong></td>
<td>3.29</td>
</tr>
<tr>
<td><strong>T4</strong></td>
<td>2.11</td>
</tr>
<tr>
<td><strong>Typical flight</strong></td>
<td>7-20 psia</td>
</tr>
</tbody>
</table>

- Deficiencies of simulation
  - Nonequilibrium chemical contamination (O and NO)
  - Low total pressure and Reynolds number
- Other potential differences between facilities
  - Turbulence intensity
  - Contamination from diaphragm material, etc.

Note that in flight only molecular oxygen would be expected to be present in the test stream, but at Mach 17 energy level the reflected-shock tunnel T4 has significant dissociation compared to the clean-flight condition. Wall pressure measured in the tube downstream of the injection point in these tests is shown in figure 23. The combustion pressure rise parameter shown is the pressure with injection into air minus the pressure for injection into nitrogen divided by the pressure with injection into nitrogen. Presenting the
data in this way shows the maximum effect of combustion on wall pressure and tends to remove some of the wave structure which otherwise produces scatter in the data. The open symbols show data from the reflected-shock tunnel T4 and the solid symbols show data from the expansion tube HYPULSE. Clearly, in figure 23, considerably more pressure rise is generated in the reflected-shock tunnel T4 which has a level of dissociation approximately equal to half of the molecular oxygen compared to free-stream undissociated air.

Reference 51 presents an analysis to understand the reasons for this difference in measured pressure. The flow in the tube is modeled with a simple one-dimensional flow conservation equation that includes three temperatures for the mixing/reacting fuel and air flowing down the tube. Initial temperature for the fuel and air are defined by the initial conditions in the experiment, and a mixed stream of fuel and air with an independent temperature generated by a finite-rate kinetic computation follows development of the flow. Wall friction, shock losses, and the rate of mixing in the tube are modeled based on the expansion tube wall pressure data and are held constant with the different initial composition of the reflected-shock tunnel flow (Table 1) to derive the predicted pressure distribution which is compared with the data in figure 24. Excellent agreement between the prediction for the reflected-shock tunnel T4 is achieved with data using this three-temperature scheme with the finite-rate kinetic model. Apparently, even though a significant amount of the molecular oxygen is withheld from reaction with hydrogen fuel as NO, the initial level of atomic oxygen present in the dissociated test gas increases heat release and the amount of reaction resulting in the greater pressure rise.

Figure 25 shows results from the same computation in terms of the energy yield predicted in the experiment. Results for the expansion tube are shown by the solid lines and results for the reflected-shock tunnel are shown by the dashed lines. The lowest pair of curves shown is for the actual pressure and temperature of the experiment. Due to the initial dissociation in the reflected-shock tunnel, the energy yield is somewhat higher than for the expansion tube. If the pressure were raised to a level of 1 atmosphere, which is more representative of a typical flight trajectory, an even greater difference would be observed; and if the temperature was also reduced to a level of 1200 K (again, more representative of flight), a still greater energy yield would be achieved with a larger difference between the reflected-shock tunnel and expansion tube resulting. The top curves show the computed result if the flow were in local equilibrium as it mixed along the length of the tube. The energy level is considerably above the actual measured result for the pressure and temperature of the experiment, indicating that finite-rate chemistry plays a dominant role on the results for the conditions of the test. Again, the intent of this comparison is to emphasize the need to account for the composition of the test media in hypervelocity combustion simulations in order to understand the implications of ground simulations on flight performance.

**Instrumentation/Measurement Requirements:**

Measurement of scramjet combustor performance is difficult to achieve in pulse tunnels since the influence of combustion is small on the easily measured quantities, such as wall pressure. An in-depth study of measurement requirements was performed by Bittner (ref. 52). This study showed that for reasonable hypervelocity combustor designs, fuel mixing and combustion efficiency are the most important combustor performance parameters (i.e., engine thrust is about proportional to combustion efficiency). Considering the measurement uncertainty and sensitivity of nondirect measurements (not including uncertainty of assumptions required to deduce performance from these indirect measurements) for deducing mixing and combustion efficiency, Bittner concludes that these performance parameters must be directly measured. Figure 26 illustrates the performance uncertainty expected based on several measurement approaches which could be used to determine combustion and mixing efficiency. The measurement uncertainty column indicates the assumed accuracy of the measurement technique, performance sensitivity to measurement is the relative change in the measurement to the performance parameter, and the performance uncertainty is the product of the measurement uncertainty and performance sensitivity. Even without considering uncertainty introduced in deducing performance from indirect measurements, direct measurements (such as water for combustion efficiency) is clearly the most accurate approach. This study also shows the limitation of many planar measurements, which are only order-of-magnitude accurate (+/- 80 percent).

Clearly the best experimental measurement for determining combustion efficiency is combustor exit water mass fraction (determined by line-of-sight laser absorption), and for mixing efficiency, is fuel
mass fraction distribution. Bittner demonstrated, by evaluation of typical CFD solutions, that 3 or 4-0.50 mm diameter laser absorption paths for a 1-inch-high combustor are adequate to resolve the combustion efficiency to +/- 5 percent uncertainty, at the combustor exit, as illustrated in figure 27. This measurement technique is being applied to combustion tests both in the NASA Ames 16-Inch RST and the NASA LaRC HYPULSE expansion tube at GASL.

