HYPersonic DRONE Design: 

A MULTIDISCIPLINARY EXPERIENCE

SCHOOL OF ENGINEERING AND APPLIED SCIENCE

JUNE 1988
UCLA'S Advanced Aeronautic Design group focused their efforts on design problems of an unmanned hypersonic vehicle. It is felt that a scaled hypersonic drone is necessary to bridge the gap between present theory on hypersonics and the future reality of the National Aerospace Plane (NASP) for two reasons: (1) to fulfill a need for experimental data in the hypersonic regime, and (2) to provide a testbed for the scramjet engine which is to be the primary mode of propulsion for the NASP.

The group concentrated on three areas of great concern to NASP design: propulsion, thermal management, and flight systems. Problem solving in these areas was directed towards design of the drone with the idea that the same design techniques could be applied to the NASP.

A seventy degree swept double delta wing configuration, developed in the 70's at NASA Langley, was chosen as the aerodynamic and geometric model for the drone. This vehicle would be air-launched from a B-1 at Mach 0.8 and 48,000 feet, rocket boosted by two internal engines to Mach 10 and 100,000 feet, and allowed to cruise under power of the scramjet engine until burnout. It would then return to base for an unpowered landing. Preliminary energy calculations based upon the flight requirements give the drone a gross launch weight of 134,000 lb. and an overall length of 85 feet.
PROPULSION GROUP

Group Members:
John Beltran
Susan Durlak
Lura Kern
Mark Lin
Peter Waschak
Jerry Yen

Faculty Advisor:
Professor R.E. Kelly
Table of Contents

1. Introduction ...................................................................................... 1
2. Hypersonic Cruise Vehicle (HCV) Configuration ...................... 3
3. Program Applications
   SCRAM ..................................................................................... 5
   STUB ........................................................................................ 6
   SEAGULL .................................................................................. 7
   EDDYBL .................................................................................... 8
   INLET ........................................................................................ 9
   CPIPE ........................................................................................ 9
4. Preliminary Design ....................................................................... 11
5. Final Design ............................................................................... 17
6. Conclusion ........................................................................................ 23

Figures ............................................................................................. 24

Appendices:
   A: Program Descriptions
      A1: SCRAM .............................................................................. 49
      A2: STUB ................................................................................ 57
      A3: SEAGULL .......................................................................... 73
      A4: EDDYBL .......................................................................... 76
      A5: INLET ................................................................................ 85
      A6: CPIPE ................................................................................ 94
   B: Hand Calculations .................................................................. 103

References ............................................................................................ 111
1. Introduction
1. Introduction

The efforts of the Advanced Aeronautics Propulsion Design group have been directed towards developing tools to analyze the effects of aircraft geometry and freestream conditions on scramjet performance. Specifically, the group has been concerned with the shock structure, boundary layer growth and inlet geometry of a Mach 10 drone research vehicle, and the impact of these parameters on the efficiency and thrust of the scramjets.

The tools used to analyze the flowfield have primarily consisted of pre-existing computer software. Six programs (STUB, INLET, CPIPE, SEAGULL, SCRAM, and EDDYBL) were used to evaluate different components of the drone's underbody and scramjet engines. SCRAM is a one-dimensional program for the estimation of scramjet performance of a hypersonic vehicle. SCRAM begins with the geometry of the vehicle and calculates gross estimates of thrust, efficiency, and other engine parameters. It is used by our group as a check for the more detailed analysis. SEAGULL calculates inviscid conical shock structures and freestream conditions for axisymmetric geometries. SEAGULL is used to calculate the shock structures over the drone forebody. EDDYBL is a two-dimensional/axisymmetric, compressible, turbulent boundary layer program, and is used in conjunction with SEAGULL to simulate the viscous flow conditions along the forebody. Using the freestream conditions from SEAGULL, EDDYBL is used to calculate the boundary layer growth along the vehicle. INLET uses an inviscid analysis to design hypersonic engine inlets. For given inlet conditions and desired combustor entrance conditions, INLET calculates the cowl geometry required. Data from SEAGULL and EDDYBL will be used to generate inlet conditions for use with INLET.
2. Hypersonic Cruise Vehicle (HCV) Configuration
Finally, CPIPE is a one-dimensional code for the analysis of scramjet combustor performance. CPIPE will use the output conditions from INLET to calculate the combustor efficiencies. An overall momentum balance on the engine should give the actual thrust of the scramjet engine configuration. The use of these codes allows a detailed analysis of scramjet engine performance from nose to tail of the vehicle.

An initial application of these codes was performed on a five-degree half-angle cone with two five-degree compression ramps leading into the inlet. Boundary layer and flow properties were determined for the forebody, and an inlet configuration was generated.

Analysis of this initial design revealed that greater forebody compression was required to produce an acceptable inlet design. Therefore, the design was altered by placing one additional five and two additional three-degree ramps between the nose of the vehicle and the two five-degree ramps leading into the inlet. The location of the three-degree ramps were varied to maximize static pressure gains. The final design provided sufficient thrust to maintain level flight at design condition.
2. Hypersonic Cruise Vehicle (HCV) Configuration

It was determined early in Winter Quarter 1988 that the Advanced Aeronautics Design Group (AAD) needed a unifying concept for the design of the hypersonic drone. This was accomplished by choosing a configuration that had been researched extensively at NASA Langley Research Center in the late 1970's. The advantage of a single tested configuration for the entire class is clear: each group would base their designs on the same vehicle, with the same restrictions applying to all three groups, thereby forcing greater interaction among the groups.

The configuration chosen by the AAD students was a 70 degree swept delta wing design conceived by NASA Langley engineers. It was a hypersonic research airplane concept, and has been dubbed the Hypersonic Cruise Vehicle (HCV) by the AAD students. Figure 1 shows the HCV as it was originally conceived. The HCV was designed to be air-launched from a B-52 at Mach 0.8 and rocket-accelerated up to a cruise speed of Mach 7 by four liquid hydrogen rocket motors mounted in the base. At this point, the scramjets would be ignited and the vehicle would cruise until burn-out, at which point it would return to base for an unpowered landing. A detailed description of the HCV's configuration is given in Figure 2. In developing the HCV design, a wind tunnel model was built, and performance characteristics of the model were collected and tabulated in Reference 1. The geometric characteristics of the model are tabulated in Table 1. The scaling for the model was 0.021 of the size conceived for the actual HCV.

The first step in adapting the HCV for the purpose of a hypersonic drone was to size it according to the needs of the class. This included calculating the amount of fuel required to go from the air-launch speed to cruise speed. From that, size and weight necessary for the fuel requirements were determined. In order to do this, it became necessary to determine a means of air-launch. The B-52 has size and weight constraints consisting of a 24-ft. wing span limit, a 9-foot fuselage
height limit, and a gross weight restriction of 70,000 pounds. This limits the size of the drone considerably and, in order to reach Mach 10, it was found that the weight limit was exceeded. Alternatives were then examined. The 747 and B-1 aircraft were determined to be viable options for the air-launch.

Therefore, the HCV is a basis upon which each group can build their design. It gives the Flight Systems group the geometry and spacing of the vehicle so that they can determine which sensors and how many are required in certain places. This would result in a more detailed and more exact design. The Thermal Management group can determine critical areas on the HCV that would require cooling. They can change the design within certain reasonable limits to satisfy their cooling system design. A definite configuration for the drone gives the Propulsion group a starting point on the design of the underbelly for the vehicle; that is, a first iteration could be performed from which they refined their design to achieve an optimum configuration.
3. Program Applications
3. Program Applications

The group explored pre-existing computer programs, as well as devising others as needed, as a method of analyzing the merits of various designs because of the time savings afforded and the ability to calculate complex flowfields that are inherent to hypersonic design. Figure 3 shows the areas of application of each of the codes in analyzing the drone. A brief description of each program follows, while a more detailed description of the workings of each code may be found in Appendix A.

SCRAM

The program SCRAM is a one-dimensional airframe-integrated scramjet simulation program. It is currently being used by NASA on the National Aerospace Plane (NASP) project as a first iteration on scramjet performance calculations. It is available to the AAD (Advanced Aeronautics Design) students on IBM PC computers and has already been run successfully using the HCV geometry as inputs (see Appendix A1 for a description of the program and a sample case).

The inputs to SCRAM include freestream conditions, scramjet geometry, and other parameters necessary for calculating the flowfield properties such as kinetic energy efficiency, which approximates momentum losses due to shocks, and temperatures along the surfaces. SCRAM calculates the flowfield characteristics for five stages of the scramjet: freestream, forebody, inlet, combustor, and nozzle. It outputs the flow conditions at the end of each stage, as well as boundary layer thicknesses. It then calculates the engine cycle performance parameters and outputs them in an overall summary.

SCRAM was used in two ways for the Hypersonic Drone concept. First, it was used to help determine the sizing of the engine. By developing a scaled drone instead of a full-sized, piloted hypersonic vehicle, there
was a concern that the scramjet engine will not be large enough to produce sufficient thrust. SCRAM can be run at the smallest size limit of the drone to determine feasibility of the design. Since SCRAM generates values that are optimistic for the engine performance, it offers a means of evaluating relative worth of designs. That is, if SCRAM's output shows that a design is not feasible, then the design will be certain not to work, and further study into that configuration would probably not be worthwhile.

STUB

The flow behind the detached bow shock structure generated by a hypersonic blunt body contains regions of both subsonic and supersonic flow. Since SEAGULL is restricted to the case of flow that is purely supersonic after a shock, there arises a need for an analysis of the shock structure and flow properties at the nose of the hypersonic drone studied by the Advanced Aeronautics Design class. The shock structure and flow properties then become inputs for the SEAGULL and EDDYBL programs.

STUB is a written program that utilizes the time-dependent technique to generate the detached bow shock structure off a hypersonic blunt body. Details of the method used by the program can be found in References 2, 3, and 4, and the governing equations in Appendix A2. The program is also listed in Appendix A2.

Briefly, the time-dependent technique employs a numerical calculation of flow properties through time steps, using the unsteady Navier-Stokes equations as the governing equations. Although the introduction of time appears to be an added complication for a steady-state flow, it becomes a necessary simplification to the problem of hypersonic blunt bodies. There currently exists no uniformly valid, steady state technique that can handle the mixed supersonic and subsonic
region behind the bow shock (Ref. 2).

Although STUB was written to analyze two-dimensional blunt bodies, an extension to axisymmetric or three-dimensional asymmetric blunt bodies could be made in the same manner as References 4 and 5. The two-dimensional nature of this analysis, however, does mean that relieving effects and other three-dimensional, axisymmetric effects for a spherically-tipped nose (as should be modelled for the hypersonic drone) are neglected. This then became a severe limitation of the program.

SEAGULL

SEAGULL is a code designed for the analysis of a two-dimensional or axisymmetric supersonic inviscid flow of an ideal gas. For the purposes of the Advanced Aeronautics Propulsion Design Group, SEAGULL was used to model the external compression along the forebody of a scramjet engine. SEAGULL provides only an inviscid solution for the forebody flowfield and shock structure, and thus the boundary layer code EDDYBL was used with SEAGULL to iteratively solve for a viscous flowfield solution.

The resulting flowfield properties calculated using SEAGULL (and EDDYBL) were used as inputs for the program INLET. One limitation of SEAGULL is that it models both an upper and lower wall; thus, the flow properties that SEAGULL outputs are for internal compression. This causes the flow on the body downstream of the shock to slow, whereas for purely external flow, it would remain constant. The slowing effect is minimized, however, by placing the walls far apart.
EDDYBL is a fully compressible, two-dimensional or axisymmetric computer program for the calculation of boundary layer properties. The program accounts for mass flux at the body surface, heat flux, pressure and temperature gradients, and both transverse and longitudinal body curvature. The program does not, however, account for the effects of shock-boundary layer interaction or separated flow. EDDYBL is currently a fully operational program and has been run for supersonic cases. For a more complete description of the program EDDYBL and sample output parameters, see Appendix A'4.

For the purposes of the Advanced Aeronautics Propulsion Design group, EDDYBL was used in conjunction with the program SEAGULL, which performed an inviscid analysis on the forebody of the scramjet. Provided with upstream conditions and geometry from SEAGULL, EDDYBL output the boundary layer characteristics of the flow field. The boundary layer thickness was used to redefine the geometry, thus creating a new inviscid input geometry for SEAGULL. This will alter the shock structure calculated for the forebody, and thus will alter the upstream conditions used for EDDYBL. This iterative procedure was used until the flowfield properties converged on a viscous solution.

EDDYBL was helpful in determining the overall performance of a given scramjet configuration by calculating the boundary layer growth at the inlet entrance (or cowl lip), and thus determining the effective mass flow entering the engine (Fig. 4). Overall, using EDDYBL in conjunction with the inviscid codes SEAGULL and INLET will provide a more realistic evaluation of the performance of a given scramjet configuration.
INLET

INLET was the code used in designing the internal cowl and centerbody geometries of the scramjet. It is a two dimensional inviscid code that uses the method of characteristics to determine the internal geometry of the inlet (see Appendix A5 for a description of the program and sample inputs and outputs). Given the conditions at the entrance to the inlet and the desired combustor entrance conditions, INLET calculates the cowl and centerbody geometries required to minimize total pressure loss across the inlet. Although the distinction between the forebody and inlet can be nebulous, the propulsion group decided to use SEAGULL mainly for external compression and INLET for internal compression. Thus, the main benefit of using INLET is that it determines the cowl and centerbody geometries for the scramjet engine. However, it was necessary to meet the desired combustor entrance requirements, as well as the upstream entrance conditions (acquired from SEAGULL). Therefore, it was necessary to iteratively use INLET and CPIPE (combustor code) to determine whether INLET could provide reasonable cowl and centerbody geometries for the combustor requirements.

