DAVID M. VAN WIE, MICHAEL E. WHITE, and GRIFFIN P. CORPENING

NASP INLET DESIGN AND TESTING ISSUES

An efficient inlet will be required for the successful operation of the National AeroSpace Plane. The major issues related to the prediction and measurement of the performance of scramjet inlets are presented.

INTRODUCTION

The primary purpose of an inlet for any air-breathing propulsive system is to capture and compress air for processing by the remaining portions of the engine. In a conventional jet engine, the inlet works in combination with a mechanical compressor to provide the proper compression for the entire engine. For vehicles flying at supersonic ($1.5 < M_\infty < 5$) or hypersonic ($M_\infty > 5$) speeds, adequate compression can be achieved by the inlet without a mechanical compressor. Because the airflow and compression ratios of the scramjet engine are provided by the inlet, an efficiently designed inlet is critical for the successful operation of the engine.

The design of an efficient inlet for the National AeroSpace Plane (NASP) will be a challenging task because of the large operating envelope. The flow-field structure within the inlet is a strong function of freestream Mach number and altitude, so an inlet designed for one speed may not be acceptable for other speeds. In general, hypersonic inlets must be designed for high speeds because the engine is extremely sensitive to losses within the inlet at those speeds. Therefore, the design of an inlet for NASP involves defining an inlet shape that operates efficiently at high speed but also provides adequate performance at lower speeds.

At hypersonic speeds, the shock waves generated by the inlet compression surfaces are swept aft at very shallow angles; thus, long inlets are required to capture and compress a given amount of airflow. Because of this feature of hypersonic flow fields, it is advantageous to integrate the inlet into the undersurface of the vehicle forebody to use the compression produced by the vehicle as part of the inlet. This so-called airframe–engine integration is a unique feature of the NASP in that the engine performance will be significantly affected by the orientation of the flight vehicle. Bolt-on engines are a luxury relegated to lower speed aircraft.

The goal of inlet design is to provide high levels of performance over the complete operating envelope of the aircraft. The performance of scramjet inlets can be discussed in terms of the amount of air captured by the inlet and the efficiency of the process used to compress the flow. Many different parameters are used to indicate the performance (see the Nomenclature in the boxed insert). Generally, these parameters allow a one-dimensional representation of the flow field at the end of the inlet, which is useful for implementation in engine cycle analysis.[1] The set of parameters selected to represent the operation must be such that a unique specification of the mass, momentum, and energy of the flow entering the combustor can be determined. The parameters most often used to describe the inlet performance are the air capture ratio, kinetic energy efficiency, and enthalpy ratio. These parameters, together with a geometric description of the inlet, can be combined to calculate the mass, momentum, and energy of the flow entering the combustor.

In addition to functioning with high performance levels over the flight regime, the inlet must have sufficient margin in its operating characteristics to withstand perturbations that may be caused by nonuniformities in the atmosphere, nuances of the flight path, or subtleties of the engine operation. These operability margins must ac-

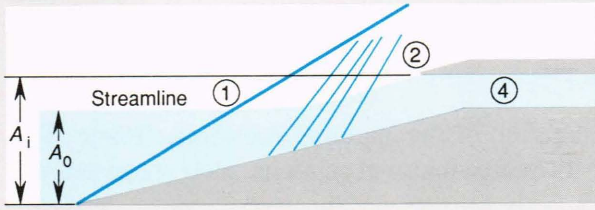
NOMENCLATURE

A	= Area
a	= Speed of sound
C_{O_2}	= Mass fraction of O_2
C_{N_2}	= Mass fraction of N_2
C_P	= Specific heat at constant pressure
C_v	= Specific heat at constant volume
e	= Internal energy per unit mass
$h = e + P/\rho$	= Enthalpy per unit mass
$h_t = h + V^2/2$	= Total enthalpy per unit mass
L	= Length of inlet
$M = V/a$	= Mach number
p	= Pressure
$q = \rho V^2/2$	= Dynamic pressure
$\mathrm{Re} = \rho VL/\mu$	= Reynolds Number
s	= Entropy per unit mass
T	= Temperature
V	= Velocity
$\gamma = C_p/C_v$	= Ratio of specific heats
η_{KE}	= Kinetic energy efficiency
ρ	= Density
μ	= Viscosity

Subscripts

0, ∞	= Freestream conditions
4	= Conditions at inlet throat
w	= Wall conditions
i	= Reference conditions

GLOSSARY OF INLET PERFORMANCE PARAMETERS



Air capture ratio = A_0/A_i — Related to the amount of air passing through the engine.

Enthalpy ratio = h_{t_4}/h_{t_0} — Related to the amount of energy lost from the captured streamtube through heat loss.

Kinetic energy efficiency = $\dfrac{h_{t_4} - h(P_0, S_4)}{h_{t_0} - h_0}$ — Ratio of the useful kinetic energy remaining in the stream to the freestream kinetic energy. Friction, shocks, and heat loss all result in lower η_{KE}.

Additive drag = $\int_1^2 P\,dA_x$ — A pseudo-force required in the momentum balance formulation of the thrust-drag accounting scheme; sometimes called spillage drag.

Compression ratio = P_4/P_0 — Variable closely related to the thermodynamic efficiency of the engine.

Aerodynamic contraction ratio = A_0/A_4 — The amount that the flow is "squeezed."

Geometric contraction ratio = A_i/A_4 — Geometric factor related to inlet design.

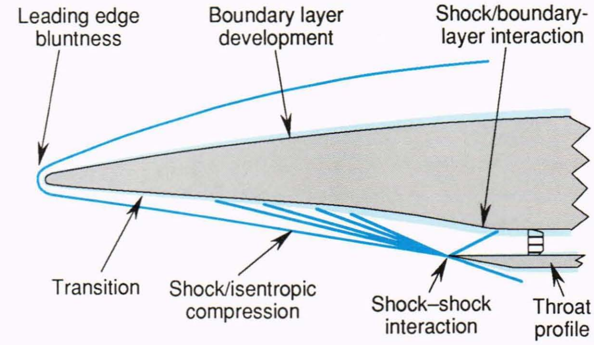


Figure 1. Schematic of the inlet flow-field features that affect inlet performance and operability.