Measurements of fuel mixing efficiency and fuel distribution are being accomplished using an approach which will be described in the following paragraphs, by tracing the injected fluid with particles. In this technique, the hydrogen fuel to be injected has a small fraction, say 3 percent of silane (SiH₄) added. Silane is very reactive and burns on contact with air or oxygen even at room temperature and in very dilute mixtures. Immediately prior to injection into the airflow, oxygen is added to this silane/hydrogen mixture sufficient to burn the silane present and produce silicon dioxide particles. These particles, typically on the order of 0.2 microns in diameter, are small enough to follow the fuel as it is injected, mixes and burns in the flow. Illumination of a cross plane in the flow with laser light can then produce a Mie scattering image of the particles which can be recorded with a fast electronic camera and thus visualize the fuel distribution in the experiment. This process is shown schematically in figure 28 for a typical combustor duct with ramp-type fuel injectors.

The process of correcting the raw image and processing it to produce a true cross plane in the flow is presented in figure 29. Both an image with injection in the flow and a reference background luminosity image are recorded. These images are subtracted and then stretched to represent a true cross plane in the flow. This image is further corrected to reflect the actual intensity profile of the laser sheet illumination to provide an accurate picture of particle concentration in a cross plane in the flow. If particles are neither created nor destroyed in the mixing/reacting flow as it proceeds downstream, then the total amount of scattered light in the corrected image from each plane should be approximately the same. Thus, the local amount of reflected light in each image, nondimensionalized by the average reflected light in each plane, becomes a measure of the local fuel concentration relative to the average concentration in the overall bulk flow. Thus, this technique allows a quantitative measure of the fuel distribution to be determined with successive pictures at successive planes downstream from the injection location in the duct. Data obtained in this way provide a direct, experimental measurements of fuel mixing efficiency, and can be used to calibrate computations of the type described in the following section. The overall intent is to use this approach to provide a calibration of CFD codes to determine the level of hypersonic combustion performance achievable in flight.

Computational Simulation

Much has been written about the computer age and the impact of modern computational fluid dynamics (CFD) on engineering in general and on aeronautics in particular. CFD has an important role to play in the understanding of hypersonic combustion. However, it must be recognized at the outset that some of the important physics of hypersonic combustion flows are either not included or at best only crudely modeled in even the most advanced computer codes. For instance, chemical reaction occurs at the molecular level but CFD treats the flow of a continuous fluid. Further, turbulence is generally modeled by some physical analogy rather than being computed directly, and any interaction between turbulence and chemical reaction is generally ignored. Also, the physical modeling of turbulence is based entirely on empiricism derived from velocity fluctuation measurements in incompressible flows; but it is known that compressibility introduces some effects that tend to suppress the generation of turbulence, and that the assumption of simple analogous behavior of mass, momentum and energy transport is inadequate for combustion flows.

In spite of these factors, CFD predictions of complex fully three-dimensional hypersonic combustion flow fields are possible today on a routine basis. Examination of three-dimensional CFD solutions can provide valuable insight to complex hypersonic combustion flows. If these computational results can be "calibrated" with data from experimental simulations of hypersonic combustion at the appropriate conditions, then in spite of incomplete physics in the analysis, the code results can become an important design tool. Once a code is "bench marked" against data, it can be used to conduct parametric variations in design variables to help improve and refine a given configuration.
It should be recognized that since the actual physical processes going on in a hypersonic combustion flow are much more complex than the CFD models, the most complex CFD model should not necessarily be the tool of first choice for analysis. For instance, the added computational complexity of finite-rate chemistry may not be justified in a hypersonic combustion flow field where reactions may not begin to proceed until the scale of the turbulence has reached a level where adequate molecular collisions of fuel and oxidizer can occur, as suggested by Swindenbank (ref. 53). Following this, local chemical equilibrium may prevail, as in the diffusive burning model for supersonic combustion envisioned by Ferri (ref. 41). A computational approach employing a combination of chemically frozen flow and a simple complete reaction model, much like the sudden freezing model developed by Bray (ref. 54) in the 1960's for the reverse process in nozzle flows, could be more accurate than a very complex finite-rate reaction scheme that models turbulence-chemistry interactions as augmented laminar diffusion. The point is that the complexity of the various parts of the computational model should be balanced to achieve efficient engineering results, as well as consistency with the real physical processes.

**Computational Methods**

Today, a multilevel approach is utilized by most numerical application organizations. This multiple-level approach uses simplified methods to screen problems, followed by more sophisticated and costly approaches for selected problems. Hypervelocity scramjet combustor flows feature predominantly supersonic flow, with only small separation regions associated with predominantly axial injection, small if any base regions, little ignition delay (due to high static temperature), and extremely cold wall—tending to stabilize the boundary layer. These characteristics allow credible solutions with the lower level approaches. CFD application for scramjet combustors at NASA relies on three codes: SPARK (refs. 55 and 56), SHIP (ref. 57), and GASP (ref. 58). These codes provide flexibility in evaluation of combustor flow characteristics and performance. The SHIP PNS code, which utilizes a boundary condition to allow PNS modeling of separated flow regions, is very fast, and used for both screening and trade studies (such as demonstrated in ref. 59). GASP or SPARK, on the other hand, are used for in-depth evaluation of flow details (ref. 60), and verification of the screening results.