CPIPE

CPIPE is a one dimensional real gas analysis of the combustion of hydrogen in air in a supersonic channel. Given the initial upstream conditions and geometry definition, CPIPE outputs the downstream one dimensional flow properties (see Appendix A6 for a description of the program and required inputs).

CPIPE was used to model the combustor region of the scramjet engine. CPIPE is a relatively simple analysis of the combustor since it uses only one dimensional conservation equations to calculate the
downstream conditions. However, more complicated analyses of the supersonic combustor are rare and tend to concentrate on the flow structure in specific areas of the combustor (i.e. directly behind a flameholding structure or fuel injection area). In addition, CPIPE has several useful features such as (1) possibilities for several fuel injection points at a variety of angles, (2) estimates of ignition and flameholding likelihood, and (3) estimates of heat flux, given temperature distribution. Thus, CPIPE served the purpose of scramjet performance analysis without being unnecessarily complex.

Since CPIPE is one dimensional it was necessary to average the flowfield values obtained from INLET in order to input the appropriate values in CPIPE.

References 10, 11, and 12, which contain temperature and pressure distributions for various combustor designs that were tested in Langley's supersonic combustor test facility, could be used to validate CPIPE. This validation was not completed due to time constraints; however, CPIPE has been used satisfactorily at NASA Langley to simulate scramjet combustor designs.
4. Preliminary Design
SCRAM was used as a first iteration on the scramjet design for the drone. It was primarily used to determine whether an engine small enough to fit onto the HCV would produce enough thrust to overcome the total aircraft drag. One of the primary weaknesses of SCRAM, however, is its dependance on wise choices of abstract efficiency parameters. Such parameters define the overall kinetic energy efficiency of the design, the pressure loss coefficients across the various shocks and through the combustors, and have a large impact upon the overall thrust delivered.

When SCRAM was run with the HCV's geometry and flight condition as input, it was found that the scramjet could supply the required thrust. With a nozzle efficiency of 1.0, SCRAM predicted a total thrust force of 43,000 pounds. This value was as low as 30,000 pounds with a nozzle efficiency of 0.96. Using information given from wind tunnel tests on the HCV configuration, [13], the total drag at Mach 10 and 100,000 feet was found to be 26,600 pounds, for an excess thrust of 3,400 pounds with 96% nozzle efficiency.

The vehicle nose was modelled as a two-dimensional, five degree ramp, with a rounded, one-inch radius nosetip. Initial guesses for the shock structure and flow properties were inputted into STUB, and final converged values were computed. These values for the shock structure and flow properties were made available too late for use, but would have served as inputs for SEAGULL and EDDYBL. As it was, the vehicle was modeled as having a sharp nose, compared to the one-inch radius nose of the actual vehicle. The next step, then, would have been to proceed through one iteration using STUB outputs.

The group began to analyze the merits of various forebody geometries. For the following forebody geometries presented, two opposing constraints were considered. Static compression of the flow by the forebody had to be attained with maximum efficiency and minimal total pressure loss. At the
same time, the flow had to be slowed sufficiently prior to the inlet so that the cowl would be made as short as possible.

The first geometry analyzed was a simple five degree cone and also acted as verification for SEAGULL. (Appendix A3). The cone did not compress or slow the flow sufficiently and required a cowl length of 150 feet to compress the flow from Mach 9 ahead of the inlet ramps to the maximum combustor speed of Mach 4.

The second geometry consisted of two ramps in addition to the basic five degree cone. (Fig. 5). The first ramp is five degrees and is located fifteen feet from the nose. A second, three degree ramp is located twenty-four feet from the nose for a total turning angle of thirteen degrees. This design resulted in a total pressure loss of 22%, and a static pressure increase of 8.97 before the inlet ramps. The Mach at the entrance to the inlet was 7.51. (Table 2).

The third geometry moved the ramps of the second design three feet further from the nose (Fig 6). This resulted in a decrease of total pressure loss at the inlet and an increase in static pressure. (Table 2). These changes improved the performance of the forebody as a compression surface. The last geometry added another three degree ramp thirty-one feet from the nose. This resulted in a total pressure loss of 23%, but a static pressure increase of 14.6. (Table 1). This geometry was the one chosen for the final design. (Fig 7).

Boundary layer properties were computed for a variety of geometries in the early stages of the design. The simplest case run was for the five degree cone which was modeled as having a sharp nose, unlike the rounded nose of the drone. This approximation introduces error in the downstream flowfield, but was necessary because of a lack of freestream data in that region. The surface roughness of the vehicle was set to .001 in., the default value of EDDYBL, which is a very smooth surface. Lacking any data for high
altitude atmospheric turbulence, the freestream turbulence intensity was set to 2\%, which is what could be expected from a well designed wind tunnel [14].

The results of the computations are presented in Figures 8 and 9. Note first the "blips" that occur for the boundary layer thickness. These are points where the program converged on an incorrect thickness, but with the next step corrected itself and converged on the correct value. Communication with the author of EDDYBL indicates that these points are stations that have difficulty converging due to the grid representation of the flowfield which result in numerical errors. The automatic addition of grid points by the program can cause EDDYBL not to converge for that station, but generally converges to the correct solution with the next step.

Note also the trend of the lower left corners of the curves to have a slight lip. This is also caused by convergence errors of the program. EDDYBL calculates the stagnation properties from input free stream conditions and uses these initial starting conditions to predict the boundary layer properties at the next step. If that prediction is incorrect and the program does not converge after twenty iterations, the program will proceed to the next station. To say that the program does not converge does not mean that the calculations diverged, merely that the solution did not converge fast enough. After several stations, however, the program begins to make correct predictions and the solution converges. Extrapolation of the curves, shown by dotted lines in Figures 8 and 9 shows that the computed boundary layer values are close to zero at the stagnation point, as they should be.

The shape of the curves in Figure 9 suggests that transition does not occur and that the flow is entirely laminar. Literature shows that transition on 10 degree cones occurs at an arc-length Reynolds number of approximately ten million, while EDDYBL shows that transition has not yet
occurred at Reynolds numbers of 48 million.[15, pp 39, 59,61]. The data presented in the literature, however, is for cones with roughness heights of .015 in., and unknown freestream turbulence intensity, both of which have an impact on transition to turbulence and are different than those used for this case.

Figure 10 shows the heat flux at the surface of the vehicle as a function of axial distance, with the surface maintained at 1000 degrees Rankine. (A misunderstanding occurred between the thermal management and propulsion groups as to the temperature that the wall would be maintained at. The thermal management group specified 1000 degrees Kelvin, which is higher than the 1000 degrees Rankine used here. The heat flux could thus be expected to be somewhat lower than that indicated.) The portion of the curve to the left of point A of Figure 10 is incorrect, suffering from the same convergence problems near the stagnation point as the boundary layer properties of Figures 8 and 9. The heating rates shown have been determined to be acceptable by the thermal management group. Active cooling would be required to maintain the skin temperature but would be within the capabilities of the cooling system.

Figures 11 through 14 show the effect of boundary layer suction at the wall. The mass removal began 31 feet from the nose at a rate of .0005 slugs/ft^2*s. (Figure 13). Figures 11 and 12 show the effects of the mass removal on boundary layer thickness. The impact is seen to be primarily on displacement, and is caused because the fluid that is being removed is the slowest and has the largest impact on displacement thickness. (Recall that displacement thickness is a measure of the mass flow rate along the body that is lost due to the slowing of the flow in the boundary layer.)

The mass removal has a profound effect on heat flux. Figure 14 shows that, although the suction has a small effect on boundary layer thickness, the heating rate at the wall was increased to a value comparable to the heating rates experienced in the first few feet of the forebody. Increased
boundary layer suction might therefore lead to unacceptably high heating rates, which would put an upper limit on the amount of boundary layer control that could be obtained through suction.

Finally, several hand-calculations were performed to estimate the total mass removed. (Appendix B). The results indicate that, for the five degree cone, the mass removal rate per unit width over the last five feet of the forebody directly before the inlet ramps would be .00425 slugs/sec*ft. Assuming a depth of 7 feet, (the width of the forebody at the inlet ramps), the total mass flow rate would be .0298 slugs/sec. The average velocity of the removed mass would be 57.2 ft/sec, thus removing 400 ft^3/sec over the 35 square foot area, which would be a large volume of air to dispose of.

Another calculation was made to determine the removal rate that would be required to remove the same amount of boundary layer as displacement thickness over 7 square feet of the forebody. The required mass flow rate would be .208 slugs/sec, with an average velocity of 400 ft/sec, and a total volume of 2795 ft^3/sec. Such a high mass removal rate would be unacceptably large.

At the outset of the Spring quarter, it was still the intention of the Propulsion group to use INLET as an axisymmetric code when designing and analyzing the scramjet inlet geometry. However, an error in the code prevented the group from utilizing INLET's axisymmetric feature. This turned out to benefit the group since the actual nature of each scramjet module is more closely represented by a two-dimensional geometry because of its location far aft of the nose (see Figure 2). Once the attention of the group was turned to a two-dimensional analysis of the scramjet modules, a test case was run using an inlet Mach number of 2.5, two ramps of five degrees each, and an exit Mach number of 1.3. The output of this result was the same as that calculated using the shock relations
5. Final Design
from Reference [2]. With the program running correctly, the first inlet geometry case was run.

The first inlet geometry case was actually a dual case. Since problems were experienced running other computer codes, it was thought that the flow could only be slowed to Mach 9.3 at the starting point where INLET calculations would begin. Hence, INLET was run for two cases: a Mach 9.3 to Mach 3 case and a Mach 9.3 to Mach 4 case. The results of these cases are shown in Figures 15 to 22. The output generated was not favorable. Inlet plane area to exit plane area ratios were 0.075 for the Mach 9.3 to Mach 3 case and 0.173 for the Mach 9.3 to Mach 4 case. This means that for an inlet area of 2 feet, the outlet area would be 1.8 inches for the Mach 3 exit condition and 4.14 inches for the Mach 4 exit condition. These small areas would not allow enough flow through the combustor to produce the thrust required to propel the drone at Mach 10 for 5 minutes. Not only was the area for these cases a problem, but the total pressure losses were enormous due to normal shock waves within the inlet. The total pressure recovery for the Mach 9.3 to 4 case was 0.09 and 0.02 for the Mach 9.3 to Mach 3 case mainly because of the normal shock wave within the inlet. From these results, it was apparent that the free stream Mach number needed to be reduced to obtain a reasonable scramjet engine design.

The solution to the problem was to recompute upstream Mach numbers. After correcting an operational error with SEAGULL, a free stream Mach number of 7.0 could be obtained in order to use as input to INLET. The final inlet geometry was generated for a Mach 7 to Mach 4 compression. The results are presented in the design section of this report.
The forebody geometry consists of three ramps: one five degree and two three degree ramps, as previously mentioned and shown in Figure 7. This resulted in 23% loss of total pressure but an increase of 14.23 in static pressure. Furthermore, the shocks lie sufficiently close to the body so that the inlet capture area is 7.1 times the width of the cowl inlet, and there is no spillage for on-design conditions. (Figure 23).

Difficulties encountered in EDDYBL resulted from the lack of shock-boundary layer modeling in the program. (Figures 24 and 25). The problem is that the boundary layer properties are discontinuous across the shock and it is difficult to determine the correct starting conditions to compute properties downstream of the shock. As seen from Figures 25 and 26, the arc-length Reynolds number increases discontinuously across the shock, despite a discontinuous decrease in freestream Mach number. However, at the second shock, the Reynolds number decreases with a decrease in Mach number, as it should. The result of these variations is reflected in the boundary layer properties. The trend in compressible boundary layers is for increasing boundary layer thickness with increasing Mach number. [2, p. 538]. This agrees with the first discontinuity of Figure 24, but not with the second. Furthermore, shock-boundary layer interaction is not modeled in EDDYBL. Although the computed flow is laminar, experiments have shown, very generally speaking, that turbulent boundary layers at Mach 7 experience a 10% increase in thickness across a shock. [16, p. 7501-A-1]. Note that the computed boundary layer decreased in thickness across the shock, showing that EDDYBL is not applicable to shock-boundary layer interactions. (Figure 24).

With the program unable to predict boundary layer properties across shocks, quantitative analysis of the design is impossible. Qualitatively,
however, a number of results may be postulated based upon the results of the single five degree cone case.

Knowing that the boundary layer is .15 ft thick at the first ramp location downstream of the nose, boundary layer control will probably be required for efficient engine operation. In addition, the boundary layer will probably undergo transition prior to entering the inlet, due to traversing three shocks and increasing pressure. Forestalling transition, either by boundary layer diversion or suction will therefore be a necessity due to both heating and engine performance considerations.

Control of the boundary layer presents a major problem. Boundary layer diversion, as opposed to boundary layer suction, was discarded due to the high cooling expected to be required on the sharp edge of the splitter plate. Experience with the five degree cone, however, indicates that large cooling rates will be required for boundary layer suction over a very large area due to the large removal rates required. This may make splitter plates more practical than boundary layer diversion because of it's simplicity. Both methods of control would require high cooling rates, as well as a large rate and volume of fluid diverted or ingested. The splitter plate would accomplish this task without pumps and other active removal devices required in boundary layer suction.

The final configuration for the inlet geometry of the scramjet was obtained by running the program INLET for a two-dimensional case by specifying a Mach number of 7.0 for the "free-stream" velocity and a Mach number if 4.0 for the exit velocity, "free-stream" meaning the flow conditions after the initial ramp compressions on the forward underbody of the drone but before the two five degree ramps on which INLET constructs the flow field. Three internal isentropic compression sections were used and the output from INLET is summarized in Figure 27 thru 32.