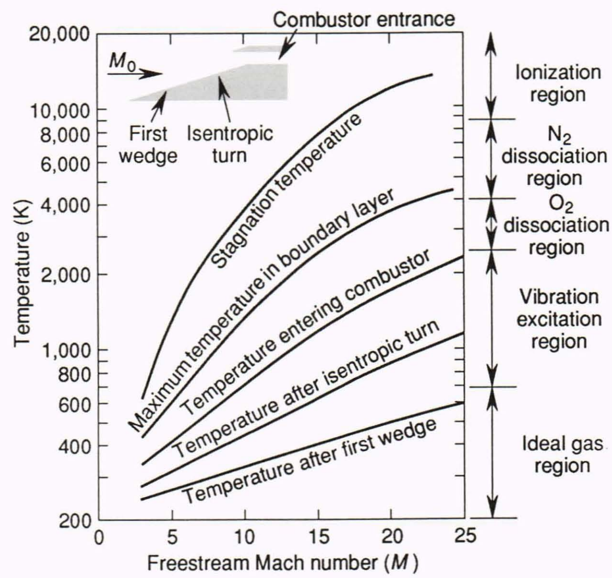


Figure 2. Typical temperatures in various regions through a hypersonic inlet.

count for shock/boundary-layer interactions, the heating limitations of the structure, and contraction/compression ratio limits. The prediction and subsequent validation of these operability limits will be vital aspects of the flight test phase of the NASP program.

Because the vehicle operates over such a wide speed regime, many important high-speed aerodynamic phenomena will exist at some portion of the flight regime on some portion of the inlet. The primary issues affecting performance and operability of an inlet system for the NASP are illustrated in Figure 1. During the development of the inlet for the NASP, many of these issues will be addressed in both analytical and experimental programs. The analytical efforts are based primarily on computational fluid dynamic (CFD) techniques that numerically solve the fundamental partial differential equations that describe the flow of a continuum fluid. In the following sections, several of the primary issues related to the design of scramjet inlets are discussed. Where appropriate, sample results from CFD solutions are used to illustrate the issues. A more detailed description of the uses of CFD can be found in Ref. 2.

HIGH-TEMPERATURE EFFECTS

At high speeds, a significant amount of kinetic energy is contained in the flow, and the temperature of the captured airstream rises as it is slowed either through the compression process or because of viscous effects. For temperatures below 600 K, air can be modeled as a perfect gas, but at higher temperatures the effects of vibrational excitation, dissociation, and ionization can become important. Typical temperatures at various locations throughout an inlet flow field that is in chemical equilibrium are shown in Figure 2. The entire flow field, with the exception of the stagnation region, can be represented by an ideal gas for speeds below approximately Mach 5. For speeds up to Mach 12, the inviscid portion of the flow field can be represented by an ideal gas, but vibrational effects in the boundary layer flow must be considered. At speeds above Mach 12, the vibrational effects in the inviscid flow and dissociation effects in the boundary layer must be taken into consideration. Ionization of the gas is restricted to the stagnation regions at the highest speeds. At flight conditions near Mach 20, the stagnation temperature can reach 11,000 K, which exceeds the temperature of the surface of the Sun.

The consideration of the gas model (i.e., ideal gas, equilibrium mixture, etc.) is important in the design of hypersonic inlets because the position and strength of shock waves can be strongly affected by high-temperature gas effects. If these effects are neglected, the flow structure and resulting inlet performance can be significantly different than predicted. As will be discussed later, this effect has important implications for experiments in ground test facilities because the high-temperature effects cannot be accurately simulated for speeds above Mach 12.

At large Mach numbers and high altitudes, the partition of energy between various states may not remain in equilibrium when the flow properties are changed rapidly. For typical NASP trajectories, these so-called nonequilibrium effects are most important in the stagnation region and in the high-temperature region within the boundary layers. As an example of the types of differences that may result from the nonequilibrium dissociation and recombination of air species, temperature profiles in the cowl-lip plane of a typical inlet at Mach 25 are shown in Figure 3. In this example, the inlet flow field was calculated using the CFD code described in Ref. 3, which solves the parabolized Navier–Stokes equations for a perfect gas, an equilibrium air mixture, or a nonequilibrium air mixture. As can be seen from the results, the high-temperature portion of the boundary layer is significantly different for the three gas models. Outside the boundary layer, an equilibrium assumption generally provides reasonably accurate results over most of the NASP trajectory.

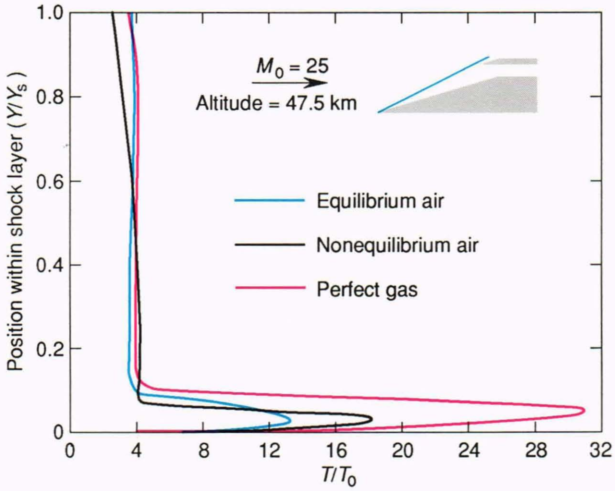


Figure 3. Effect of gas model on the temperature profile at the cowl lip plane of a typical inlet.

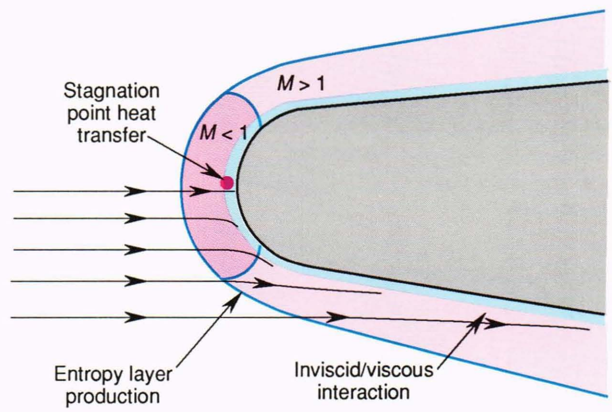


Figure 4. Characteristic flow-field features near a blunt leading edge.