**NASA LaRC Combustor CFD Codes**

<table>
<thead>
<tr>
<th>Code</th>
<th>Formulation</th>
<th>Turbulence</th>
<th>Chemistry</th>
<th>Relative Solution Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>SPARK</td>
<td>Complete</td>
<td>B-L</td>
<td>General FR Chem.</td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td>PNS</td>
<td>B-L</td>
<td>General FR Chem.</td>
<td>0.2</td>
</tr>
<tr>
<td>SHIP</td>
<td>PNS</td>
<td>B-L, K-ε, Q-ω, Comp. Correct.</td>
<td>Complete Reaction</td>
<td>0.1</td>
</tr>
<tr>
<td>GASP</td>
<td>Complete or Space Marching</td>
<td>B-L, K-ε</td>
<td>General FR Chem.</td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>B-L, K-ε</td>
<td>General FR Chem.</td>
<td>0.3</td>
</tr>
</tbody>
</table>

All of these methods have been demonstrated capable of predicting scramjet fuel mixing and combustion efficiency. Because of the large grid requirements to resolve the combustor wall and injected fuel mixing and combustion simultaneously, only the SHIP PNS approach is used to resolve the entire combustor flow field. The GASP code has been verified capable of predicting separated flow regions associated with shock/boundary-layer interaction and flow over steps, using the Goldberg corrected Baldwin Lomax turbulence model (ref. 61).

Confidence in CFD's predictive capability has been enhanced by comparison with experimental cold-flow mixing simulations of high-speed scramjet combustors, including transverse hydrogen injection into a Mach 4 airflow (refs. 72-74), helium into Mach 3 and 6 airflows (refs. 68-71), and flush wall and ramp injection of air into a Mach 2 airflow (ref. 72-74). Generally, agreement with fuel distribution, peak injectant concentration decay, and fuel mixing efficiency are acceptable (refs. 59-60 and 75, 76) in the combustor "far field" (i.e., at length required for near complete mixing). Typical examples for the three codes are presented in the following paragraphs.

Comparison of GASP predicted and experimentally measured injectant mole fraction contours for staged, transverse air injected into a Mach 2 air test gas are presented in figure 30 (ref. 76). Fuel
penetration, spreading, and decay of the plume core maximum concentration are accurately modeled, indicating that the GASP code models the flow physics to an acceptable level to reliably predict complex combustor fuel injection and mixing flow fields. Comparison of SPARK predicted and experimentally measured injectant mole fraction contours for low-angled, flush-wall helium injected into a Mach 3, low-enthalpy air test gas are presented in figure 31 (ref. 77). Excellent agreement with the vertical centerline helium mass fraction contours indicate that the SPARK code can accurately model fuel mixing flow fields. Unfortunately, the figure of merit for these "validation" studies is not fuel contour replication, but a bulk parameter used in design analysis, *fuel mixing efficiency*.

Most low enthalpy experimental tests provide details of the flow field, but not sufficient definition to calculate the fuel mixing efficiency used in engine cycle analysis modeling of scramjet performance. Historically, mixing efficiency, $\eta_m$, is defined (ref. 63) as that fraction of the least available reactant (i.e., oxygen or fuel) which would react if the fuel-air mixture were brought to chemical equilibrium without additional mixing. Thus in fuel rich regions, all of the local oxygen is considered "mixed", while in fuel lean regions all of the fuel is mixed. Two definitions of mixing efficiency are required—one for flows which are globally fuel rich, and one for flows which are lean. For fuel lean flows,

$$
\eta_m = \frac{m_{H_2, mix}}{m_{H_2, total}} = \frac{A_{\alpha=0} \alpha_R \rho u dA}{A_{\alpha=0} \alpha_R u dA}
$$

where: $\alpha$ is hydrogen (fuel) mass fraction:

$$
\alpha = (1-\alpha) \frac{\alpha_s}{1-\alpha_s} \text{ for } \alpha > \alpha_s
$$

$\alpha_s$ is the $H_2$ stoichiometric mass fraction (0.0285)

$\alpha_{s=0}$ is the area enclosed by zero $H_2$ contour defining the extent of mixing region

$m_{H_2, mix}$ is mixed $H_2$ mass flow rate

$m_{H_2, total}$ is total flow rate from the flow-field integration.

When $\alpha_{max} < \alpha_s$, $\eta_m$ equals 1.0.

Comparison of CFD predictions with experimentally measured fuel mixing efficiency is presented in figure 32, which illustrates a comparison of the SHIP3D PNS calculation (ref. 59) for transverse hydrogen injection into a Mach 4 airflow (ref. 66). Results of the SHIP calculations for these 6 injector configurations are within +/- 10 percent of the experimental correlation which was developed and presented in ref. 66. In fact, SHIP predictions are in better agreement, and exhibit the same trends as, the experimental data. This comparison illustrates that the PNS code can accurately model the fuel mixing, even for transverse injectors, which have small upstream regions of flow re-circulation. Comparisons with ramp injectors and low angled flush wall injection (ref. 57) exhibit similar accuracy in modeling fuel injector flow physics, and fuel mixing efficiency.

Recent advances in pulse tunnel combustor technology have allowed direct quantitative measurement of fuel mixing in hypervelocity combustor flow fields, using the fuel plume imaging discussed above. These images are integrated to provide fuel mixing efficiency. A typical experimental
fuel plume image is presented in the right half of figure 33. This image was obtained at flight Mach 13.5 simulation in the HYDROPE expansion tube facility. Fuel was injected (φ =1.0) 3 inches upstream of this image, using a swept ramp injector photographically scaled from that tested by Northam (ref. 78). This image illustrates the effect of vortical stretching of the fuel plume, and is integrated to provide fuel mixing efficiency. Numerical simulation using the SPARK code, presented on left, closely approximates the fuel plume shape, penetration and spreading, peak fuel mass fraction, and integrated fuel mixing efficiency. This (and other) comparison demonstrates acceptable simulation of the fuel mixing process in these hyper-velocity combustor conditions, and justifies the use of CFD for design studies. (Additional "validation" of CFD for these flows is required, as data become available.)