Figure 27, Inlet Area vs. Horizontal Distance, shows the inlet cross-sectional area measured axially from the cowl lip position. If the graph is scaled by setting the initial area equal to 1.07 square feet,
assuming a 1.0 foot depth because it is a two-dimensional case, then 0.36
on the nondimensional scale is 1.07 feet on the actual drone. This 1.07 feet
dimension is the approximate inlet length shown in Figure 2. This scaling
will allow the discussion of the results to be related to the actual
dimensions of the drone. Using the scale 0.36 = 1.07 feet, it can be seen in
Figure 27 that the inlet area contracts rapidly from 1.07 feet (0.36) to
0.456 feet (0.154) in a distance of 3.2 feet (1.08). This sharp decrease in
area then tapers off and the final inlet area is 0.394 feet (0.133) which
occurs 20.0 feet (6.75) aft of the initial spike tip. Note that the area is
approximately 0.394 square feet (0.133) from a distance of 19.1 (6.44) to
20.0 (6.75) feet. This 0.9 feet is needed to ensure uniform properties
across the exit plane give an area contraction of 2.7 times, meaning that
the area at the entrance plane is 2.7 times greater than the exit plane area.

The relative locations of the five degree ramps and the cowl lip
can be seen in Figure 28. Again, using a scale of 0.36 = 1.07 feet it can be
seen that the intersections of the two five degree ramps is at 6.87 feet
(2.32) behind the spike tip of the first ramp and the cowl lip is located
14.2 feet (4.78) aft of the spike tip of the first ramp. The intersection of
the two five degree ramps was calculated by INLET so that the shock
waves generated by the two ramps intersect at the cowl lip. Placing both
shock waves at the cowl lip minimizes flow spillage and increases engine
efficiency. Figure 28 also shows that the minimum cross-sectional area of
the inlet occurs at 18.4 feet (6.2) along the axis of the drone and remains
constant up to the exit plane at 10.0 feet (6.75).

Mach number and pressure distributions on the cowl contour are
shown in Figures 29 and 30, respectively. The magnitude of the Mach
number can be traced in the output from INLET from the spike tip of the
first five degree ramp to the exit plane of the inlet. The "free-stream"
Mach number is 7.0. Behind the shock wave generated from the first five
degree ramp the Mach number is 6.106. The Mach number after the second

19
shock wave originating at the intersection of the two five degree ramps is 5.401. These results can be calculated using the Prandtl-Meyer oblique shock relations found in reference [2]. The shock wave originating from the cowl lip further reduces the Mach number to 4.267. This Mach number remains constant because INLET assumes that the cowl contour is a straight wall for the first 1.73 feet (0.583) for the geometry calculated. At 15.88 feet (5.359), the cowl contours change from being a straight wall to being an isentropic compression surface and assumes an appropriately curved shape. The flow is compressed from a Mach number of 4.267 at 15.88 feet (5.359) to a Mach number of 4.00 at 17.63 feet (5.950). From Figure 30 it can be seen that the pressure increases as Mach number decreases. The static pressure ratio increases from 14.9 at the entrance of the inlet to 21.2 at the exit plane of the inlet which means that approximately one-third of the compression is taking place internally.

An extremely important feature of all supersonic flow engines is the total pressure loss, which dictates the efficiency of the engine. High total pressure loss indicates a low engine efficiency while a low total pressure loss indicates high engine efficiency. The total pressure ratio along the centerbody contour is plotted in Figure 31 and the total pressure ratio along the cowl contour is plotted in Figure 32. It can be seen that the total pressure ratio along the centerbody contour behind the first shock drops to 95% of its initial value behind the second shock. Past the second shock, the flow along the centerbody contour encounters no other shock waves because the flow is compressed isentropically, which means no further total pressure loss occurs. Behind the oblique shock wave at the cowl lip the total pressure is only 77% of its "free-stream" value. Because the cowl lip shock wave is cancelled at the centerbody by INLET, there are no internal shock reflections. Since the flow is compressed isentropically internally, the total pressure along the cowl surface is constant. Since the
The configuration used for the combustor is illustrated in Figure 33. It is a two-dimensional box combustor with an entrance height of 5 inches, and a constant width of 11.2 inches (the combustor length is 10.0 feet). There are two fuel injection points, at 3.75 ft. and 5.0 ft. respectively in the axial direction. Information regarding the injectors is given in Table 3. The temperature distribution as a function of axial distance is given in Figure 34. There is a steady rise in temperature until the location of the first injector, after which it levels off and even begins to decrease. This corresponds to a leveling off of the combustor efficiency (see Figure 35) at a value near 0.95 after the first injector. This gradual rise in combustion efficiency starting at the entrance to the combustor is due to the manner in which CPIPE "distributes" the reacted fuel equivalence ratio over the combustor. Thus, by the time the flow reaches the first injector, a large fraction of the fuel injected at that point has started combustion. After the first injector, there is a constant level of reaction. This explains the drop in static pressure (Figure 36) after the first fuel injection point. The pressure due to the combustion process is constant while the area of the combustor continues to increase. The Mach number displays a similar behavior (Figure 37), stabilizing at a constant value after the first injector. The exit Mach from the combustor is approximately 2.3.

With the combustor exit properties analyzed by the program CPIPE, the flow field inside the nozzle was modeled using the method of characteristics. Reference [2] provides the algorithm used to implement the computer code. In applying the method of characteristics, the flow...
field is assumed to be irrotational, inviscid and adiabatic. In addition, the characteristic lines are determined to a first order approximation.

Once the nozzle exit velocity was determined to be 11,000 ft/sec, a momentum balance was performed to determine the thrust. The control volume used for the momentum balance was defined by the equivalent inlet capture area under the nose of the HCV and the vertical nozzle exit plane. There was no spillage for the on-design condition; hence, it was not included in the momentum balance.

The thrust available with the nozzle design illustrated in Figure 28 was calculated to be 7646 lbs. per unit span, which is approximately 8.99% of the thrust available in isentropic expansion. Enthalpy losses due to heat flux and viscous effects at the upper nozzle wall, as well as plume drag, which was not accounted for, are expected to further reduce the efficiency of the nozzle although there should still be adequate thrust.

Note that a certain errors are propagated in locating the mesh points with the first order approximation. Reference 17 provides the thrust coefficient measured on a similar model tested at Mach 10. The thrust coefficient for this design is calculated to be 0.36, which falls between 0.1 and 0.92, the range of thrust coefficients presented in reference 2.
6. Conclusion
Various forebody geometries were evaluated for their ability to efficiently compress free stream flow prior to ingestion by scramjet inlets. Also, boundary layer growth on these forebodies was investigated, and boundary layer control, in the form of surface suction, was found to be somewhat effective. Inlet geometry design was investigated, and it was found that inlet speeds greater than Mach 7 would not allow sufficient capture area for the compression required.

A final design for the forebody consisted of a five-degree half-angle cone with a five degree compression ramp placed 18 feet aft of the nose, and two additional compression ramps placed at 27 and 31 feet. The Mach number entering the inlet was 7.2, and an inlet geometry was designed using two pre-inlet five degree compression ramps. The cowl geometry was generated to yield an isentropic compression to Mach 4 at the entrance to the combustor. The total length of the two pre-inlet ramps and cowl was twenty feet.

Although EDDYBL was not able to compute shock-boundary layer interaction and was therefore unable to calculate flow properties accurately at the shocks generated at the three degree ramps, results from the single five degree half-cone were used to obtain trends in boundary layer growth on the forebody. Boundary layer suction was found to require high suction velocities and large volumes of fluid ingested, accompanied by severe increases in surface heating.

CPIPE was used to compute combustor performance of the combustor, and the method of characteristics was used to roughly compute the expansion in the nozzle. Finally, a momentum balance was performed and the thrust generated by the design was calculated to be 58,000 lb, which was slightly higher than the values output by SCRAM, and was sufficient to propel the drone at a Mach 10 cruise speed.

6 Conclusion
FIGURES

AND

TABLES
FIGURE 1: HCV Configuration

FIGURE 2: Model general dimensions. All dimensions have been normalized by the body length (f = 50.8 cm).
### TABLE I. GEOMETRIC CHARACTERISTICS OF MODEL

<table>
<thead>
<tr>
<th>Wing</th>
<th>Area, reference (includes fuselage intercept), m² (in²)</th>
<th>0.043 (67.200)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Area, exposed, m² (in²)</td>
<td>0.023 (36.121)</td>
</tr>
<tr>
<td></td>
<td>Area, wetted, m² (in²)</td>
<td>0.047 (72.242)</td>
</tr>
<tr>
<td></td>
<td>Span, m (in.)</td>
<td>0.217 (8.542)</td>
</tr>
<tr>
<td></td>
<td>Aspect ratio</td>
<td>1.086</td>
</tr>
<tr>
<td></td>
<td>Root chord, at fuselage center line, m (in.)</td>
<td>0.353 (13.896)</td>
</tr>
<tr>
<td></td>
<td>Tip chord, m (in.)</td>
<td>0.085 (3.355)</td>
</tr>
<tr>
<td></td>
<td>Taper ratio</td>
<td>0.241</td>
</tr>
<tr>
<td></td>
<td>Mean aerodynamic chord, m (in.)</td>
<td>0.248 (9.779)</td>
</tr>
</tbody>
</table>

| Sweepback angles: | Leading edge, deg | 70 |
|                  | 25-percent-chord line, deg | 64 |
|                  | Trailing edge, deg | 0 |
|                  | Dihedral angle, at airfoil mean line, deg | -3.64 |
| Incidence angle, deg | 0 |
| Airfoil section | (See fig. 5(a)) |
| Airfoil thickness ratio: | Exposed root | 0.05 |
| | Tip | 0.06 |
| Leading-edge radius at | Fuselage-line chord, m (in.) | 5.08 $\times 10^{-4}$ (0.020) |
| | Tip, m (in.) | 5.08 $\times 10^{-4}$ (0.020) |
| | Area of both elevons, m² (in²) | 0.005 (7.161) |

| Forward delta wing: | Area exposed, outside of fuselage, forward of wing | 0.002 (3.394) |
| | leading edge, m² (in²) | 0.002 (3.394) |
| | Leading-edge sweep, deg | 80 |

| Tip fin: | Area, each, m² (in²) | 0.004 (5.868) |
| | Span, m (in.) | 0.069 (2.730) |
| | Aspect ratio | 1.274 |
| | Root chord, m (in.) | 0.086 (3.383) |
| | Tip chord, m (in.) | 0.029 (1.135) |
| | Taper ratio | 0.336 |
| | Mean aerodynamic chord, m (in.) | 0.062 (2.445) |

Scale for original: HCV = 0.021
### TABLE I. - Concluded

Sweepback angles:
- Leading edge, top, deg: 55.0
- Leading edge, bottom, deg: 70.1
- Trailing edge, top, deg: 21.3
- Toe-in angle, deg: 7.5

Airfoil section:
- Leading-edge radius, m (in.): $5.08 \times 10^{-4}$ (0.020)

Center vertical tail:
- Area, exposed, m$^2$ (in$^2$): 0.007 (11.492)
- Span, exposed, m (in.): 0.086 (3.380)
- Aspect ratio of exposed area: 0.994
- Root chord, at fuselage surface line, m (in.): 0.128 (5.040)
- Tip chord, m (in.): 0.045 (1.760)
- Taper ratio: 0.349
- Mean aerodynamic chord of exposed area, m (in.): 0.093 (3.664)

Sweepback angles:
- Leading edge, deg: 55.0
- Trailing edge, deg: 24.6

Airfoil section:
- Thickness ratio at:
  - Tip: 0.106
  - Root: 0.106
- Leading-edge radius, m (in.): $5.08 \times 10^{-4}$ (0.020)

Fuselage:
- Length, m (in.): 0.508 (20.000)
- Maximum height, m (in.): 0.071 (2.782)
- Maximum width, m (in.): 0.073 (2.866)
- Fineness ratio of equivalent round body: 6.822
- Planform area, m$^2$ (in$^2$): 0.026 (40.445)
- Wetted area, m$^2$ (in$^2$): 0.083 (128.460)
- Wetted area, with wing on, m$^2$ (in$^2$): 0.078 (120.695)
- Wetted area, with both delta wings on, m$^2$ (in$^2$): 0.077 (118.747)
- Base area, m$^2$ (in$^2$): 0.002 (3.726)

Complete model, with both delta wings:
- Planform area, m$^2$ (in$^2$): 0.052 (79.960)
- Aspect ratio of planform: 0.913

Scale for original $H/C_L = 6.62$
FIGURE 3
CODE APPLICATION

SCRAM
SIMPLE OVERALL PERFORMANCE
1-D, INVISCID

STUB
DETACHED SHOCK
2-D, INVISCID

EDDYBL
BOUNDARY LAYER
2-D, VISCOUS

INLET
INLET DESIGN
2-D, INVISCID

SEAGULL
CONICAL SHOCK STRUCTURE
2-D, INVISCID

EDDYBL and SEAGULL

COMBUSTOR ANALYSIS
1-D, INVISCID

FIGURE 4 - Effective Mass Flow entering engine.