BLUNT LEADING EDGE EFFECTS

Because of the large heating rates, the forebody of the vehicle and the engine cowl lip will be constructed with blunt leading edges. As shown schematically in Figure 4, the flow-field features at a blunt leading edge include a curved bow shock with attendant vorticity production, a subsonic flow region with resulting high fluid temperatures, and the beginning of a boundary layer with resultant high heat transfer rates. These flow-field phenomena can be calculated using CFD techniques to solve the complete Navier–Stokes equations. In addition, several approximate analyses are available for investigating aspects of the blunt nose flow field. The stagnation-point heat-transfer analysis of Fay and Riddell[4] is an example of a simplified analysis that provides important engineering information.

Before quantitatively discussing the stagnation-point heat transfer, an important flow phenomenon that may produce very high heating rates at high speeds needs to be described. This phenomenon occurs at high speeds when inlets are designed such that the forebody bow shock intersects the bow shock produced by the cowl lip. When this shock–shock interaction occurs, a very complicated flow field can result near the cowl lip. Several different types of interactions have been identified, depending on the position and strength of the forebody shock with respect to the stagnation region.[5] As shown schematically in Figure 5, the type of interaction that results in the highest heat transfer produces a combined shear layer/shock system that impinges on the surface of the leading edge. At the point of impingement, a high pressure, high heat transfer region is produced locally. One of the questions that must be answered as part of the vehicle development is whether or not active cooling systems can ensure that the structure will survive under these locally high heat transfer conditions.

By far the most important features of hypersonic blunt-body flow fields are the high temperatures and heating rates that are produced. As an example of the heating levels encountered in a NASP inlet, the stagnation heat transfer rates are shown in Figure 6 for a typical ascent trajectory. In Figure 6, the heating rates on the vehicle nose and cowl lip, as calculated with the Fay and

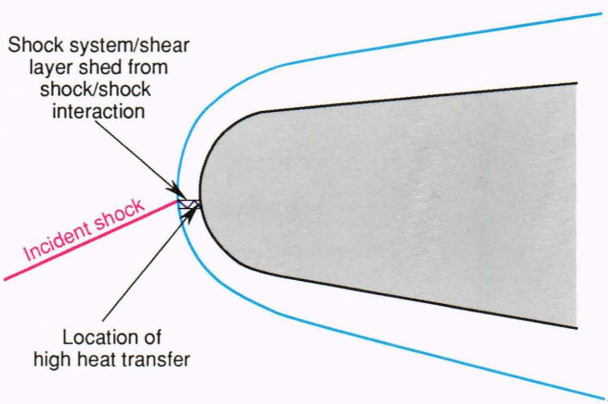


Figure 5. Schematic of a type of shock-shock interaction at the cowl lip that is capable of producing very high heat transfer conditions.

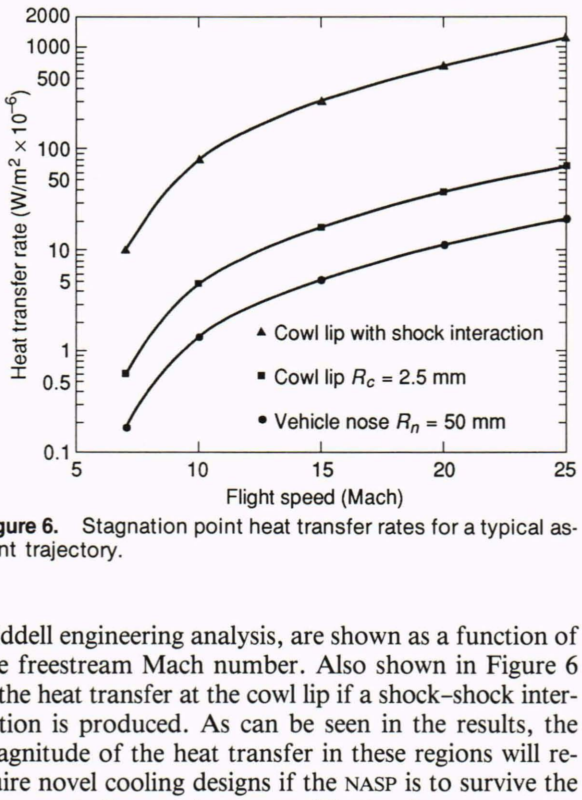


Figure 6. Stagnation point heat transfer rates for a typical ascent trajectory.

Riddell engineering analysis, are shown as a function of the freestream Mach number. Also shown in Figure 6 is the heat transfer at the cowl lip if a shock–shock interaction is produced. As can be seen in the results, the magnitude of the heat transfer in these regions will require novel cooling designs if the NASP is to survive the hypersonic flow environment. (The heating rate may be as high as 1.2×10^9 W/m^2 when a shock–shock interaction occurs at the cowl lip at Mach 25. As a point of reference, the entire output of a moderate-size nuclear power plant would be required to provide this heating rate to a 1-m^2 piece of material.)

BOUNDARY LAYER DEVELOPMENT

The heat transfer and friction losses within an inlet flow field are generally restricted to narrow regions along the surfaces of the inlet, which are called boundary layers. The understanding of the development of these boundary layers is crucial to the understanding of hypersonic inlet operation, because the majority of the inlet losses occur in these regions. The prediction and measurement of these friction and heat transfer losses represent the major challenges in predicting inlet performance.

In addition to the losses within the boundary layer, the interaction between the viscous and inviscid portions of the flow field becomes important at high speeds. This coupling complicates the design process significantly because the majority of the inlet design tools that are available use inviscid flow assumptions. At subsonic and lower supersonic speeds, the boundary layer effects can be superimposed on the inviscid flow field, but at hypersonic speeds, the inviscid and viscous flow fields must be calculated simultaneously, which requires an iterative design procedure.