**Combustor Performance Index - Thrust Potential**

Use of CFD for combustor design results in more information than generally available from experimental studies. Trends from this mass of information can be effectively utilized to optimize the combustor and engine design, but only by selection of appropriate parameters for determination of design goodness. One measure of the combustor performance is energy availability (refs. 79 and 80), defined as the combustor exit flows potential for generating vehicle thrust. Energy availability is destroyed in the burner by injector blockage, mixing, wall heat transfer, and frictional drag on the injector and combustor walls. Generally, it is increased by fuel injectant momentum, the axial wall pressure integral, and the release of energy into the flow by exothermic chemical reaction. The combustor exit flow expands in the nozzle and along the after body where the bulk of the work potential added in the combustor is realized by increased axial wall pressure—hence the generation of vehicle thrust. The ultimate engineering significance of any combustor analysis, experiment or numerical simulation must be measured by its success in increasing understanding of the ability of the engine to produce thrust. The goal of such studies must lead to understanding of and accurate prediction of the thrust potential for use in vehicle design efforts.

Earlier work on flow losses (or thrust potential) has been performed by many workers, notably by Swithenbank (ref. 79) who identified concerns with mixing enhancement strategies which could entail greater flow losses than performance gains recovered from the additional mixing. Czysz and Murthy (ref. 80) present an excellent treatment of useful work availability (or exergy) in high-speed propulsion systems. Kamath, et al. (ref. 81) among others, have used an inverse approach, using an entropy-based approach to the description of flow losses (effectively 'lost' work rather than 'available' work). The performance parameter presented herein, thrust potential, was developed by Riggins (refs. 82 and 83).

The approach taken to analyze thrust potential is twofold. First, the three-dimensional CFD-generated flow field is one-dimensionalized (ref. 83), using a scheme which conserves all mass fluxes (including individual species mass fluxes), momentum fluxes, and energy fluxes between three-dimensional solution and the one-dimensional representation of the solution. Next, the flux-equivalent one-dimensionalized flow is expanded (from any or all cross-sectional plane in the combustor) in an ideal, or reference, nozzle. This ideal expansion to a referenced area or pressure provides a net combustor-nozzle thrust or the net thrust potential. This net thrust potential is then either nondimensionalized by an ideal thrust, to form a combustor effectiveness parameter (ref. 83), or the actual net thrust is used directly, either for comparative purposes or to build more sophisticated design models. This latter approach is most useful in trade studies on a specific engine (inlet/nozzle) or flight vehicle configuration. More sophisticated engine or vehicle design models use this combustor-nozzle thrust potential in conjunction with inlet and weight models and incorporate thermal balance to specify minimum fuel equivalence ratio, φcool.

The result of applying this technique to "post-process" a typical scramjet combustor CFD solution is illustrated in the sketch below. This sketch presents a typical distribution of thrust potential though a combustor. For this example, the inflow plane "A" has some thrust generating potential (as inlet drag was not considered in this case). Upstream of the fuel injection, "B", frictional drag and heat loss to both the combustor walls and to any intrusive injector surfaces, injector generated shock waves and pressure drag, reduce the thrust potential. The region of the flow between "B" and "C" is dominated by the injection of fuel itself; the sharp increase being due to the jet axial momentum addition.
Exothermic reaction occurs in region "C" to "E" and is responsible for increasing the thrust potential; in the region "C" to "D" the benefit of energy release associated with the mixing and combustion overcomes the ever-present losses due to friction, heat transfer to the combustor walls, and shocks. Finally, although combustion continues in the region between "D" and "E," the gain is overcome by the losses. The designers task is to find injector and combustor designs which optimize the peak thrust potential, with acceptable peak heat flux, total heat load, and engine length (weight).

The greatest limitation of this method of quantifying combustor thrust potential is associated with the one-dimensionalization process. Irreversible loss in the available thrust potential, which is analogous to that occurring in a multistream mixing process, can be seen both in an increase in integrated entropy or a corresponding decrease in thrust potential. Evaluation of the magnitude of the mixing loss resulting from the one-dimensionalization process has been studied by individually expanding the flow in each cell in the combustor CFD flow-field plane, and summing the resulting thrust potential. The thrust potential obtained without averaging is approximately the same as that computed by using the one-dimensional method, with errors less than 2 percent for most cases examined, except in the injection near-field, where the flow is highly nonuniform and errors associated with one-dimensionalization are most significant.

Application of the thrust potential model to compare competing fuel injector concepts is demonstrated in figure 34 (ref. 83). This thrust potential comparison is for a Mach 7 flight simulation of an expanding combustor, with two fuel injectors: a 30° flush-wall sonic injector, and a 10° swept-ramp injector. Both injectors provide comparable fuel mixing and combustion efficiency. Differences in the thrust potential are seen ahead of injection, where ramp drag reduces the thrust potential of the ramp injector. In the region downstream of injection, axial-directed jet momentum imparts a large increase in thrust potential. In the nearfield, $0.15 < x < 0.3$, the ramp injector mixes/reacts faster than the flush wall injector, hence the rapid rise in thrust potential. In the far field, mixing for the wall jet catches up with the ramp case, and the thrust potential exceeds that of the ramp higher. The difference between the two cases is about equal to the ramp injector drag. The small differences observed in figure 34 are greatly exaggerated in the Mach 15 regime, where, for example, combustor shear increases dramatically, to 25-50 percent of the engine net thrust.