Acelf = Effective Capture Area
Table 2: Forebody Compressive Qualities

<table>
<thead>
<tr>
<th>Forebody</th>
<th>$P_2/P_{oo}$</th>
<th>$M_2$</th>
<th>$P_2/P_{oo}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1</td>
<td>.7829</td>
<td>7.51</td>
<td>8.97</td>
</tr>
<tr>
<td>#2</td>
<td>.8798</td>
<td>7.85</td>
<td>9.17</td>
</tr>
<tr>
<td>#3</td>
<td>.7780</td>
<td>7.33</td>
<td>14.63</td>
</tr>
</tbody>
</table>

FIGURE 5: Design #1
FIGURE 6: Design #2

FIGURE 7: Final Forebody Geometry
FIGURE 10: Heat Flux on Five Degree Cone

![Heat Flux Graph](image)

ORIGINAL PAGE IS OF POOR QUALITY

FIGURE 11: Boundary Layer Growth vs. Axial Distance on Five Degree Cone With Suction

![Boundary Layer Growth Graph](image)
FIGURE 12: Boundary Layer Growth vs. Reynolds Number on Five Degree Cone
With Suction

FIGURE 13: Mass Flux on Five Degree Cone
FIGURE 14: Heat Flux on Five Degree Cone
With Suction

FIGURE 15
Inlet Area vs. Horizontal Distance
Mesh 9.3 to Mean 4 Compression
FIGURE 16
Cowl and Centerbody Contours
Mach 9.3 to Mach 4 Compression

Vertical distance from Body

Horizontal Distance Along Body

FIGURE 17
Mach Number on Cowl Surface
Mach 9.3 to Mach 4 Compression

Mach Number

Horizontal Distance Along Body

Normal Shock
FIGURE 18
Pressure on Cowl Surface vs. Distance
Mach 9.3 to Mach 4 Compression

ORIGINAL PAGE IS
OF POOR QUALITY

FIGURE 19
Inlet Area vs. Horizontal Distance
Mach 9.3 to Mach 3 Compression
FIGURE 22
Pressure on Cowl Surface vs. Distance
Mach 3.3 to Mach 3 Compression

ORIGINAL PAGE IS OF POOR QUALITY

FIGURE 23: Shock Structure on Final Design
FIGURE 26: Mach Number vs. Axial Distance on Final Design

ORIGINAL PAGE IS OF POOR QUALITY

FIGURE 27
Inlet Area vs. Horizontal Distance
Mach 7 to Mach 4 Compression
FIGURE 30
Pressure Ratio on Cowl Surface
Mach 7 to Mach 4 Compression

FIGURE 31
Total Pressure Ratio vs. Axis Distance
Mach 7 to Mach 4 Compression

TOTAL PRESSURE RECOVERY ALONG CENTERBODY CONTOUR

ORIGINAL PAGE IS OF POOR QUALITY
FIGURE 32
Total Pressure Ratio vs. Axial Distance
Mach 7 to Mach 4 Compression

TOTAL PRESSURE RECOVERY ALONG COWL CONTOUR

ORIGINAL PAGE IS OF POOR QUALITY

FIGURE 33  Geometry of the Combustor

Wall of Combustor

Height (feet)

Axial distance (feet)
TABLE 3 Information on Injectors

<table>
<thead>
<tr>
<th></th>
<th>Injector #1</th>
<th>Injector #2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Distance</td>
<td>3.75 ft.</td>
<td>5.0 ft</td>
</tr>
<tr>
<td>Fuel stagnation</td>
<td>1000 K</td>
<td>1000 K</td>
</tr>
<tr>
<td>Temperature</td>
<td>4758000.0 N/m²</td>
<td>4758000.0 N/m²</td>
</tr>
<tr>
<td>Mach Number</td>
<td>1.0</td>
<td>3.0</td>
</tr>
<tr>
<td>Injector angle</td>
<td>0.78 Radians</td>
<td>0.0 Radians</td>
</tr>
<tr>
<td>from horizontal</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wall angle at injector</td>
<td>0.0 Radians</td>
<td>1.57 Radians</td>
</tr>
</tbody>
</table>
FIGURE 34: Combustor Temperature vs. Combustor Axial Distance

ORIGINAL PAGE IS OF POOR QUALITY
FIGURE 35: Combustor Efficiency vs. Combustor Axial Distance
FIGURE 36: Combustor Pressure vs. Combustor Axial Distance
FIGURE 37: Combustor Mach No. vs. Combustor Axial Distance

ORIGINAL PAGE IS
OF POOR QUALITY
FIGURE 38 - ISOLATED LIFT DRAG AND MOMENT COEFFICIENTS

NOTES:
1. "A" Point of application of forces.
2. Free stream pressure has been subtracted from all surfaces prior to Integrations.

<table>
<thead>
<tr>
<th>(M_w)</th>
<th>(L_x/L_y)</th>
<th>(L_y/L_x)</th>
<th>(e)</th>
<th>(C_{Lx})</th>
<th>(C_{Ly})</th>
<th>(C_{mx})</th>
<th>(P/q)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>6.52</td>
<td>18.54</td>
<td>6.0</td>
<td>1.50</td>
<td>-0.136</td>
<td>-0.145</td>
<td>0.737</td>
</tr>
<tr>
<td></td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.89</td>
<td>-0.127</td>
<td>-0.193</td>
<td>0.747</td>
</tr>
<tr>
<td></td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>0.69</td>
<td>0.94</td>
<td>1.113</td>
<td>0.010</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>(M_w)</th>
<th>(L_x/L_y)</th>
<th>(L_y/L_x)</th>
<th>(e)</th>
<th>(C_{Lx})</th>
<th>(C_{Ly})</th>
<th>(C_{mx})</th>
<th>(P/q)</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.0</td>
<td>6.52</td>
<td>18.54</td>
<td>6.0</td>
<td>1.50</td>
<td>-0.136</td>
<td>-0.145</td>
<td>0.737</td>
</tr>
<tr>
<td></td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.89</td>
<td>-0.127</td>
<td>-0.193</td>
<td>0.747</td>
</tr>
<tr>
<td></td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>0.69</td>
<td>0.94</td>
<td>1.113</td>
<td>0.010</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>(M_w)</th>
<th>(L_x/L_y)</th>
<th>(L_y/L_x)</th>
<th>(e)</th>
<th>(C_{Lx})</th>
<th>(C_{Ly})</th>
<th>(C_{mx})</th>
<th>(P/q)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>6.52</td>
<td>18.54</td>
<td>6.0</td>
<td>1.50</td>
<td>-0.136</td>
<td>-0.145</td>
<td>0.737</td>
</tr>
<tr>
<td></td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.89</td>
<td>-0.127</td>
<td>-0.193</td>
<td>0.747</td>
</tr>
<tr>
<td></td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>0.69</td>
<td>0.94</td>
<td>1.113</td>
<td>0.010</td>
</tr>
</tbody>
</table>

Original page is of poor quality.
Appendix A1: SCRAM
Description

SCRAM is a one-dimensional airframe-integrated scramjet engine simulation program. It was written at the NASA Langley Research Center to give a first-order estimate of scramjet performance. SCRAM performs a real gas analysis for a five-stage model. These stages include the freestream, the vehicle forebody, the inlet, the combustor, and the exhaust nozzle. Once the flow for each stage is analyzed, the overall performance is calculated and a summary is given.

SCRAM is capable of analyzing: (1) oxygen, hydrogen, and air injection into the inlet and combustor; (2) axisymmetric or planar forebodies; (3) mixed combustion; (4) the boundary layer for a flat plate; (5) multistage perpendicular or parallel injection; (6) equilibrium or frozen flow referenced by area ratio in the combustor; (7) several internal hydrogen fuel mixing models; (8) a swept, symmetric wedge inlet, calculating inviscid real gas performance. The program only works for started inlets, and flow must be supersonic through the inlet and nozzle portions of the code. In addition, SCRAM is not usable if the combustor flow becomes subsonic. It is currently configured to handle two-dimensional engines only.

The freestream flowfield is calculated in Subroutine FSC1, where, given altitude, Mach number, and other parameters, it computes static temperature and pressure from a table based on the 1979 Standard Atmospheric Table. The other freestream flow properties are calculated from subroutines which find the real gas composition and the dynamic state variables.

The forebody flow properties are calculated in Subroutine FLOWF2. This subroutine accounts for forebody shocks, forebody boundary layer heat transfer, and friction losses, given the freestream conditions and forebody parameters. It utilizes a specified forebody kinetic energy efficiency that approximates engine capture flow momentum losses due to
the forebody shock system. Boundary layer properties and heat transfer through the boundary layer are calculated iteratively.

Subroutine INLET1 computes the flow properties for the inlet. It models flow through the inlet as supersonic flow undergoing a one-dimensional compression process with heat transfer and friction losses. Area ratio, length, and kinetic energy efficiency are the required inputs. INLET1 also is capable of handling oxygen/hydrogen mass flow injection.

The flow through the combustor is analyzed in Subroutine COMBUST. COMBUST uses perpendicular hydrogen fuel injection for up to seven injector stations; one of these stations can also be set for parallel injection. It approximates shock losses with the use of the kinetic energy efficiencies input by the user. Flow is initially assumed to be in equilibrium, but provisions can be made for frozen flow to occur. Regenerative cooling may also be taken into account.

Subroutine NOZZLE1 describes a one-dimensional expansion process simulating gas expansion for a scramjet nozzle. It computes boundary layer properties for freestream Mach numbers greater than 6.0, but neglects them for freestream Mach numbers less than 6.0.

Finally, subroutine SCRAP1 computes engine cycle performance parameters. It utilizes conservation of momentum equations applied to the engine as a whole, using engine geometry and flow properties computed by the component routines as inputs. It outputs thrust, specific impulse, and the coefficient of thrust as the scramjet performance parameters.
<table>
<thead>
<tr>
<th>B</th>
<th>1</th>
<th>4 [10 DEG INLET, CRG=5.5, MO=7, (HPB RECIPE)]——&gt; 4 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>33863.2</td>
<td>7.0000</td>
</tr>
<tr>
<td>2.5594</td>
<td>-8.8888</td>
<td>0.9990</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.0000</td>
</tr>
<tr>
<td>5.5000</td>
<td>-8.8888</td>
<td>555.55</td>
</tr>
<tr>
<td>0.0000</td>
<td>-888.88</td>
<td>0.0000</td>
</tr>
<tr>
<td>5.5000</td>
<td>-8.8888</td>
<td>0.10581</td>
</tr>
<tr>
<td>1.0000</td>
<td>1.0000</td>
<td>1.0000</td>
</tr>
<tr>
<td>0.0001</td>
<td>0.1000</td>
<td>1.0000</td>
</tr>
<tr>
<td>0.0100</td>
<td>0.0550</td>
<td>0.0804</td>
</tr>
<tr>
<td>0.0100</td>
<td>0.0677</td>
<td>0.09945</td>
</tr>
<tr>
<td>1.0000</td>
<td>200.00</td>
<td>555.55</td>
</tr>
<tr>
<td>-88888.8</td>
<td>-8888.8</td>
<td>-88888.8</td>
</tr>
<tr>
<td>1</td>
<td>3</td>
<td>2</td>
</tr>
<tr>
<td>-8888</td>
<td>1.0000</td>
<td>555.55</td>
</tr>
<tr>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

ORIGINAL PAGE IS OF POOR QUALITY
SAMPLE SCAN OUTPUTS

ORIGINAL PAGE IS OF POOR QUALITY
ORIGINAL PAGE IS OF POOR QUALITY
APPENDIX A2: STUB
ORIGINAL PAGE IS OF POOR QUALITY
INTRODUCTION

Since steady supersonic flow fields are governed by hyperbolic differential equations, whereas steady subsonic flow fields are described by elliptic differential equations (Reference 2), the mixed region behind the detached bow shock structure cannot be handled by any steady-flow technique. However, the unsteady Navier-Stokes conservation equations become hyperbolic with time, irregardless of the sonic nature of the flow. Therefore, the time-dependent technique becomes a simplified means through which the steady-state solution is attained.

Program Description

Details of the time-dependent technique are given in References 2, 3, and 4, and only a brief description is outlined here.

The flow field was covered with a mesh and transformed into a system of rectangular coordinates. Initial guesses for the shock structure and flow properties were inputted and non-dimensionalized. The flow field was divided into four regions and treated separately: [1] the shock, treated as a discontinuity; [2] the body, examined in the same manner as the shock; [3] the inner points between the shock and body, analyzed by the unsteady conservation equations; and [4] the upper boundary of the inner points, computed through linear extrapolation and restricted to lie beyond the sonic line. Time steps were taken in accordance to the CFL stability criterion outlined in References 2 and 3.

First, the shock was analyzed for each time step. At a particular time step, an initial guess for the shock velocity was made (the shock will
"move" through time until it converges onto its "steady-state" location), and the new shock location was found. Flow properties at the shock were determined by the Rankine-Hugoniot relations, presented in Reference 3.

A quasi-one-dimensional characteristic line was then drawn from this new shock location through time to an old inner point. Flow properties were obtained, and the characteristic line was computed. The information obtained was used in the compatibility equation given in Reference 3, and flow field variables obtained from the compatibility equation were then compared to values obtained from the Rankine-Hugoniot equations. A new shock velocity was guessed until these flow field variables converged. Thus, an iterative process was established: properties based on a given shock velocity were obtained from the Rankine-Hugoniot equations; different values were obtained from a characteristics approach and were determined independently by a compatibility equation; another shock velocity was guessed, and the process was repeated until flow field properties at the shock converged. Finally, another time step was taken, and the entire process was repeated.

A similar computation was performed for the body. The process was simplified, since the body points did not move with time.

The governing equations utilized for the inner points between the shock and body are presented in the form given in Reference 3.

The spatial derivatives on the right-hand side were evaluated using finite differences. MacCormack's predictor-corrector technique was utilized in determining the flow field properties after a given time step, and details of the procedure can be found in Reference 2.