The development of the boundary layer flow along the inlet is strongly influenced by the variable entropy layer produced by the curved forebody bow shock, by the transition from laminar flow to turbulent flow, by the effects of adverse pressure gradients, and by interactions with shock waves. Currently, a large degree of uncertainty exists in the prediction of transition from laminar flow to turbulent flow in hypersonic boundary layers. This uncertainty exists in both the prediction of the onset of transition and the prediction of the length of the transition zone. These prediction uncertainties lead to a large uncertainty in the inlet performance, and represent one of the primary contributors to the overall uncertainty in hypersonic vehicle performance. The magnitude of the problem can be seen by using estimates of the transition location (such as those found in Ref. 6) and models for the transition length (such as that found in Ref. 7). As an example, Figure 7 shows the heat transfer along the surface of an inlet at Mach 25 for various assumed transition locations and transition lengths. The effect of transition can be seen in Figure 7 as a rapid rise in the local heating rate near the point of transition. For this example, it can be seen that the local heat transfer rate can vary by a factor of 50 depending on the transition assumption. The impact of the uncertainty of transition location on inlet performance can be significant. Various attempts at improving the prediction of transition are under way and include analysis efforts based on laminar flow stability theory and experimental efforts conducted in specially built hypersonic quiet wind tunnels. Despite these various attempts, many investigators believe that a flight experiment may be required to better quantify the transition region.

The interaction of shock waves with boundary layers is another important consideration in all supersonic and hypersonic inlet designs. As illustrated in Figure 8, shock-wave/boundary-layer interactions can result from compression corners, reflected shocks, shock cancellations, or sidewall compression corners. Under certain circumstances, the low momentum portion of the boundary layer cannot negotiate the pressure rise associated with the shock wave, and a separated boundary layer results. The structure of a shock-wave/boundary-layer interaction with separated flows is substantially more complicated than the attached flow counterpart. Large local heating conditions and unsteady flow effects are characteristic features of hypersonic separated flows. Most of these separated flow phenomena are very undesirable.

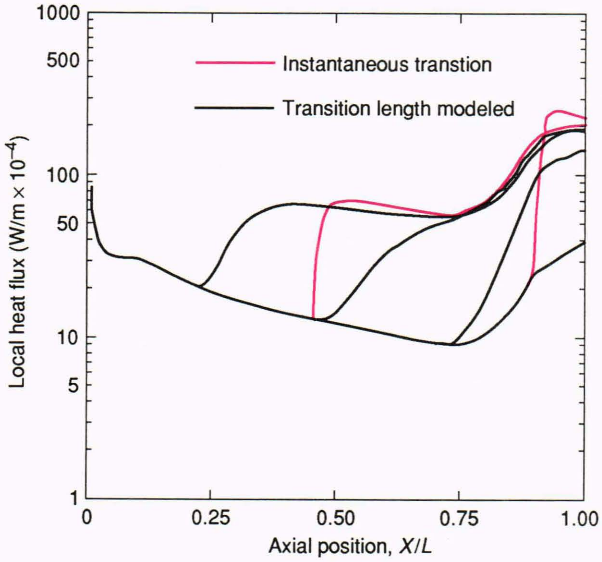


Figure 7. Local wall heat transfer rates for various transition criteria at a speed of Mach 25 and an altitude of 47.5 km.

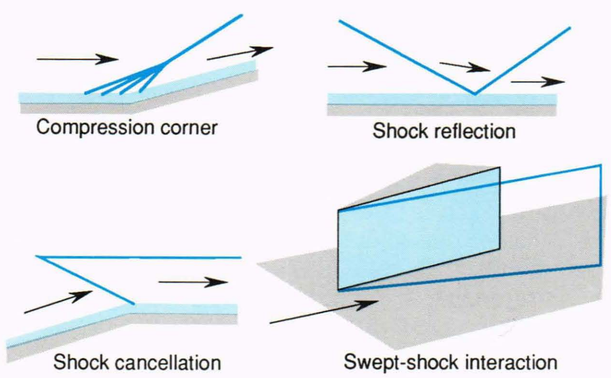


Figure 8. Possible type of shock-wave/boundary-layer interactions.

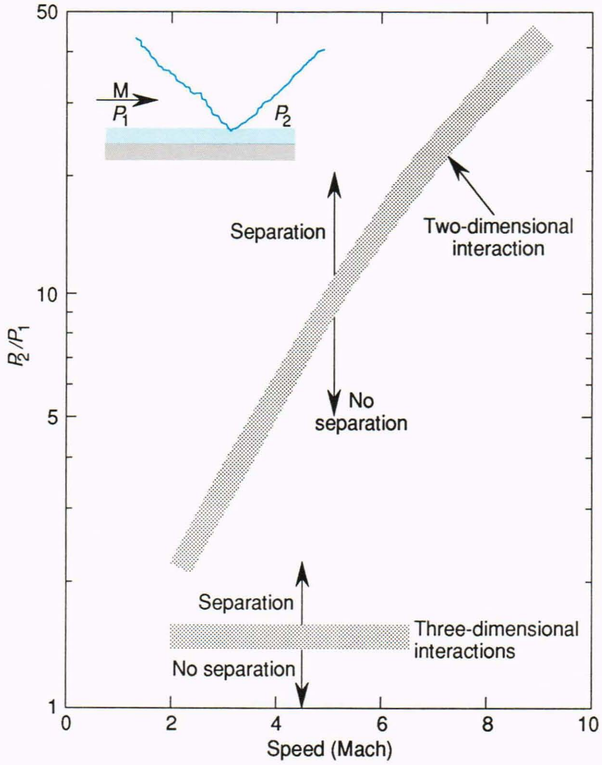


Figure 9. Pressure rise required to separate a boundary layer.

The goal of most inlet designs is to produce an inlet that does not result in any separated boundary layers. The designer must understand the magnitude of the pressure rises that will separate a boundary layer for various types of interactions, and then design the inlet so that this pressure ratio is never exceeded. As an example, the pressure ratios required to separate a boundary layer for two- and three-dimensional interactions are shown in Figure 9.[8,9] As can be seen from this figure, a two-dimensional shock wave can be much stronger before separating a hypersonic boundary than an equivalent three-dimensional shock. Because inlets are usually designed to produce flow structures very near the separation limits, extensions of these separation criteria to specific flow-field structures will be required for the design of hypersonic inlets.