Concluding Remarks

The purpose of this chapter was to present an assessment of scramjet combustor technology, following 8 years of extensive effort to expand this technology for single-stage-to-orbit speeds, approaching Mach 20. Facilities and test techniques have been brought on line to provide combustor simulation to Mach 14-20, albeit at somewhat low-flight dynamic pressures. Methods for extending these to full simulation were discussed, and plans are under way to accomplish this expansion in the near future. Advances in instrumentation for pulse facility testing currently provide accurate measurement of scramjet fuel mixing, wall pressure and heat flux; and work is progressing to resolve both combustion efficiency and combustor wall shear. Advances in CFD application have provided solutions which, although lacking in some physics, provide significant insight into this new, complex high-speed combustor environment.
Nevertheless, comparisons with existing data provide surprising similarity. Improvements in design methods are based on the experimentally anchored CFD methods, such as the use of thrust potential. These design methods provide a consistent methodology for improving the combustor design.

NOZZLE

The nozzle completes the propulsion flow path, and has the job of expanding the high-pressure and temperature gas mixture generated within the inlet and combustor into a high-velocity exhaust with greater momentum than the captured airflow, thus generating net thrust. In the hypersonic vehicle, this is accomplished by expanding the combustor exit flow starting within the engine module and continuing the expansion over a large portion of the aftbody of the vehicle. In this expansion process, potential energy is changed to kinetic energy, and the shape of the aftbody gives a direction to the propulsion flow path, setting the angle of the gross thrust vector relative to the vehicles flight direction. These two factors must be given primary consideration in the design of the hypersonic nozzle: the efficient generation of thrust, and the aerodynamic balance of the vehicle. At hypersonic speeds, these factors are critical since the nozzle is working on the total flow through the engine and net propulsive thrust is a small difference between two large numbers, i.e., the gross thrust exiting the nozzle and the ram drag of the streamtube entering the inlet. The nozzle thrust direction dominates the trim of the vehicle at hypersonic speeds. The impact of these factors on the design of the combined vehicle and propulsion system is discussed in the next section along with other nozzle issues that must be considered at lower speeds.

Performance of the nozzle is a result of the upstream flow process as well as the flow process that occurs within the nozzle. There are five principal loss mechanisms that will be discussed: flow profile at the nozzle entrance, failure to recombine dissociated species, skin friction, flow divergence, and underexpansion losses. Flow profiles at the nozzle entrance are a result of boundary-layer growth from the bodyside and cowl surfaces, inlet shock waves that were not canceled at the throat and passed through the combustor, and the fuel injection, mixing and combustion process. The effect of these profiles is not clear, given the complex nature of their interaction with the expanding flow and their effects on the other loss mechanisms. There has been some thought of altering the flow profile with speed by changing the way fuel is proportioned between bodyside and cowl fuel injectors, thereby altering the gross thrust vector angle and improving the aerodynamic balance of the vehicle over a range of Mach number.

The effect of the other flow mechanisms on nozzle performance are somewhat better understood, and always result in a loss to the scramjet flowpath. A typical measure of nozzle performance is CFG, which is defined as gross thrust of the streamtube at the exit of the nozzle referenced to the equilibrium ideal thrust of the flowpath streamtube expanded to the free-stream static pressure. A good nozzle would have a value of CFG of about .96, with around .97 representing a practical upper limit. The magnitude of these losses as well as their relative importance is a strong function of the design approach for the propulsion system. Perhaps the most complex of these is the dissociation losses, which are those losses resulting from the flow "freezing" in the rapid expansion process within the nozzle so that energy is not generated by the recombination of free radicals produced by the combustion process. This is not a significant problem at low speeds since dissociation within the combustor is a strong function of static temperature. Thus, features and performance levels within the engine that affect static temperature also affect dissociation and the potential for losses through freezing the flow in the nozzle expansion process. Figure 35 gives the potential for losses in CFG due to freezing dissociated flow for propulsion parameters that include combustor fuel equivalence ratio, inlet contraction ratio, and inlet kinetic energy efficiency for a Mach 14 flight condition. Dissociation losses are maximized at a fuel equivalence ratio of unity corresponding to the highest combustor exit temperatures. Note that the losses are particularly sensitive to the inlet design parameters. A high contraction ratio is desirable at high Mach numbers in order to increase static pressure and temperature for rapid ignition and combustion; but, as can be seen, high contraction must be limited to avoid high losses due to kinetic freezing affects. In addition, kinetic losses are particularly sensitive to a reduced inlet kinetic energy efficiency. Analysis of finite-rate chemical kinetics of the expansion process generally leads to the conclusion that the flow is very close to a frozen process. Efforts to minimize this loss must rely on revising component design parameters to limit static temperature and dissociation at the combustor exit (such as some combustor divergence) or find ways to catalyze reaction in the nozzle flow and drive the expansion process further toward equilibrium.
The effects of friction within the nozzle on nozzle CFG is illustrated in figure 36. Friction is reduced as the flow expands at the nozzle throat, but because the nozzle is so large the total drag becomes significant. A larger portion of the friction is generated in the first half of the nozzle where pressures are higher so that the behavior of the boundary layer in the initial portions of the nozzle is most important in controlling friction and heat transfer. Efforts have been made to evaluate the possibility of the boundary relaminarizing just downstream of the nozzle throat (ref. 84). The forming of laminar flow within the nozzle would significantly reduce friction drag and heat transfer.

The final two losses that will be considered are due to flow divergence and underexpansion. Flow divergence is a result of streamlines at the nozzle exit being at different angles, or internal flow divergence, and the mean flow direction being at a direction different than that of the flight path. Internal flow divergence is a result of the nozzle shape and three dimensional features, and can be analyzed using CFD calibrated against test results (ref. 85). Underexpansion losses are a result of the nozzle not being large enough to fully expand the flow to the ambient pressure, and is constrained by the physical size of the airframe aftbody. Note that nozzle performance can be very sensitive to both of these nozzle design parameters.