Finally, for an upper boundary condition beyond the sonic line, the flow is supersonic. That is, any information obtained there will not affect properties lying below the sonic line, and upper boundary values may then
simply be linearly extrapolated

All flow field properties were therefore computed in this manner for each time step taken.
This program will utilize the time-dependent technique to construct the shock structure generated by a supersonic blunt body.

Limitations:
- First-order accuracy for spatial derivatives
- 2-D analysis

NOTE: M, N = 8, 8

PROGRAM STUB
COMMON p, gamma, vinf, pinf, roinf, ainf, k,
    u(8,8), v(8,8), p(8,8), ro(8,8), sp(8,8), bp(8,8), b(8),
    dbdy(4)

Need to input actual numbers for m, n in ALL COMMON blocks
Also, need m, n for REAL block below

Real p, gamma, vinf, pinf, roinf, ainf, k,
    u, v, ro, sp, bp, b, dbdy,
    eps, epsb, epst, delx, dely, delt, wt0(4), theta(4),
    capc(8,8), capp(8,8), capr(8,8), capb(8,8), psi(8,8),
    zeta(4), del(4), pause, delzet

Integer m, n, step, ichoice, jk, tt
print*, 'Please input number of grid points (n, m):
read*, n, m
print*, 'Please input number of time steps:
read*, step
call INDAT(m, n, epss, epsb, epst)
call START(m, n, delx, dely, delt)
Do 10 j = 1, m
    wt0(j) = 0.
          10
          Do 50 jk = 1, step
call ATSHI(wt0, delt, eps, theta)
call ATBODY(m, n, delx, delt, epsb)
call CAP(m, n, theta, delx, wt0, capc, capp, capr, capb, psi, zeta, del)
call ATINFR: capc, capp, capr, capb, psi, dely, delzet, del, delt.
call BNDRY(m, n)
pause = step/jk
if (pause.ge.0.25.and.pause.lt.0.5) then
    print*, 'Number of time steps: ', jk
read*, ichoice
    if (ichoice.eq.1) then
        call OP(m, n, jk)
    else
        goto 60
    endif
elseif (pause.ge.0.5.and.pause.lt.0.75) then
    print*, 'Number of time steps: ', jk
read*, ichoice
    if (ichoice.eq.1) then
        call OP(m, n, jk)
    else
        goto 60
    endif
endif
else if (pause.ge.0.75.and.pause.lt.0.9) then
   print*, 'Number of time steps: ',jk
   read*, ichoice
   if (ichoice.eq.1) then
      call OP(m,n,tt)
   else
      goto 60
   endif
endif
50 continue
60 print*, 'END OF PROGRAM ',
   print*, 'Number of time steps = ',jk
   call OPFIN(m,n,step)
   stop
end

C======================================================================
C SUBROUTINE INDAT
C======================================================================
C
C Subroutine INDAT(m,n,epss,epsb,epst)
C
COMMON pi, gamma, vinf, pinf, roinf, ainf, k,
     u(8,8), v(8,8), p(8,8), ro(8,8), sp(8,2), bp(8,2), b(8),
     dbdy(4)
integer m, n
Real pi, gamma, vinf, pinf, roinf, ainf, k, u,v,p,ro,sp,bp,b,
     epss,epsb,epst, arg, dbdy
open(3, file='shock.dat')

Upstream conditions and number of grid points are inputted
(m refers to the total number of y points, n refers to the
total number of x points); k refers to axisymmetric geometry
read(3,*) vinf, pinf, roinf, gamma, k, epss, epsb, epst
arg = gamma*pinf/roinf
ainf = SORT(arg)

j refers to the y location; inputted are the x locations for
the given i (x location); thus, the arrays contain a set of 2
numbers: the x and y location for the body and shock in
the physical plane
Do 20 j = 1, m
   read(3,*) (bp(j,i),i=1,2), (sp(j,i),i=1,2), dbdy(j)
 20 continue

Initial guesses for u, p, and ro are inputted: i and j refer
to points in the physical plane equidistant from each other;
thus, the arrays contain one number: the value of the property
at the location x, y (i,j)
Do 40 j = 1, m
   Do 30 i = 1, n
      read(3,*) u(j,i), v(j,i), p(j,i), ro(j,i)
 30 continue
40 continue
close(3)
return
end
Subroutine START(m,n,delx,dely,delt)

COMMON pi, gamma, vinf, pinf, roinf, ainf, k,
    u(8,8), v(8,8), p(8,8), ro(8,8), sp(8,2), bp(8,2), b(8),
    dbdy(4)
Real pi, gamma, vinf, pinf, roinf, ainf, k, u, v, p, ro, sp, bp, b,
- dbdy,
- delx(m), dely(m), delt, deltx, delty, arg, a

Integer n, m

delt = 1000.0
Do 20 j = 1, m
  delx(j) = (1.0/FLOAT(n-1))*(sp(j,1) - bp(j,1))
  dely(j) = bp(m,2)/FLOAT(m-1)
  b(j) = bp(j,2)
Do 10 i = 1, n
  p(i,j) = p(i,j)/pinf
  ro(i,j) = ro(i,j)/roinf
  u(i,j) = u(i,j) * SORT(roinf/pinf)
  v(i,j) = v(i,j) * SORT(roinf/pinf)
  arg = gamma*p(i,j)/ro(i,j)
  a = SORT(arg)
  deltx = delx(j)/(u(i,j) + a)
  delty = dely(j)/(v(i,j) + a)
  if (deltx.lt.delty.and.deltx.lt.delt) then
    delx = deltx/1.5
  elseif (delty.lt.deltx.and.delty.lt.delt)
    delt = delty/1.5
  endif
  continue
  continue
  return
end

C==================================
C SUBROUTINE CAP
C==================================
C
Subroutine ChF(m,n,theta,delx,w,capc,capp,capr,capb,psi,
    zeta,del)
COMMON pi, gamma, vinf, pinf, roinf, ainf, k,
    u(8,8), v(8,8), p(8,8), ro(8,8), sp(8,2), bp(8,2), b(8),
    dbdy(4)
Real pi, gamma, vinf, pinf, roinf, ainf, k, u, v, ro, sp, bp, b, dbdy,
    theta(m), delx(m), w(m), capc(m,n), app(m,n), capr(m,n),
    capb(m,n), psi(m,n), zeta(m,n), del(m)
Integer m, mi, n, ni
mi=m-1
ni=n-1
Do 20 j = 1, m
  Do 10 i = 1, n
    del(i,j) = sp(i,j) - bp(j,1)
    zeta(i,j) = (i-1)*delx(j)
    capc(j,i) = (zeta(j,i)-1.0)*dbdy(j) - (zeta(j,i)/
        TAN(theta(j)))
    capp(j,i) = ALOG(p(j,i))
    capr(j,i) = ALOG(ro(j,i))
    capb(j,i) = (u(j,i)-w(j)*zeta(j,i)+v(j,i)*capc(j,i))/
        (sp(j,i)-bp(j,i))
  end do 10
end do 20
This is good for converting inner point parameter values into values at a certain location

Subroutine OP

x and y locations

bp(j,2) = y location (at j)
bp(j,1) = x location for given y location (j)
n1 = n-1

location of particular grid point at y (= bp(j,2)):
Do 20 j = 1, m
   y(j) =
   dx = (1.0/(FLOAT(n1)) * (sp(j,1) - bp(j,1))
   Do 10 i = 1, n
      x(j,i) = (n-1) * dx + bp(j,1)
   10    continue
20    continue

Thus, the value at that x and y location is:

sp(j,1)

The shock's x value at a given j/y location is:

sp(j,1)

return

SUBROUTINE ATSHK

Subroutine ATSHK(u,0,delt,eps,theta)

COMMON pi, omega, vinf, pinf, roinf, ainf, k,
   u, v, r, ro, sp, bp, dbdy,
   Real pi, omega, vinf, pinf, roinf, ainf, u, v, p, ro, sp, bp, dbdy,
   Integer j, jj, jff, ii, m, n, ml, ni, choice

The shock point is moved, according to wt0, and at the same y
location; also found is theta, the angle between the tangent
to the shock and the x-axis

Do 10 j = 1, m
   snew(:,1) = wt0(i) * delt + sp(j,1)
The Rankine-Hugoniot equations are used to determine u, p, and ro on the shock.

\[ \text{wouess}(j) = 1.0 \]

Do 20 j = 1, m

\[ \text{call EOSH(wouess(j),theta(j),ushk0(j),pshk0(j),rshk0(j),vshk0(j))} \]

arg = gamma*pshk0/rshk0

ashk = SORT(arg)

A characteristic is drawn from the new shock point to a point A, and u, p, and ro are interpolated at A(t0).

\[ \text{delt} = \text{ushk} - \text{ashk} \]

\[ \text{ptap}(1) = \text{spnew}(j,1) + \text{delt}*\text{delt}*\text{SIN(theta}(j)) \]

\[ \text{ptap}(2) = \text{spnew}(j,2) - \text{delt}*\text{delt}*\text{COS(theta}(j)) \]

\[ \text{pta}(1) = (\text{ptap}(1) - \text{bp}(j,1))/(\text{spnew}(j,1) - \text{bp}(j,1)) \]

\[ \text{pta}(2) = \text{ABS}(\text{ptap}(2)) \]

\[ m1 = m - 1 \]

\[ n1 = n - 1 \]

\[ ii = 1 \]

\[ jj = 1 \]

\[ \text{delx} = \text{bp}(m,2)/\text{float}(m1) \]

\[ \text{delz} = 1.0/\text{float}(n1) \]

\[ \text{incr} = \text{pta}(2)/\text{delx*jj} \]

if (incr.ge.1.) then

\[ jj = jj + 1 \]

go to 120

else

\[ \text{incr} = \text{pta}(1)/(\text{delz} * ii) \]

if (incr.ge.1.) then

\[ ii = ii + 1 \]

go to 130

endif

endif

\[ \text{ypos}(1) = \text{delz} + jj \]

\[ \text{ypos}(2) = \text{delz} + (jj-1) \]

\[ \text{zpos}(1) = \text{delz} + ii \]

\[ \text{zpos}(2) = \text{delz} + (ii-1) \]

\[ jj = jj + 1 \]

\[ ii = ii + 1 \]

if (jj.gt.m.or.ii.gt.n) then

print*, 'THERE IS A PROBLEM IN ATSHIK.'

print*, 'Please begin with a new w.'

go to 1150

endif

\[ \text{call INTERP2(zpos1,zpos2,pta}(1),u(jj,ii),u(jj,ii-1),uy1) \]

\[ \text{call INTERP2(zpos1,zpos2,pta}(1),v(jj,ii),v(jj,ii-1),vy1) \]

\[ \text{call INTERP2(zpos1,zpos2,pta}(1),p(jj,ii),p(jj,ii-1),py1) \]

\[ \text{call INTERP2(zpos1,zpos2,pta}(1),ro(jj,ii),ro(jj,ii-1),roi1) \]

\[ \text{call INTERP2(zpos1,zpos2,pta}(1),u(jj-1,ii),u(jj-1,ii-1),uy2) \]

\[ \text{call INTERP2(zpos1,zpos2,pta}(1),v(jj-1,ii),v(jj-1,ii-1),vy2) \]

end
A new characteristic slope is computed, and A locations are evaluated and compared, and iterated until convergence.

\[
\text{delt} = \frac{(\text{delt} + (\text{ua} - \text{aa}))/2.}{1}
\]

If (\text{comp} > \text{epss} or \text{compy} > \text{epss}) then

\[
\text{pta}(1) = \text{pta}(1)
\]
\[
\text{pta}(2) = \text{pta}(2)
\]

else

\[
\text{pta}(1) = (\text{pta}(1) - \text{bp}(j,1))/(\text{sp}(j,1) - \text{bp}(j,1))
\]
\[
\text{pta}(2) = \text{ABS}(\text{pta}(2))
\]

endif

The right side of the compatibility equation is computed at A and at the new shock point.

\[
\text{rhs}(\text{a}) = (\text{a} + \text{aa}/\text{gamma}) * (\text{sp}(j,1) - \text{bp}(j,1))
\]

\[
\text{rhs}(\text{a}) = (\text{a} + \text{aa}/\text{gamma}) * (\text{sp}(j,1) - \text{bp}(j,1))
\]

\[
\text{ja} = \text{j}^{\text{ja}}
\]

if (\text{ja} \cdot \text{eq} \cdot 1) then

\[
\text{rhs}(\text{h}) = \text{a}
\]

goto 500

else if (\text{ja} \cdot \text{eq} \cdot 0) then

\[
\text{ja} = \text{j} - 1
\]
endif

\[
\text{rsh}(\text{h}) = (\text{a} * \text{aa}/\text{gamma}) * (\text{sp}(\text{h}) - \text{sp}(\text{h}-1)) + \text{sin}(\text{theta}(\text{h}))
\]

\[
\text{rsh}(\text{h}) = (\text{a} * \text{aa}/\text{gamma}) * (\text{sp}(\text{h}) - \text{sp}(\text{h}-1)) + \text{sin}(\text{theta}(\text{h}))
\]

500

\[
\text{rsh}(\text{h}) = 0.5 * (\text{rsh}(\text{h}) - \text{rsh}(\text{h}))
\]

\[
\text{wguess} \text{ is evaluated and a new guess is inputted if desired}
\]

\[
\text{ucmat} = (\text{a} * \text{aa}/\text{gamma}) * (\text{sp}(\text{h}) - \text{sp}(\text{h}-1)) + \text{sin}(\text{theta}(\text{h}))
\]

\[
\text{diff} = \text{ABS}(\text{ucmat} - \text{rsh})
\]

write(*,1000) ushk, ucmat, diff

1000 format(//,

'U from Rankine-Hugoniot = ', E10.6, //,

'U from compatibility = ', E10.6, //,

'Difference = ', E10.6, //)

print*,

print*, 'CONTINUE ITERATION (1) YES; (2) NO?'