The effect of an adverse pressure gradient on the development of a turbulent boundary layer is an additional area requiring further investigation. Computational tools are available for estimating the characteristics of the boundary layers in these regions, but large uncertainties exist in the heuristic turbulence model contained in these tools.[10] Experimental investigations of the development of hypersonic turbulent boundary layers in regions of adverse pressure gradients are required. The results of these experiments will be used to evaluate and improve turbulence models for hypersonic flows.

MEASUREMENT OF HYPERSONIC INLET PERFORMANCE

Simulation Requirements

Tests of a full-scale inlet model will not be possible before the first flight of the NASP, because of the large size of the aircraft and the enormous power levels that would be required to operate a facility. Before this flight test phase, much of the inlet development work must be accomplished with either computational tools or experiments with scale models that can be conducted in available ground test facilities. Because full duplication is not possible, only certain flow phenomena can be investigated in a given experiment. The simulation parameters that must be matched between the flight condition and the ground-based experiments depend on the flow-field features to be investigated (see Table 1).

Very little capability exists for simulating the high-temperature aspects of the hypersonic inlet flow fields in ground-based facilities because of either excessive energy requirements, temperature limits of materials, or pressure limitations. Because of these limitations, ground-based tests are used to investigate particular features of the flow field, such as shock/boundary-layer interactions, rather

Table 1. Aerodynamic simulation parameters.

Parameter requiring matching	Portion of flow field simulated
M_∞, γ	Inviscid flow field
M_∞, γ, Re, T_w/T_0	Perfect gas viscous flows
V, P_0, T_0, T_w, C_{O_2}, C_{N_2}	Real gas viscous flows

Table 2. Accuracies of inlet performance measurements required to maintain 1% uncertainty in engine I_{sp}.

	Mach 5	Mach 10	Mach 20
Air capture (%)	1–2	2–3	2–3
Heat loss (%)	≈100	30–50	10–20
η_{KE} (%)	1–2	0.2–0.5	≈0.05

than to provide full simulation. Existing facilities can match M_∞, γ, Re, and T_w/T_∞ for speeds up to about Mach 15, which allows investigation of many inviscid and viscous phenomena.

The extrapolation of measured phenomena to flight conditions can best be accomplished using computational tools. Some of these extrapolations may be large, so the pre-flight prediction of the high-speed performance of the NASP will rely heavily on computational tools. This heavy reliance on computational techniques increases the importance of experimental validation of the analysis techniques when applied to the basic phenomena. A comprehensive effort to validate all aspects of computational predictive capabilities is under way using tests that can be conducted in available facilities.

Required Accuracies

Despite difficulties in simulation in ground-based facilities, inlet tests can be used to study many aspects of the inlet flow field. As discussed earlier, measurements of the mass, momentum, and energy of the flow at the exit of the inlet are required to assess the true performance of an inlet. By investigating the sensitivity of the engine operation to the inlet performance, a measure of the required accuracy in performance measurements can be obtained. Using generic engine cycle calculations, an assessment was made of the required accuracy of inlet performance measurements, and the results are shown in Table 2 for speeds of Mach 5, 10, and 20. At each speed, the required accuracy in a given performance parameter is such that 1% uncertainty in engine specific impulse would result if all other parameters were known exactly. As can be seen in these results, the required accuracy in the measurement of the air capture ratio does not change appreciably over the speed range, but the required measurement accuracy of both the heat loss and η_{KE} becomes much more severe as the speed increases. This extreme sensitivity of the engine performance to the inlet operation at high speeds creates the need for the development of advanced instrumentation techniques that can achieve these required accuracies. Before discussing some of these advanced techniques, a brief description of the characteristics of available hypersonic facilities is in order.

Facilities

Three types of hypersonic flow facilities are available for inlet testing. The first type is a conventional continuous-flow wind tunnel where the air is pumped to high pressure, heated, ducted through the test section and over the model, cooled, and recirculated back through the pumps. Because of the large power requirements, these facilities are generally limited to speeds below Mach 10. These facilities present many challenges in the design of inlet models because the model must be actively cooled to provide the proper T_w/T_0 for boundary layer simulation. An example of this type of facility is Tunnel C of the Von Karman Facility at the Arnold Engineering Development Center.[11] This tunnel has a test section 1.27 m in diameter and runs continously at Mach 10 conditions at a total pressure of 130 atm and a total temperature at 1000 K.

The second class of hypersonic flow facility encompasses a range of different devices collectively labeled pulse facilities. The most widely used pulse facility is a shock tunnel, which uses high-pressure gas to drive a shock wave through a tube containing the test gas. The passage of the shock causes the gas to be compressed and heated. This high-pressure, high-temperature gas is then exhausted through a nozzle to provide the proper Mach number. Because these facilities use traveling waves to generate the flow conditions, the run time is typically on the order of several milliseconds. These facilities can generate high-speed, high-Reynolds-number conditions, but the short run times present several challenges in the measurement of inlet performance. Interestingly, the model design is usually simpler than models used in continuous flow facilities because the model temperature does not rise significantly over the course of a run, thus eliminating the need to actively cool the model. An example of this type of pulse facility is the 96-in. shock tube at Calspan, which can run at total pressures up to 1300 atm and stagnation temperatures up to 6000 K.[12] Simulation from Mach 10 to Mach 20 is achievable using a nozzle with an exit diameter of 1.22 m. Run times are typically less than 10 ms.

The third class of facility, which falls somewhere between the first two classes, is the blowdown facility wind tunnel. In these open-loop facilities, high-pressure air is initially stored in tanks and then ducted through a heater, nozzle, test section, and diffuser, and then exhausted to either the atmosphere or a vacuum sphere. These facilities generally have run times that range from less than one second to several minutes, depending on the operating conditions. The achievable test conditions in this type of facility are limited by either heating limitations or pressure containment limitations. An example of this type of facility is Tunnel 9 at the Naval Surface Warfare Center in White Oak, Md.[13] This tunnel operates at

Mach 10 and Mach 14 using nitrogen at total pressures up to 1900 atm and total temperatures up to 2200 K, with run times on the order of 0.5 s. The test section is 1.52 m in diameter.

Note that only pulse facilities can generate temperatures high enough to investigate real gas effects, but even those facilities cannot simulate both the proper temperature and Reynolds number at the same time.

Measurement Techniques

Various experimental techniques have been analyzed in an attempt to identify methods able to determine the inlet performance to a sufficient degree of accuracy. The measurement of the air ratio, heat loss, and kinetic energy efficiency will be addressed in the following sections.