ENGINE/VEHICLE SYSTEM INTEGRATION

Forebody / Inlet

The vehicle forebody serves as the external compression portion of the inlet. In designing the hypersonic inlet, it is desirable to maximize external forebody compression to minimize internal inlet surface area and heat load. In addition to providing inlet compression, the forebody must provide a uniform distribution of air across the vehicle ahead of the propulsion module. Figure 38 illustrates a poor forebody design in that the static pressure distribution ahead of the propulsion modules results in a large accumulation of boundary layer in the center of the forebody. Such an airflow distribution would cause an unacceptably thick boundary layer and airflow loss in the center propulsion module. The importance of finite-rate chemistry in calculating lateral airflow distribution as well as flow-field profiles between the body and cowl is also illustrated in this figure. Studies have been conducted (ref. 86) to determine forebody shapes that maximize compressive performance and minimize airflow distortion.

In addition to airflow compression and distribution, forebody shape also plays an important part in determining the point of boundary-layer transition. A delayed transition from a laminar to a turbulent boundary layer reduces friction drag on the forebody and has a significant effect on overall vehicle drag at hypersonic speeds. Also, a reduced rate of boundary-layer growth results in thinning the boundary layer entering the propulsion module, thereby increasing the airflow processed through the propulsion system. The result would be a significant increase in payload for an SSTO vehicle (fig. 39). Unfortunately most aerodynamic research is conducted in "noisy" facilities that have turbulence levels sufficient to affect boundary-layer transition. Research in "noisy" tunnels would predict a much earlier transition point on the forebody as indicated by a comparison with "quiet" tunnel data at supersonic speeds. In addition, a higher transition Reynolds number would be predicted for a cone in a "noisy" tunnel as opposed to a flat plate, although linear theory and "quiet" tunnel data would give the opposite result. Confirming these trends at high hypersonic Mach numbers will be important to define the forebody shape that would be most desirable to maximize vehicle performance.

Nozzle / Aftbody

A unique feature of the hypersonic airframe-integrated propulsion system is a propulsion module separating large forebody and aftbody surfaces (fig. 40). The detailed design of these surfaces is dictated by propulsion requirements at the high end of the vehicle's design speed range, resulting in extreme off-design conditions at lower speeds. These problems start early, as illustrated in figure 40. At rotation, the large aftbody becomes nearly parallel to the ground plane, and is subject to a reduced pressure resulting in a loss of lift when propulsion exhaust is simulated. These effects, which are influenced by elevon, wing, and nozzle geometry (ref. 87), could result in a serious design penalty, given the inherently poor aerodynamic performance of hypersonic vehicle shapes at subsonic speeds.
Another consequence of the large aftbody is high transonic drag. The nozzle must be designed for a very high-pressure ratio at hypersonic flight conditions, leaving a highly overexpanded nozzle at transonic speeds. Figure 41 summarizes results from nozzle tests conducted in the 16-Foot Transonic Tunnel at Langley Research Center. Results show that pressure drops below atmospheric pressure immediately downstream of the cowl exit, and then recovers back to atmospheric pressure at approximately half of the nozzle length. The downstream pressure recovery back to atmospheric pressure results from the overexpansion shock originating from the cowl lip as well as outside airflow filling in the large base area. Nozzle sidewall fences inhibited pressure recovery, which leads to a much higher level of base drag. Note in the lower part of the figure that drag maximizes at Mach 1.2 rather than Mach 1, and is a function of nozzle aftbody expansion angle. Tests have also been conducted on a nozzle aftbody (refs. 88 and 89) in the Langley 20-Inch Mach 6 Tunnel (fig. 42) to determine performance characteristics. Parametric tests included the nozzle sidewall fence and air or simulant gas to represent the nozzle exhaust flow. The simulant gas was a cold mixture of gases intended to properly reproduce the engine exhaust flow ratio of specific heats throughout the nozzle expansion process. Note that measured nozzle forces are increased when the exhaust flow is simulated as compared to results using air. In addition, increases in nozzle thrust and lift occur when a flow fence is installed since the nozzle is not overexpanded and exhaust flow containment within the nozzle maximizes thrust at higher speeds. In contrast, at transonic speeds (fig. 41) the configuration without sidewalls would have less base drag.

As would be expected, the nozzle aftbody has the potential for producing large lift forces and resulting pitching moments which would be imposed on the vehicle. Previous studies conducted on research airplane concepts (refs. 90 and 91) have shown the magnitude of the lift force to be a function of the aftbody expansion angle, the vehicle shape, and the axial location of the propulsion module on the vehicle. Effects of propulsion forces as a function of axial engine location are given in (fig. 43). The results shown in this figure assume shock-on-lip at Mach 10, so that airflow and resulting thrust at the combustor exit increase as the propulsion module is moved aft and the distance between the body and forebody shock wave becomes greater. Nozzle expansion angle also increases along with a decrease in nozzle length, both acting to reduce lift. The result is a nose-up pitching moment that requires a positive elevon deflection to trim the vehicle (fig. 44), which in turn produces a drag and reduces thrust margin. At hypersonic speeds, elevation trim drag can dominate overall vehicle drag leading to a large loss of installed propulsive performance. A challenge of the hypersonic vehicle is to deal with a balance between thrust, lift, and pitching moment over a wide speed range and at different propulsive power settings, imposing additional criteria on the design of the integrated propulsion system.