read*, ichoq

if (ichoq.eq.1) then

write(*,2000) wguess(j)

2000 format(//, 'New recent guess for w = ', E10.6, //)

endif
read wguess(j)
go to 100
else
call EOSHL(wguess(j),theta(j),ushk,pshk,roshk,vshk)
      u(j,n) = vshk*COS(theta(j)) + ushk*SIN(theta(j))
      v(j,n) = vshk*SIN(theta(j)) - ushk*COS(theta(j))
      p(j,n) = pshk
      ro(j,n) = roshk
      wt0(j) = wguess(j)
      sp(j,1) = spnew(j,1)
      sp(j,2) = spnew(j,2)
endif
continue
return
end

C
C==============================================
C
C SUBROUTINE ATEODY
C==============================================
C
Subroutine ATEODY(m,n,dely,delt,epsb,phi,capp,capr,capv)

COMMON pi,gamma,vinf,pinf,roinf,aonf,k,
      u(8,8),v(8,8),p(8,8),ro(8,8),sp(8,2),bp(8,2),
      dbdv(4)

Real pi,gamma,vinf,pinf,roinf,aonf,k,u,v,p,ro,sp,bp,ddv,
      phi(m),capp(m),capr(m),capv(m),dely,dpdy,drdy,drdt,
      dvdy,v1,diff,delt,epsb

Integer m,n,jjj

Do 10 j = 1, m
    jjj = j
    if (jjj.eq.1) then
        phi(1) = pi/2.
        go to 5
    elseif (jjj.eq.m) then
        jjj = m - 1
    endif
    arg = (sp(jjj+1,2) - sp(j-1,2))/(sp(jjj+1,1)
           - sp(j-1,1))
    phi(j) = ATAN(arg)
    capp(j) = ALOG(p(j,1))
    capr(j) = ALOG(ro(j,1))
    capv(j) = v(j,1)/SIN(phi(j))
10 continue

C
C J = 1 is evaluated already
C
Do 20 j = 2, m
    p(j,1) = pi/1.1-(roinf*vinf*vinf)+(roinf*vinf*vinf*
                     COS(phi(j)))**2)
    dpdy = (capp(j+1)-capp(j-1))/(dely*2.0)
    drdy = (capr(j+1)-capr(j-1))/(dely*2.0)
    drdt = (i.0*gamma*(dpdy-gamma*drdy)*v(j,1)
    dvdy = (capv(j+1)-capv(j-1))/(dely*2.0)
    capr(j) = capr(j) + drdt*delt
    ro(j,1) = (2.718281828)**capr(j)
    dvdt = -v(j,1)*dvdy-(p(j,1)/ro(j,1))*dpdy*SIN(phi(j))
    v1 = (capv(j) + dvdt*delt)*SIN(phi(j))
    diff = ABS(v(j,1)-v1)
    if (diff.gt.0.001) then
        v(j,1) = v1
        go to 15
    else
SUBROUTINE ATINTR

ATINTR calculates interior point properties:
\[ \text{capr}(i,j), \ u(i,j), \ v(i,j), \ \psi(i,j) \]

Note that input values need to be transferred to subroutine;
also, variable \((m,n)\) needs to have \(m\) and \(n\) transferred;
don't forget that \((m,n)\) refers only to INTERIOR points.

Subroutine ATINTR(cap, capr, capp, capb, psi, dely, delzet, delt, del)

COMMON pi, gamma, vinf, pinf, roinf, ainf, k,
\[ u(8,8), v(8,8), p(8,8), ro(8,8), sp(8,2), bp(8,2), \]
\[ dbdy(4) \]
Real \[ pi, gamma, vinf, pinf, roinf, ainf, k, u, v, p, ro, sp, bp, dbdy, \]
cap(m,n), capr(m,n), capp(m,n), capb(m,n), psi(m,n),
delzet, dely, delt(m), delt, drdz, drdy, dudz, dudy, dvdz, dvdy,
\[ \psi d i z, d p s i d y, d p d z, d p d y, \]

C*******NOTE: The next 3 lines require \(m\)'s actual value

\[ \text{dudt}(8,8), \ \text{dvdt}(8,8), \ \text{dpsidt}(8,8), \ \text{drdt}(8,8), \ rbar(8,8), \]
\[ \text{ubar}(8,8), \ \text{vbar}(8,8), \ \text{psibar}(8,8), \ \text{psinew}(8,8), \]
\[ \text{unew}(8,8), \ \text{vnew}(8,8), \ \text{psinew}(8,8), \]
dudt1, dvdt1, ddtdt1, drdt1, drdtav, dudtav, dvdtav, dstdt,

Integer \(n, m, n1, m1\)
\[ \text{nl} = n - 1 \]
\[ \text{delzet} = 1.0 * \text{FLOAT}(n1) \]
\[ \text{ml} = m - 1 \]
Do 20 j = 1, n1
    Do 10 i = 2, m1
    
For all interior \((i, j)\), spatial derivatives are calculated

\[ \text{call FWD(capr}(j, i), \text{capr}(j+1, i), \text{delzet}, \text{drdz}) \]
\[ \text{call FWD(cap}(j, i), \text{cap}(j+1, i), \text{dely}, \text{dudy}) \]
\[ \text{call FWD(u}(j, i), \text{u}(j+1, i), \text{delzet}, \text{dudz}) \]
\[ \text{call FWD(v}(j, i), \text{v}(j+1, i), \text{delzet}, \text{dvdz}) \]
\[ \text{call FWD(\psi}(j, i), \text{v}(j+1, i), \text{dely}, \text{dudy}) \]
\[ \text{call FWD(\psi}(j, i), \text{psibar}(j+1, i), \text{delzet}, \text{dpsidz}) \]
\[ \text{call FWD(\psi}(j, i), \text{psinew}(j+1, i), \text{dely}, \text{dpsidy}) \]
\[ \text{call FWD(p}(j, i), \text{p}(j+1, i), \text{delzet}, \text{dpsidz}) \]
\[ \text{call FWD(p}(j, i), \text{p}(j+1, i), \text{dely}, \text{dpsidy}) \]
\[ \text{call EOMAT(j, i, capc}(j, i), \text{capp}(j, i), \text{cap}(j, i), \text{capb}(j, i), \]
\[ \text{psibar}(j, i), \text{psinew}(j, i), \]
\[ \text{dudy, dudz, dvdy, dvdz, dudy, dvdy, dvdz, drdy, drdz, dvdy, dvdz}, \]
\[ \text{drd} t(j, i) \]
\[ \text{rbar}(i, i) = \text{cap}(j, i) + \text{drdt}(j, i) * \text{delt} \]
\[
\text{v} \left( \text{v} \right) = \text{v} \left( \text{v} \right)^{\gamma} \\
\text{p} \left( \text{p} \right) = \text{p} \left( \text{p} \right)^{\gamma} \\
\text{u} \left( \text{u} \right) = \text{u} \left( \text{u} \right)^{\gamma} \\
\text{r} \left( \text{r} \right) = \text{r} \left( \text{r} \right)^{\gamma}
\]

**ORIGINAL PAGE IS OF POOR QUALITY**

**J = 1 is first analyzed; the rest are analyzed accordingly**

**Do 15 i = 2, n1**

**call REAR(rbar(1,i),rbar(1,i-1),delzet,drdz)**

**call REAR(rbar(1,i),rbar(2,i),dely,drdy)**

**call REAR(ubar(1,i),ubar(1,i-1),delzet,dudy)**

**call REAR(ubar(1,i),ubar(2,i),dely,dudy)**

**call REAR(vbar(1,i),vbar(1,i-1),delzet,dvdz)**

**call REAR(vbar(1,i),vbar(2,i),dely,dvdy)**

**call REAR(psibar(1,i),psibar(1,i-1),delzet,dpsidz)**

**call REAR(psibar(1,i),psibar(2,i),dely,dpsidy)**

**call REAR(pbar(1,i),pbar(1,i-1),delzet,dpdzeta)**

**call REAR(pbar(1,i),pbar(2,i),dely,dpdy)**

**call EOMDT(1,i,capc(1,i),capp(1,i),capr(1,i),capb(1,i),**

**psi(1,i),del(1),drdz,dudz,dvdz,dpdz,dpdy,dvdy,drdy,**

**dpdv,dpsidz,dpsidy,dudt1,dvdt1,dstd1,drdt1)**

**drdtav = 0.5 \times (drdt(j,i) + drdtl)**

**dudtav = 0.5 \times (dudt(1,i) + dudt1)**

**dvdtav = 0.5 \times (dvdt(1,i) + dvdtl)**

**dstdav = 0.5 \times (dpsidt(1,i) + dstdl)**

**caprnw(j,i) = capr(1,i) + (drdtav \times delt)**

**unew(j,i) = u(1,i) + (dudtav \times delt)**

**vnew(j,i) = v(1,i) + (dvdtav \times delt)**

**psinew(j,i) = psi(1,i) + (dstdtav \times delt)**

**continue**

**Do 30 i = 2, n1**

**call REAR(rbar(j,i),rbar(j,i-1),delzet,drdz)**

**call REAR(rbar(j,i),rbar(j,i-1),dely,drdy)**

**call REAR(ubar(j,i),ubar(j,i-1),delzet,dudy)**

**call REAR(ubar(j,i),ubar(j,i-1),dely,dudy)**

**call REAR(vbar(j,i),vbar(j,i-1),delzet,dvdz)**

**call REAR(vbar(j,i),vbar(j,i-1),dely,dvdy)**

**call REAR(psibar(j,i),psibar(j,i-1),delzet,dpsidz)**

**call REAR(psibar(j,i),psibar(j,i-1),dely,dpsidy)**

**call REAR(pbar(j,i),pbar(j,i-1),delzet,dpdzeta)**

**call REAR(pbar(j,i),pbar(j,i-1),dely,dpdy)**

**call EOMDT(j,i,capc(j,i),capp(j,i),capr(j,i),capb(j,i),**

**psi(j,i),del(i),drdz,dudz,dvdz,dpdz,dpdy,dvdy,drdy,**

**dpdv,dpsidz,dpsidy,dudt1,dvdt1,dstd1,drdt1)**

**drdtav = 0.5 \times (drdt(j,i) + drdtl)**

**dudtav = 0.5 \times (dudt(j,i) + dudt1)**

**dvdtav = 0.5 \times (dvdt(j,i) + dvdt1)**

**dstdav = 0.5 \times (dpsidt(j,i) + dstdl)**

**caprnw(j,i) = capr(j,i) + (drdtav \times delt)**

**unew(j,i) = u(j,i) + (dudtav \times delt)**

**vnew(j,i) = v(j,i) + (dvdtav \times delt)**

**psinew(j,i) = psi(j,i) + (dstdtav \times delt)**

**continue**

**Do 105 j = 1, n1**

**Do 90 i = 2, n1**

**u(j,i) = unew(j,i)**

**v(j,i) = vnew(j,i)**

**re(j,i) = (2.718281828)**(caprnw(j,i))

**arg = psinew(j,i) + gamma*caprnw(j,i)**

**p(j,i) = (2.718281828)**arg

**continue**

**105 Continue**
SUBROUTINE BNDRY

Subroutine BNDRY(m : n)

COMMON pi,gamma,vinf,pinf,roinf,k,
   . u(8,8),v(8,8),sp(8,2),sp(8,2),bp(8,2),
   . ddbd(4)

Real pi,gamma,vinf,pinf,roinf,k,u,v,p,ro,sp,bp,ddb

Integer m,n

Do 10 i = 1, n
   u(m,i) = ((u(m-1,i)-u(m-2,i))/u(m-2,i)) + u(m-1,i)
   v(m,i) = ((v(m-1,i)-v(m-2,i))/v(m-2,i)) + v(m-1,i)
   p(m,i) = ((p(m-1,i)-p(m-2,i))/p(m-2,i)) + p(m-1,i)
   ro(m,i) = ((ro(m-1,i)-ro(m-2,i))/ro(m-2,i)) + ro(m-1,i)
   10 Continue

SUBROUTINE FWD

FWD generates the forward spatial derivative at (i,j):

dvardij

Subroutine FWD(vari,vari,ij,dx,dvardij)

Real vari,vari,ij,dx,dvardij

dvardij = (vari - vari,ij)/dx

SUBROUTINE REAR

REAR generates the rearward spatial derivative at i,j:

dvardij

Subroutine REAR(vari,vari,ij,dx,dvardij)

Real vari,vari,ij,dx,dvardij

dvardij = (vari - vari,ij)/dx

end
SUBROUTINE INTERP

INTERP interpolates for a value answer

Subroutine INTERP(row1, row2, mid, col1, col2, answer)
Real row1, row2, mid, col1, col2, answer

answer = ((mid-row2)*(col1-col2))/(row1-row2) + col2
return
end

C===============================================
C SUBROUTINE EOMOT
C===============================================

EOMOT computes values for time derivatives of the
equations of motion at (i,j):
drdt, dudt, dvdt, dpsidt

Subroutine EOMOT(j,i,capc,capp, carp, capb, psi, del, drdz, dudz,
dvdy, dudy, dvdt, dpdz, dpsyidz, dpsyidy, dudt, dvdt, dpsi, drdt, dudt)