AIR CAPTURE MEASUREMENTS

The measurement of air capture ratio in continuous flow and blowdown wind tunnels is straightforward, using available mass flow meters. In the use of these meters, which are attached to the aft end of an inlet, the airflow is diffused to subsonic conditions within a plenum before being discharged through a calibrated sonic orifice. If carefully used, mass flow meters can provide the required accuracies for air capture measurements in continuous flow and blowdown wind tunnels. Mass flow meters cannot be used in pulse facilities, because the required steady-flow condition through the mass flow meter can not be achieved during the short run time.

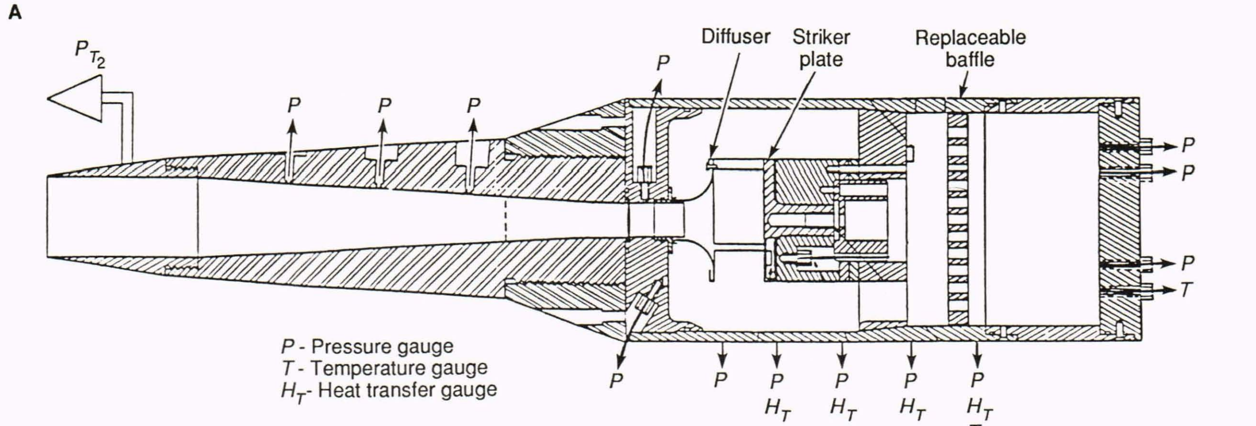
A mass flow measurement technique for pulse facilities that uses unsteady flow measurements has been under development at APL.[14] In this technique, a plenum of known volume is attached to the aft end of an inlet. For a short-duration run, as would occur in a pulse facility, the pressure in the plenum rises linearly as the plenum fills, and a measurement of this rate of pressure rise can be used to determine the mass flow through the inlet.

Initial investigations of this technique were performed using a Busemann inlet model with an attached plenum tested in the joint Ryerson/University of Toronto gun tunnel at Mach 8.3 and in the Calspan Corporation 48-in. shock tunnel. The model used in these tests is shown in Figure 10. During the gun tunnel test, various volumes were tested to assess the operation of the technique. As shown in the plenum pressure traces in Figure 11, the pressure rise is approximately linear over the course of the 10-ms steady portion of the run, and the rate of pressure rise is approximately proportional to the inverse of the plenum volume. For the conditions encountered in these initial tests, the rate of pressure rise was not exactly inversely proportional to the plenum volume, because the heat transfer to the plenum was significant. When the heat loss is included in the data analysis, the maximum error of the plenum filling technique is in the range of $\pm 4\%$. Further work will be required to refine the technique such that the required maximum error of 1% to 3% is achieved.

Heat Loss Measurement

A measurement of the heat loss from the captured airstream can be performed using either heat transfer gauges or an optical thermal mapping technique. In ei-

A



B

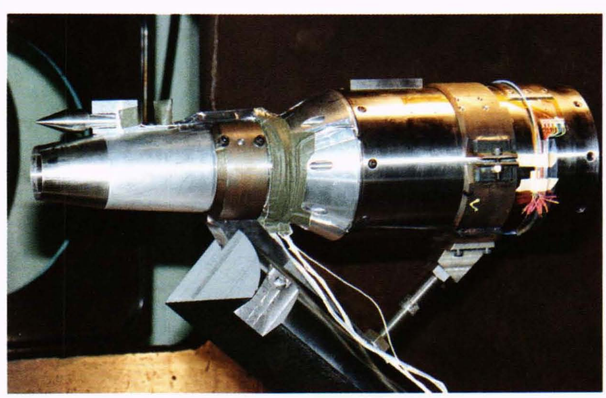


Figure 10. A Busemann inlet/plenum assembly for demonstrating the usefulness of using a filling plenum to measure mass flow. **A**. A schematic of the assembly. **B**. Photograph of the assembly installed in the Ryerson/University of Toronto Mach 8.3 gun tunnel.