CONCLUDING REMARKS

This chapter has presented an overview with depth added in selected parts of the 30+ years of research involvement with airframe-integrated ramjets and scramjets at the NASA Langley Research Center. Following the recent resurgence of interest in hypersonic airbreathing propulsion for single-stage-to-orbit vehicles revolving around the National Aero-Space Plane (NASP) Program (perhaps more appropriately termed "Aerospace Plane II"), the industry and the country are at a decision point for hypersonic propulsion technology. The prudent path would seem to follow a continued effort to exploit the NASP investment in infrastructure and technology by continuing a highly focused program to continue refinement and innovation in hypervelocity propulsion technology. Given the changing world political situation and reordered national priorities of the 1990's, the logical culmination of this propulsion technology program might be a rocket-boosted demonstration of a specially designed high-performance scramjet engine flight package rather than an orbital-capable manned airplane (i.e., the X-30) which was the goal of the NASP Program. While many challenging and interesting issues and problems remain in the Mach less than 8-10 speed range, the real proof of principal for (airbreathing) scramjet application to orbital transportation is in the level of performance achievable in flight at Mach 16+. The NASP investment has provided the tools (impulse facilities, CFD codes, measurement techniques, trained people and infrastructure) to pursue and demonstrate the required propulsion technology. The challenge remains to apply these tools to bring the Aerospace Plane II era to a successful conclusion in the 1990's and make available high-performance hypervelocity propulsion as a viable option for future space transportation systems.
ACKNOWLEDGMENT

The authors are pleased to acknowledge the numerous contributions of their colleagues in NASA, other government labs, and the aerospace industry to the ideas and accomplishments described in this paper. While all are recognized at least in part by direct reference where possible, specific additional mention is due Dr. Aaron Auslander for his contributions to the sections on "Flow Nonuniformities..." and "Interactive Inlet Design." Also, credit for the ideal to use smoke and mirrors (Mie scattering) infuel plume imaging is due in part to Dr. Clay Rogers at Langley, and for successful implementation of the technique, credit is due to the HYPULSE team at GASL including at least Dr. John Erdos, Dr. Jose Tamagno, Mr. Rich Trucco, and the dedicated staff who support them.

NOMENCLATURE

Al  Ramp contraction angle
a, b, c Constants in Thrust correlation, equation (3)
Cs  Vacuum thrust coefficient
cp  specific heat at constant pressure, J/kg
D1  Damkohler's first number (see text)
D2  Damkohler's second number (see text)
Dhc heat of combustion, J/kg
F  Stream thrust, (P+rv^2)
Fs  Film fuel fraction
f  fuel equivalence ratio ñ (f/a)/((f/a)stoichiometric
f/a  fuel air ratio
GW  wall-to-stream stagnation enthalpy ratio (see text)
H  stagnation enthalpy, J/kg
Hc  Combustor height
h  static enthalpy, J/kg
h  Ramp height, in
h  efficiency
L  length, m
Ls  Combustor width fueled by on ramp
M  Flight Mach number
ni  species volume fractions
P  static pressure, kPa
Pr  Prandtl number (see text)
q  wall heat flux, J/m^2sec
q  angle, degrees
RC  Transition section contraction parameter
Re  Reynolds number (see text)
r  density, kg/m^3
St  Stanton number (see text)
T  static temperature, K
T  Engine Thrust, Station 0 to 8, Lb/ft-width.
t  time, sec
u  velocity, m/sec
w  Ramp base width, in.
α  mass fraction
f  fuel equivalence ratio
φ  fuel equivalence ratio = (f/a)/(f/a)stoichiometric
η  efficiency
νi  species volume fractions
θ  angle, degrees
ρ  density, kg/m^3
Subscripts

c \quad \text{combustor or combustion}
m \quad \text{mixing}
w \quad \text{wall value}
o \quad \text{Free stream, Station "o"}
2 \quad \text{Transition section entrance, end of "2-D" inlet}
2.5 \quad \text{Transition section exit, Combustor entrance.}
8 \quad \text{Nozzle exit (h=174")}

REFERENCES


17. Dr. P. J. Ortwerth; private communications.

18. Dr. V. Quan; private communications


33. Ta'asah, S., Kuruvila, G., and Salas, M. D.; Aerodynamic Design and Optimization in One Shot, AIAA 92-0025.


49. Private Communication: Dr. Aaron H. Auslender, Lockheed Engineering and Sciences Company.

50. Private Communication: Facility data from Reference 7 were used to compute test section conditions with the NENZF computer code.


Ramjet, $Ma = 3-6$

- High inlet contraction ratio at Mach 6
- Subsonic diffuser
- High inlet losses and low Rayleigh losses
- Nozzle throat throttle control
- High internal pressures

Scramjet, $Ma = 6-20$

- Low inlet contraction ratio at Mach 6
- No subsonic diffuser
- Low inlet losses and high Rayleigh losses
- No nozzle choke required
- Low internal pressures

Fig. 1 Hypersonic engine cycles

Fig. 2 Engine Performance
Stoichiometric heat release \[ \approx \frac{69}{Ma_{\infty}^2} \]

- Scramjet performance is most sensitive to component efficiencies at high Mach numbers.
- High energy level of airstream required for combustion research.

Fig. 4: Hypervelocity scramjet design challenge
Figure 5.