COMMON pi, gamma, vinf, pinf, roinf, ainf, k,
    u(8,8), v(8,8), p(8,8), ro(8,8), sp(8,8), bp(8,8),
    dbdy(4)

Real pi, gamma, vinf, pinf, roinf, ainf, k, u, v, p, ro, sp, bp, dbdy,
capc, capp, carp, capb, psi, del, drdz, dudz, dvdy, dudy, dvdt, dpdz,
dpsyidz, dpsyidy, dpsi, drdt, dudt, dvdt, dpsi, drdt, y

Integer i, j, m, n
y = 1.0

drdt = -((capb * drdz) + (dudz/del) +
    ((capc/del) * dvdy) + (v(j,i)*drdy) +
    (k*(v(j,i)/y))

dudt = -((capb*dudz)+(v(j,i)*dudv)+((p(j,i)/(ro(j,i)*del))
    * dpdzeta))

dvdt = -((capb*dvdz)+(v(j,i)*dvdy)+((p(j,i)/ro(j,i)) * dpdzeta)
    + ((p(j,i)/ro(j,i)) * dpdy))

dpsi = -((capb * dpsidz) + (v(j,i) * dpsidy))

return
end

C===============================================
C SUBROUTINE EOSHK
C===============================================

Rankine-Hugoniot relations

Subroutine EOSHK(w, theta, ushk, pshk, roshk, vshk)

COMMON pi, gamma, vinf, pinf, roinf, ainf, k,
    u(8,8), v(8,8), p(8,8), ro(8,8), sp(8,8), bp(8,8),
    dbdy(4)

Real pi, gamma, vinf, pinf, roinf, ainf, k, u, v, p, ro, sp, bp, dbdy,
w, theta, unel, ushk, pshk, roshk, vshk

unel = (gamma-1.0)*((vinf-w)*SIN(theta))**2+2.*ainf*ainf
ushk = unel/((gamma+1.0)*((vinf-w)*SIN(theta))**2-(gamma-1.0))/(gamma+1.0)

pshk = (2.0*((vinf-w)*SIN(theta))**2-(gamma-1.0))/(gamma+1.0)

roshk = ((gamma+1.0)*pshk+(gamma-1.0))/(gamma+1.0)

vshk = vinf * COS(theta)

return
end
**SUBROUTINE OP**

**OF outputs values in the physical plane**

Subroutine OP(m,n,tt)

```
COMMON pi,gamma,vinf,pinf,roinf,ainf,k, u(8,8),v(8,8),p(8,8),ro(8,8),sp(8,8),bp(8,8),
   ddbv(4)
```

Real pi,gamma,vinf,pinf,roinf,ainf,k,u,v,p,ro,sp,bp,ddb,

C****** NOTE! the next line requires actual m,n values

C

```
x(8,5),y(8),dx
```

Integer tt

```
Open(file='shock.out')
```

```
1000    TIME STEP NUMBER: ',I4,/,)
```

```
1020    OUTPUT FROM ST JR
```

```
1040    PHYSICAL LOCATIONS OF GRID POINTS: ',//,
```

```
1060    format(5X, 'X Y',//)
```

```
Do 20 j = 1, m
```

```
v(j) = bp(j,2)
dx = (1.0/(FLOAT(n-1)))*(sp(j,1)-bp(j,1))
```

```
Do 10 i = 1, n
```

```
x(i) = (n-i) * dx + bp(j,1)
```

```
write(4,1050) i,x,j,y
```

```
format(5X,14,2X,F8.4,2X,F8.4)
```

Do 10 j = 1, m

```
v(j) = bp(j,2)
dx = (1.0/(FLOAT(n-1)))*(sp(j,1)-bp(j,1))
```

```
Do 10 i = 1, n
```

```
x(i) = (n-i) * dx + bp(j,1)
```

```
write(4,1080) y(j),sp(j,1),bp(j,1)
```

```
format(5X,F8.4,1X,F8.4)
```

```
Do 40 j = 1, m
```

```
write(4,1100) y(j)
```

```
format(5X,F8.4,1X,F8.4)
```

```
Do 60 j = 1, m
```

```
v(j) = bp(j,2)
dx = (1.0/(FLOAT(n-1)))*(sp(j,1)-bp(j,1))
```

```
Do 10 i = 1, n
```

```
x(i) = (n-i) * dx + bp(j,1)
```

```
write(4,1200) x(i),u(i),v(i),p(i),ro(i)
```

```
format(5X,F8.4,2X,F8.4)
```

```
Do 60 j = 1, m
```

```
write(4,1250) y(j),u(j),v(j),p(j),ro(j)
```

```
format(5X,F8.4,2X,F8.4)
```

```
Close(4)
```

return

```
end
```
Real pi, gamma, vinf, pinf, roinf, ainf, k, u, v, p, ro, sp, bp, dbdy,
C
C****** NOTE: the next line requires actual m,n values
C
  x(8,8), y(8), dx
 Integer tt
 Open(2, file='shock.fin')
  write(2, 1000) tt
  format(///,' TIME STEP NUMBER: ', I4, //)
  write(2, 1020)
  format(///,' OUTPUT FROM STUB: ', //)
  write(2, 1040)
  format(5X, 'I X J Y ',//)
  do 20 j = 1, m
    y(j) = bp(j, 2)
    dx = (1.0/(FLOAT(n-1)))*(sp(j, 1) - bp(j, 1))
    do 10 i = 1, n
      x(i, j) = (n-1) * dx + bp(j, 1)
      write(2,1050) i, x(i, j)
  10 continue
  continue
  write(2, 1060)
  format(///,' PHYSICAL LOCATIONS OF SHOCK AND BODY POINTS: ', //)
  //, ' (Note that x coordinates are presented for a given y)',
  5X, ' Y X locations: SHOCK BODY ', //)
  do 40 j = 1, m
    write(2,1080) y(j), sp(j, 1), bp(j, 1)
  40 continue
  write(2, 1100)
  format(///,' PHYSICAL LOCATIONS AND VALUES OF FLOW PROPERTIES: ', //)
  do 60 j = 1, m
    write(2,1150) y(j)
  1150 format(///,' Y LOCATION: ', F8.4, //)
  write(2, 1175)
  format(5X, ' X U V P RO ',//)
  do 50 i = 1, n
    write(2,1200) x(i, j), u(j, i), v(j, i), p(j, i), ro(j, i)
  1200 format(5X, F8.4, 2X, F8.4, 2X)
  50 continue
  60 continue
 Close(2)
 return
end
APPENDIX A3: SEAGULL
Description of the Program SEAGULL

The program SEAGULL is a code designed for the analysis of two-dimensional or axisymmetric supersonic inviscid flow of an ideal gas. All the discontinuities are treated explicitly. The limitation of the program is that the axial component of the mach number should remain supersonic during the execution. Although originally designed for internal flows, SEAGULL can be used for external flow due to optional input of the lower or upper wall geometry. The geometry for the upper wall and lower wall are assigned in subroutines GEOMB and GEOMC. Descriptions of the input and outputs are given on the following page. The code could also be used to merge several ducts into one analysis.

For the purposes of the propulsion group, the program is used to calculate the flow properties along the forebody of the hypersonic drone. The flow parameters obtained from this program will be used for the boundary layer code EDDYBL to obtain the shape of the boundary layer. Once the boundary layer geometry is known it could be added to the forebody geometry for a better estimation of the flow properties along the forebody.
The input parameters

- Steps between output
- Number of sharp corners on upper wall and lower wall (10 maximum each), their locations and angles in degrees
- Type of the flow (2D or axisymmetric)
- Mesh points
- Geometry description
- Constant or variable Gamma
- Mach number
- Distance to end the run
- Free stream Pressure, Temperature, Gamma, Inclination in degrees, Mach number, Gas constant
- Lower and upper wall temperatures

Output description

- Forces and Moments on the wall and on the R-Z plane
- Viscous and Inviscid forces
- Heat transfer from upper and lower wall
- The pressure, temperature, velocity component in the R and Z direction, the mach number, the streamline inclination in degrees, total temperature, for each mesh point and its R location.
Verification: Seagull

For the five degree cone, the downstream values output from SEAGULL at Mach 10 upstream conditions are:

\[
\begin{align*}
M &= 8.271 \\
P/P_\infty &= 3.060 \\
\rho/\rho_\infty &= 2.139
\end{align*}
\]

From [22], for a five degree cone with no angle of incidence to the freestream and \( M_\infty = 10 \), we have:

\[
\begin{align*}
M_\infty &= 9.925 \\
P/P_\infty &= 2.95 \\
\rho/\rho_\infty &= 1.8033
\end{align*}
\]

A comparison of these values illustrates a limitation of the code; specifically, SEAGULL's downstream values are somewhat in error. Also, SEAGULL shows that the flowfield downstream of the shock decreases in velocity with increasing distance from the nose. These effects are caused by the duct flow modelling that SEAGULL uses. This effect is minimal, however, and the properties on the wall change by less than 5% over the entire forebody if the walls of the duct are kept far apart to minimize the compression caused by the internal modeling of SEAGULL. This introduces some error into the downstream flow properties. This error is unavoidable due to the necessity to model multi-ramp geometries, which was not possible by merely using conical shock tables because the ramps are not subject to uniform freestream conditions.
APPENDIX A4: EDDYBL
EDDYBL is a fully compressible, two-dimensional or axisymmetric computer program for the calculation of boundary layer properties. The program accounts for mass flux at the body surface, heat flux, pressure and temperature gradients, and both transverse and longitudinal body curvature. The program has the capability to start from a stagnation point and proceed through transition into turbulence. The equations used by the program cannot account, however, for the effects of shock-boundary layer interaction or separated flow. If the program encounters separation, execution terminates at that point, although the data is saved for all calculations prior to that point. Also, the program does not account for karman vortices or its effects in any way.

The calculation of the boundary layer properties can be made in two ways: using the program's two-equation model or its multiscale model [ref. 14]. In short, the two equation model uses conservation of mass and momentum in conjunction with an equation for the turbulent mixing energy, and another equation for the turbulent dissipation rate [ref. 4, p.2]. These last two equations account for the creation and destruction of turbulence.

The multiscale model differs from the two equation model in several ways. Although the multiscale model also uses the conservation of mass and momentum, as well as the same turbulent mixing energy equation, the multiscale model uses a slightly different equation for the turbulent dissipation rate to "predict the effects of plane strain and shear on homogenous turbulence." [ref.19, p.4]. The largest difference between the two models lies in the way that they treat the Reynolds' Stress tensor, which accounts for vortices and turbulence.
The two-equation model lumps all of the vortices, or "eddy", into one term. The multiscale model lumps the eddies into "large" and "small" eddies. [From, p.7] This scaling of the turbulence proves to be more accurate than two-equation model. Our applications use the multiscale model exclusively, and so further discussion will be limited to the multiscale model. References 1 through 4 offer a much more complete description of the two models used.

As previously mentioned, Eddyfl is a fully compressible boundary layer code. In addition to the other capabilities previously mentioned, Eddyfl has the capability to start from user-supplied boundary layer properties. That is, if the user knows what the properties of the boundary layer are at a certain point in the flowfield, then computation can begin at that point and there is no need to make calculations for points further upstream. Furthermore this allows restarting Eddyfl from previous runs. In our application, we will use Eddyfl to calculate boundary layer properties to a point where the free stream conditions change. We then restart computation from the old boundary layer properties with the new free stream conditions. In this way we proceed in a piece wise manner along the body in question.

There are many input parameters for Eddyfl. The free stream velocity, static temperature and pressure may be either be input directly or may be calculated by the program itself. The program will calculate the freestream properties based upon the far upstream total pressure, total temperature, Mach number and shock wave angle. Eddyfl uses the Prandtl-Meyer Relations to calculate the free stream properties downstream of the shock.
The properties of the free stream gas itself are also defined by the user. The ratio of specific heats, the specific gas constant, and both the laminar and turbulent Prandtl numbers must be defined by the user. The turbulent Prandtl number has been determined experimentally to be 0.9 for most flows, but communication with the author indicates that more accurate results are obtained if the turbulent Prandtl number is set to 8/9. The laminar Prandtl number is a function of viscosity, which is in turn a function of temperature. Its value is calculated by the user and input for each free stream condition. Another free stream property that is user-supplied is the free stream turbulence intensity. This value varies from 0.05% for well designed wind tunnels to 100% for turbulent jets. (ref. 14, p.257).

The properties of the surface itself are described by three separate data files. The first data file contains \( N \) arc-length-pressure pairs that define the static pressure at the body's surface. (The user determines the number of arc-length-pressure pairs, \( N \)). Specifying the pressure in this manner allows the user to impose pressure gradients upon the boundary layer. ENOYVA1 uses a cubic spline fit to determine pressures at intermediate points.

The second file defines the thermal properties of the surface. Again, \( N \) sets of arc length, temperature and pressure values are specified. Note that only temperature or heat flux need be specified. If temperature is specified, heat flux will be calculated. If heat flux is specified, then temperature will be calculated.

The last data file specifies the body geometry of the surface and the rate of mass addition or removal at the wall. \( N \) sets of five numbers define the mass flux and body geometry. Four of the numbers describe the arc length, axial distance, the radius from the center...
line, and the local curvature for a given arc length. The fifth number specifies the mass flux at that station.

Finally, within the program itself the user defines the angle that the free stream makes with the center line of the body. Also, the roughness height Reynolds's number is defined to allow for different surface roughnesses.