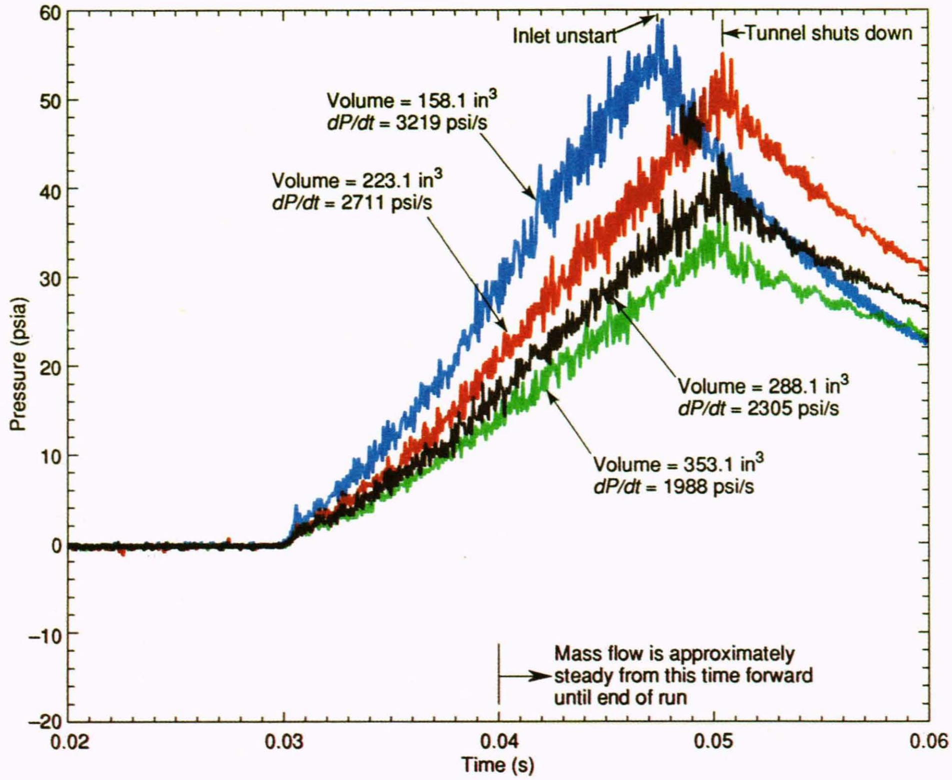


Figure 11. Plenum pressure traces, showing the effect of plenum volume on filling rate.

ther technique, a rate of change of the surface temperature is used to obtain a measurement of the local heat transfer, and the overall heat loss is obtained using an integration of the local measurements. In regions of high gradients, the resolution of the measurements becomes important in defining the overall heat transfer, and thermal mapping techniques offer an improved resolution capability. One limitation of the thermal mapping techniques is that the measurements on the inside portions of the model are usually not available. In general, the accuracy of heat loss measurements is adequate if the gradients in heat transfer can be resolved.

Kinetic Energy Efficiency Measurements

Many techniques are available for measuring inlet efficiency, but few can achieve the required accuracy at high speeds. One such technique that shows promise is the drag measurement technique illustrated in Figure 12. A measurement of the drag on the captured airstream can be combined with measurements of air capture and heat loss to produce an accurate measurement of the inlet efficiency.[15] In practice, it is nearly impossible to build an inlet model that measures the drag on the captured airstream directly. The drag on the captured airstream is caused by both additive forces (which are pressure forces exerted by the fluid not captured) and pressure and friction forces exerted by the inlet surfaces. If the additive drag is small relative to the total inlet drag, complementary theory can be used to estimate this portion of the drag on the captured airstream. The measurement of the drag exerted by the inlet model on the captured

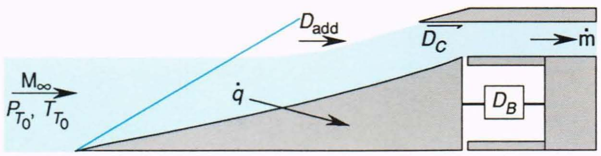


Figure 12. Schematic of drag measurement technique for measuring inlet efficiency.

stream requires an identification of the wetted surfaces. For simple inlet shapes, the identification of the wetted surfaces is trivial, but as the inlet shape becomes more complex, the uncertainty in the location of the wetted surfaces becomes large.

If the inlet surfaces wetted by the captured airstream can be identified, a measurement of the drag forces on these areas can be attempted. The simplest method for determining the drag is to measure the pressure and heat transfer distributions over the inlet surfaces. The pressure drag can be determined directly from an integration of the individual pressure measurements. The heat transfer gauges are used in combination with theory to estimate the skin friction. An integration of the estimated skin friction distribution allows an estimate to be made of the frictional drag. This technique for the measurement of drag assumes that all gradients can be resolved with the available instrumentation and that the heat transfer information can be related to the skin friction.

When the accuracy resulting from integration pressure and friction forces is not adequate, a force balance can be incorporated into the model design, as illustrated in Figure 12. A scramjet inlet model incorporating this measurement technique has been fabricated under the Generic Hypersonic Inlet Test Program sponsored by the Air Force Aero Propulsion and Power Laboratory. As seen in Figure 13, the forebody of the model is segmented such that the portion of the forebody wetted by the capture airstream is isolated from the remaining portions of the model. This center section of the forebody is attached to a force balance.

In the practical implementation of the forebody force technique, it is difficult to build a model where the inlet surfaces wetted by the captured airstream are the only surfaces attached to the balance. This results in a balance measurement that contains either more or less drag than desired. Pressure and heat transfer measurements in combination with theory are used to correct the balance measurements such that an assessment of the total drag of the captured airstream can be obtained. Because theory must be used in conjunction with measurements, an assessment of the accuracy of this technique is closely tied to a particular inlet design. For two-dimensional types of inlets such as the one shown in Figure 13, detailed analysis of the accuracy of this technique has been undertaken, and the technique should provide the required accuracy for flight speeds up to about Mach 15. For example, the uncertainty in measured inlet kinetic energy efficiency has been estimated to be no worse than 0.2% at Mach 10.

CONCLUSIONS

The development of an efficient inlet system that can operate over the entire flight regime is critical for the successful operation of the NASP. The design of an inlet presents many challenges because many different high-speed fluid dynamic issues must be addressed at some point in the design process. Most of these issues can be addressed either in experiments conducted in ground facilities or by using computational tools, but a relatively high level of uncertainty may exist in some areas prior to the first flight. At the present time, the design of an inlet for the NASP appears achievable, but detailed validation of predicted performance levels throughout the flight envelope is still required.

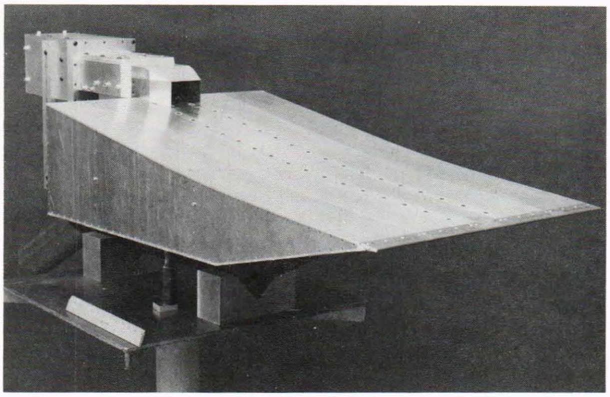


Figure 13. Scramjet inlet model built to test the forebody drag measurement technique.