STATIC CONDITIONS AT INLET THROAT

M = 14, Q = 2000psf, γ = 1.4

NOTE: MUST HAVE EFFICIENT INLET TO AVOID DISSOCIATION

Static pressure, psia

High efficiency inlet
Four shock inlet

Two shock inlets

Initial δ

- 6
- 8
- 10

Static temperature, °R

1800 2000 2200 2400 2600 2800 3000 3200 3400 3600

Figure 6.- Airframe-propulsion system integration.
FIG. 7

AFT-SWEEP vs FORWARD SWEEP INLETS

All sweep
- Inlet is easier to start
- Increased flow spillage
- Increased throat flow distortion

Forward sweep
- Inlet is harder to start
- Increased mass capture
- Reduced throat flow distortion

---

$M_1 = 4.03, \text{ Sweep } = 30^\circ, \frac{C_x}{T_x} = 1.0$

- Inlet started
- Inlet unstated
- Inlet self starting

Inviscid theory
- Viscous, $\delta^* = 0.04$ in.

Inviscid, 0 sweep

---

Fig. 8 - No-strut inlet comparison with data.
Figure 9. Single fin interaction flowfield projected onto a conical crossplane.

Navier-Stokes Solution  Euler Solution

**FIG. 10, CFD AT INLET THROAT**
Figure 11. Bodyside centerline pressure distributions at maximum back pressure. \(W/g=3.0\).
**FIG 12.** 2-D INLET/ISOLATOR PARAMETRIC MODEL

**FIG 13.** ISOLATOR BACKPRESSURE CHARACTERISTICS

- Short cowl, no foreplate
- Cowl deflected down 2.5°

- No back pressure
- Approaching
- Maximum
- Back pressure
- Inlet unstart

Pressure rise across inlet and isolator:

Distance through inlet, isolator and nozzle
Figure 16. Total enthalpy requirements for hypersonic combustion simulation.

Figure 17. Total enthalpy capability of selected pulse facilities for hypersonic combustion simulation.
Figure 3. Total pressure capability of selected facilities for hypersonic combustion simulation.

Figure 4. Ground simulation of scramjet flight combustor conditions with stagnation heating.
Energy relative to stoichiometric heat release for H₂-air combustion

Figure 5. Ground simulation of scramjet flight combustor conditions comparing stagnation and non-stagnating heating.

Figure 6. Test gas oxygen dissociation level at simulated combustor entrance conditions.
Figure 7. Mach 17 combustion experimental apparatus - axisymmetric model.

Figure 8. Comparison of combustion pressure rise.
Figure 9: Assessment of experimental Mach 17 combustion pressure rise.

Figure 10: Computed energy yield for Mach 17 combustion experiment.
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Water mass fraction</td>
<td>1.0</td>
<td>+/- 5%</td>
<td>+/- 5%</td>
</tr>
<tr>
<td>Water number density</td>
<td>0.8</td>
<td>+/- 5%</td>
<td>+/- 5.1%</td>
</tr>
<tr>
<td>Total Wall heat load</td>
<td>3.4</td>
<td>+/- 3%</td>
<td>+/- 10%</td>
</tr>
<tr>
<td>Oxygen mass fraction</td>
<td>0.6</td>
<td>+/- 80%</td>
<td>+/- 48%</td>
</tr>
<tr>
<td>Average pressure</td>
<td>3.4</td>
<td>+/- 3%</td>
<td>+/- 10%</td>
</tr>
<tr>
<td>Oxygen utilization</td>
<td>1.0</td>
<td>+/- 80%</td>
<td>+/- 80%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Peak fuel mass fraction</td>
<td>0.5</td>
<td>+/- 5%</td>
<td>+/- 2.5%</td>
</tr>
<tr>
<td>Peak fuel number density</td>
<td>0.9</td>
<td>+/- 5%</td>
<td>+/- 5.1%</td>
</tr>
<tr>
<td>Comb. exit min. temperature</td>
<td>1.1</td>
<td>+/- 80%</td>
<td>+/- 95%</td>
</tr>
</tbody>
</table>

Figure 26. - Performance uncertainty based on various measurement approaches \( M=15, \phi=1.5 \)

Figure 27. - Indicated performance sensitivity vs. number of measurements.
Figure 28. Schematic of fuel plume imaging in HYPULSE.

Figure 29. Fuel plume image processing procedure.
Figure 20: Comparison of injectant mole fraction contours with PLIF image, in the streamwise planes (Y/D=0.0, Y/D=0.5).

Figure 30: Comparison of Experimental (VPI & SU) and Numerical simulation of a 30° flush wall injection into a Mach 3 air flow.
Figure 32.- Predicted and measured fuel mixing for normal sonic H₂ injection into a Mach 4 airflow.
CFD - Data Comparison of Fuel Plume
10° SRI, Mach 13.5 in HYPULSE
H₂ / H₂O / SiO₂(seed) Fuel, Phil(H₂) = 1, 3" From Jet

<table>
<thead>
<tr>
<th>Computational</th>
<th>Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normalized H₂</td>
<td>Normalized Intensity of Mie Image in Fuel Plume</td>
</tr>
<tr>
<td>Number Density Flux</td>
<td></td>
</tr>
</tbody>
</table>

ηₘ = 0.44

ηₘ = 0.42

Figure 33.
Fig. 35. Effect of Inlet & Combustor on Nozzle Kinetic Losses @ M = 14

Fig. 36. Effect of Nozzle Friction on Performance @ M = 14
Fig 37: Effect of Divergence and Underexpansion on Nozzle Perf, QM=19
Fig. 28 Forebody analysis on a typical SSTO vehicle

Fig. 29 Effects of boundary-layer transition on vehicle performance
Fig. 21. Hypersonic vehicle at takeoff.

Fig. 41. Aftbody performance at transonic speeds.
Fig. 6  Hypersonic nozzle exhaust simulation; effect of flow fence and simulant gas

Fig. 7  Propulsion forces as a function of scramjet location at Mach 10
Fig. 7: Engine axial location effects at Mach 10