REFERENCES

[1] Pandolfini, P. P., Billig, F. S., Corpening, G. P., and Corda, S., "Analyzing Hypersonic Engines Using the Ramjet Performance Analysis Code," *APL Tech. Rev.* (limited distribution) **2**(1) (in press).

[2] Waltrup, P. J., White, M. E., and Van Wie, D. M., "Engine Operation and Key Technical Issues in High-Speed Propulsion for the National AeroSpace Plane," *APL Tech. Rev.* (limited distribution) **2**(1) (in press).

[3] Bhutta, B. A., and Lewis, C. H., "Three-Dimensional Hypersonic Non-equilibrium Flows at Large Angles of Attack," AIAA Paper 88-2568, American Institute of Aeronautics and Astronautics, New York (1988).

[4] Fay, J. A., and Riddell, F. R., "Theory of Stagnation Point Heat Transfer in Dissociated Air," *J. Aeronaut. Sci.* **25**(2), 73–85 (1958).

[5] Holden, M. S., Wieting, A. R., Moselle, J. R., and Glass, C., "Studies of Aerothermal Loads Generated in Regions of Shock/Shock Interaction in Hypersonic Flows," AIAA Paper 88-0477, American Institute of Aeronautics and Astronautics, New York (1988).

[6] Berkowitz, A. M., Kyriss, C. L., and Martellucci, A., "Boundary Layer Transition Flight Test Observations," AIAA Paper 77-125, American Institute of Aeronautics and Astronautics, New York (1977).

[7] Dhawan, S., and Narasimha, R., "Some Properties of Boundary Layer Flow During Transition from Laminar to Turbulent Motion," *J. Fluid Mech.* **3** (1958).

[8] Korgegi, R. H., "Comparison of Shock Induced Two- and Three-Dimensional Incipient Turbulent Boundary Layer Separation," *AIAA J.* **13**(4), 534 (1975).

[9] Holden, M. S., "Shock Wave–Turbulent Boundary Layer Interaction in Hypersonic Flow," AIAA Paper 72-74, American Institute of Aeronautics and Astronautics, New York (1972).

[10] Gilreath, H. E., and Schetz, J. A., "Research on Turbulence in Hypersonic Engines," *APL Tech. Rev.* (limited distribution) **2**(1) (in press).

[11] *Test Facility Handbook,* Arnold Engineering Development Center, Tullahoma, Tenn. (1984).

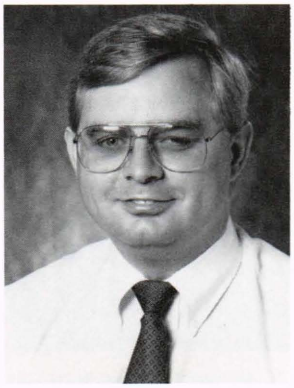
[12] *Hypersonic Shock Tunnel Description and Capabilities,* Arvin Calspan Corp., Buffalo, N.Y. (1987).

[13] Panaranda, F. E., and Freda, M. S., *Aeronautical Facility Catalogue: Wind Tunnels,* NASA RP-1132, National Aeronautics and Space Administration, Washington (1985).

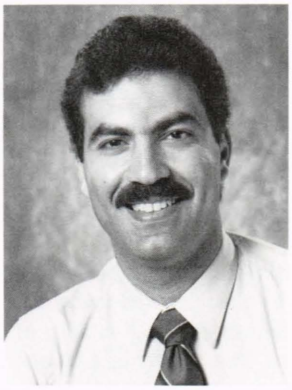
[14] Van Wie, D. M., Corpening, G. P., Mattes, L. A., Carpenter, D. A., Molder, S., and McGregor, R., "An Experimental Technique for the Measurement of Mass Flow of Scramjet Inlets Tested in Hypersonic Pulse Facilities," AIAA Paper 89-2331, American Institute of Aeronautics and Astronautics, New York (1989).

[15] Balent, R. L., and Stava, D. J., *Test Results of a Supersonic/Hypersonic Combustion Force Balance Inlet Model,* AFFDL-TR-66-69, Air Force Flight Dynamics Laboratory, Wright-Patterson AFB, Ohio (1966).

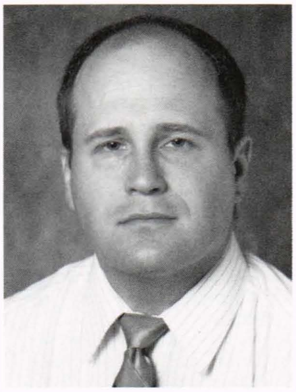
THE AUTHORS



DAVID M. VAN WIE received a Ph.D. in aerospace engineering from the University of Maryland in 1986. Since joining APL in 1983, he has worked in the Propulsion Group of the Aeronautics Department on the development of high speed ramjet and scramjet engines. He is currently the supervisor of the Preliminary Design and Analysis Section and is the Program Manager for the Generic Hypersonic Inlet Test Program.



MICHAEL E. WHITE received B.S. (1980) and M.S. (1981) degrees in aerospace engineering from the University of Maryland. He joined the Propulsion Group in APL's Aeronautics Department in 1981 and became a member of the Senior Professional Staff in 1986. He is now Deputy Program Manager for the National AeroSpace Plane Program and Supervisor of the Internal Aerodynamics Section. Mr. White's primary area of expertise is inlet development for high-speed propulsion.



GRIFFIN P. CORPENING received a B.S. and M.S. in civil engineering from the University of Wyoming in 1978 and 1984, respectively. He received a Ph.D. in aerospace engineering from the University of Maryland in 1988. He was employed as an engineering hydrologist at Western Water Consultants in Laramie, Wyoming, from 1981 to 1982 and was a civil engineering instructor at the University of Wyoming from 1982 to 1983. Since joining APL in 1987 he has worked in the Propulsion Group of the Aeronautics Department on the development of ramjet and scramjet engines. Included in this work has been the experimental investigation of inlet performance measuring techniques applicable to pulse facility environments, Project Engineer for the Rocketdyne Direct Technical Support Task, and analysis of various direct-connect and semi-free-jet scramjet combustor tests. He is a member of the American Institute of Aeronautics and Astronautics and serves on the Society of Automotive Engineers Hypersonics Propulsion Committee.