The grid resolution for the finite difference method is also determined by the user. Initial and final arc lengths, maximum number of stream wise steps and the initial step size are all defined by the user. The step size between stations will be changed by the program itself as needed to maintain accuracy and conserve time. Execution will be terminated when either the specified arc length has been reached or the program has reached the maximum number of stream wise steps. Also, the user can specify the number of grid points normal to the surface. More points will increase the resolution but will also increase computing time. The number of points normal to the surface remains constant. Also, the user must choose the model to be used. The closure coefficients for the models may also be changed, but the author has shown the default values to perform well when compared to experimental results. [ref. 11,18]. Also, the velocity profile of the boundary layer obeys the Sutherland Viscosity Law, although a power law may be invoked by the user.

Finally, the user must define boundary layer properties if starting from his own predetermined conditions. The coefficient of skin friction, the boundary layer thickness, shape factor, and frictional wall based on momentum thickness must all be supplied by the user in addition to the parameters mentioned above.
The program outputs boundary layer properties for each streamwise station [see attached sheet and ref 42, pp. 23-24]. The output listing is very complete and easily lends itself to comparison with experimental data. Additionally, the program outputs a boundary layer profile at the last station and at any intermediate station requested by the user. It also creates an output file that can be used to restart from the last station's properties.

The program has several basic limitations. First, it is limited to laminar or turbulent, non-rarefied boundary layer analysis. As previously mentioned, the program cannot account for separation or shock-boundary layer interaction. Also, free stream values remain constant throughout a given run, and if the free stream is indeed variable then the program must be run in a series of streamwise sections, each section maintaining constant free stream conditions. Finally, the program is limited to two-dimensional analysis, although the effects of streamwise curvature are accounted for in the axisymmetric case.
Program EDDYBL is a product of DCW Industries and has been tested by the author for compressible flows ranging as high as Mach 5 and including the effects of cooled walls and adverse pressure gradients. [19, p.11-13]. Accuracy of the computed values is shown to be within 8% of experimental results.

Despite verification of the program's accuracy, there are a number of factors affecting the program's application to hypersonic boundary layers. Among these are surface roughness and freestream turbulence. Another possibly important aspect which was not considered in this application were real gas effects, which become important at high Mach numbers. The group's original intention was to compare the results of this program with those of another boundary layer program which accounted for real gas effects. This comparison was never accomplished. However, results in literature for high Mach number flows and communication with researchers at NASA indicate that the freestream conditions under consideration are not sufficiently severe to warrant consideration of real gas effects. [16].
The following quantities are printed in the integral parameter portion of Program EDDYBL output.

<table>
<thead>
<tr>
<th>Name</th>
<th>Symbol/Equation</th>
<th>Definition</th>
<th>Dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td>BETA</td>
<td>$\beta = (2t/U_e) dU_e/dt$</td>
<td>Pressure gradient parameter</td>
<td>None</td>
</tr>
<tr>
<td>CFE</td>
<td>$c_{fe} = 2\tau_w/\rho_e U_e^2$</td>
<td>Skin friction based on edge density</td>
<td>None</td>
</tr>
<tr>
<td>CFW</td>
<td>$c_{fw} = 2\tau_w/\rho_w U_e^2$</td>
<td>Skin friction based on wall density</td>
<td>None</td>
</tr>
<tr>
<td>D/T</td>
<td>$H = \delta^e/\theta$</td>
<td>Boundary-layer thickness</td>
<td>ft</td>
</tr>
<tr>
<td>DELTA</td>
<td>$\delta^s$</td>
<td>Displacement thickness</td>
<td>ft</td>
</tr>
<tr>
<td>DLTAST</td>
<td>$\delta^t$</td>
<td>Dimensionless pressure gradient</td>
<td>ft</td>
</tr>
<tr>
<td>DPEDS</td>
<td>$d(P_e/\rho_{ew} U_e^2)/ds$</td>
<td>Dimensionless surface dissipation rate</td>
<td>None</td>
</tr>
<tr>
<td>DTEDX</td>
<td>$dT_e/dT$</td>
<td>Transformed edge temperature gradient</td>
<td>None</td>
</tr>
<tr>
<td>DUEDX</td>
<td>$dU_e/dT$</td>
<td>Transformed edge velocity gradient</td>
<td>None</td>
</tr>
<tr>
<td>ECOUNT</td>
<td></td>
<td>Number of points where $k &lt; 0$</td>
<td>None</td>
</tr>
<tr>
<td>EMAX</td>
<td>$\sqrt{\beta^e (p k)_{max}}/\tau_w$</td>
<td>Maximum turbulent energy parameter</td>
<td>None</td>
</tr>
<tr>
<td>ES</td>
<td>$S$</td>
<td>Dimensionless surface dissipation rate</td>
<td>Btu/ft$^2$/sec/°R</td>
</tr>
<tr>
<td>ETAEDG</td>
<td>$N_e$</td>
<td>Mesh point number at B.L. edge</td>
<td>None</td>
</tr>
<tr>
<td>HD</td>
<td>$H = q_w/(T_w - T_{aw})$</td>
<td>Heat transfer coefficient</td>
<td>Btu/ft$^2$/sec/°R</td>
</tr>
<tr>
<td>IEDGE</td>
<td>$N$</td>
<td>Total number of mesh points in B.L.</td>
<td>None</td>
</tr>
<tr>
<td>ITRO</td>
<td></td>
<td>Number of iterations for convergence</td>
<td>None</td>
</tr>
<tr>
<td>M</td>
<td>$m$</td>
<td>Streamwise step number</td>
<td>None</td>
</tr>
<tr>
<td>ME</td>
<td>$M_e$</td>
<td>Edge Mach number</td>
<td>None</td>
</tr>
<tr>
<td>MUE</td>
<td>$u_e$</td>
<td>Edge molecular viscosity</td>
<td>slug/ft/sec</td>
</tr>
<tr>
<td>NSKIP</td>
<td></td>
<td>Number of points below $u = USTOP$</td>
<td>None</td>
</tr>
<tr>
<td>NSTE</td>
<td>$h/\rho_e U_e \rho_C$</td>
<td>Stanton number based on edge density</td>
<td>None</td>
</tr>
<tr>
<td>NSTW</td>
<td>$h/\rho_e U_w \rho_C$</td>
<td>Stanton number based on wall density</td>
<td>None</td>
</tr>
<tr>
<td>NUE</td>
<td>$Pr_f h_a/\mu_e C_p$</td>
<td>Nusselt number based on edge viscosity</td>
<td>None</td>
</tr>
<tr>
<td>NUW</td>
<td>$Pr_f h_a/\mu_w C_p$</td>
<td>Nusselt number based on wall viscosity</td>
<td>None</td>
</tr>
<tr>
<td>PE</td>
<td>$P_e$</td>
<td>Edge pressure</td>
<td>lb/ft$^2$</td>
</tr>
<tr>
<td>QSD</td>
<td>$q_w$</td>
<td>Surface heat flux</td>
<td>Btu/ft$^2$/sec</td>
</tr>
<tr>
<td>RE</td>
<td>$\rho_e$</td>
<td>Edge density</td>
<td>slug/ft$^3$</td>
</tr>
<tr>
<td>REQELT</td>
<td>$Re_{\delta^e} = \rho_e U_e \delta^e/\mu_e$</td>
<td>Displacement thickness Reynolds number</td>
<td>None</td>
</tr>
<tr>
<td>RES</td>
<td>$Re_s = \rho_e U_s \delta^s/\mu_e$</td>
<td>Arc length Reynolds number</td>
<td>None</td>
</tr>
<tr>
<td>RETHET</td>
<td>$Re_s = \rho_e U_s \delta^s/\mu_e$</td>
<td>Momentum thickness Reynolds number</td>
<td>None</td>
</tr>
<tr>
<td>RFCCTOR</td>
<td>$r$</td>
<td>Recovery factor</td>
<td>None</td>
</tr>
<tr>
<td>RMI</td>
<td>$r_0$</td>
<td>Body radius</td>
<td>ft</td>
</tr>
<tr>
<td>RWALD</td>
<td>$\rho_w v_w$</td>
<td>Surface mass flux</td>
<td>slug/ft$^2$/sec</td>
</tr>
<tr>
<td>S</td>
<td>$s$</td>
<td>Arc length</td>
<td>ft</td>
</tr>
<tr>
<td>TAU D</td>
<td>$\tau_w$</td>
<td>Surface shear stress</td>
<td>lb/ft$^2$</td>
</tr>
<tr>
<td>TE</td>
<td>$T_e$</td>
<td>Edge temperature</td>
<td>°R</td>
</tr>
<tr>
<td>THETA</td>
<td>$\theta$</td>
<td>Momentum thickness</td>
<td>ft</td>
</tr>
<tr>
<td>UE</td>
<td>$U_e$</td>
<td>Edge velocity</td>
<td>ft/sec</td>
</tr>
<tr>
<td>UTAU</td>
<td>$u_T$</td>
<td>Friction velocity</td>
<td>ft/sec</td>
</tr>
</tbody>
</table>
The following quantities are printed in the profiles portion of Program EDDYBL output.

<table>
<thead>
<tr>
<th>Name</th>
<th>Symbol/Equation</th>
<th>Definition</th>
<th>Dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td>i</td>
<td>i</td>
<td>Mesh point number</td>
<td>None</td>
</tr>
<tr>
<td>y/delta</td>
<td>y/δ</td>
<td>Dimensionless normal distance</td>
<td>None</td>
</tr>
<tr>
<td>u/Ue</td>
<td>u/U_e</td>
<td>Dimensionless velocity</td>
<td>None</td>
</tr>
<tr>
<td>yplus</td>
<td>y^+ = u_+ / U_e</td>
<td>Sublayer distance coordinate</td>
<td>None</td>
</tr>
<tr>
<td>uplus</td>
<td>u^+ = u / U_e</td>
<td>Sublayer velocity coordinate</td>
<td>None</td>
</tr>
<tr>
<td>k/Ue^2</td>
<td>k/U_e^2</td>
<td>Dimensionless turbulent kinetic energy</td>
<td>None</td>
</tr>
<tr>
<td>omega</td>
<td>ν_ω U_e^2</td>
<td>Dimensionless dissipation rate</td>
<td>None</td>
</tr>
<tr>
<td>eps/mu</td>
<td>μ_ε / μ</td>
<td>Dimensionless eddy viscosity</td>
<td>None</td>
</tr>
<tr>
<td>L/delta</td>
<td>Lδ / (ωδ)</td>
<td>Dimensionless turbulent length scale</td>
<td>None</td>
</tr>
<tr>
<td>uv/tauw</td>
<td>-ρu′v′ / τ_w</td>
<td>Dimensionless Reynolds shear stress</td>
<td>None</td>
</tr>
<tr>
<td>T/Te</td>
<td>T/T_e</td>
<td>Temperature ratio</td>
<td>None</td>
</tr>
</tbody>
</table>
APPENDIX A5: INLET
A computer program entitled "Design of Supersonic Inlets by a Computer Program Incorporating the Method of Characteristics" (Ref. 11), dubbed "Inlet" by the propulsion group, was obtained to aid in the design of the inlet of the scramjet. Inlet is a sophisticated program written in 1968 by Bernhard H. Anderson of the NASA Lewis Research Center in Cleveland, Ohio. Inlet is based on steady, inviscid, irrotational, two-dimensional or axisymmetric flow. Its main solution scheme utilizes the method of characteristics, which was originally developed in 1929 but only practically implemented after the advent of modern computers. The method of characteristics as used in Inlet has its root in two-dimensional potential flow theory. Not only is the potential function, \( \phi \), included (\( \phi \) is actually incorporated into the method of characteristics equations), but the stream function, \( \psi \), is also included for determination of the diffuser's physical boundaries by numerical mass flux integration. The oblique shock relations are also extensively utilized to determine the flowfield; hence continuity, momentum, and energy equations are also considered in the calculations. Additional equations incorporated which yield design consideration data are mass flow spillage and additive drag equations. The numerical techniques used by Inlet to solve these equations simultaneously include finite difference techniques, Runge-Kutta numerical integration, and iteration.

Inlet is particularly useful in designing scramjet inlets since it allows the internal boundaries of the cowl or centerbody walls along with freestream conditions to be prescribed. This
results in the calculation of the internal cowl and centerbody wall geometries required for the input flow conditions. A useful application of Inlet is to calculate the internal cowl and centerbody boundaries from given upstream flow conditions and desired downstream conditions at the entrance of the diffuser. This is done by specifying forebody ramp geometry and freestream flow conditions along with desired shock pattern at the cowl lip and internal to the diffuser. Inlet then calculates the geometry required to cancel or reflect the shock at either the cowl or centerbody boundary by using the characteristic equations. Concurrently, Inlet determines the placement of the boundary by considering mass flux integration and performing iterations until the mass flux obtained by the placement of the boundary equals the initial mass flux entering the diffuser. In this manner, the internal boundaries are constructed and printed out as part of the output data. The boundaries can then be used to study the off-design conditions for the scramjet inlet just constructed.

Some important phenomenon to be studied in the next two quarters include the boundary layer, shock wave-boundary layer interaction, and chemical dissociation in the boundary layer. Because Inlet does not account for these phenomenon, a comparison of the results obtained from Inlet at high freestream Mach numbers (M>5) with experimental data in that regime will be essential in justifying the use of Inlet for the design of the forebody and diffuser geometry. Such a comparison performed in the Inlet description report (Ref 11) revealed good Inlet results for Mach numbers less then 2.5. However, comparisons at M = 3.85
begin to show deviation between the theoretical and experimental data, probably due to the increased boundary layer effect at this higher Mach number. Since boundary layer effects and shock wave-boundary layer interaction become increasingly more significant at Mach numbers greater than 5, the determination of the extent of this deviation for Mach numbers at the design region is essential if Inlet is to be useful for the final design of the forebody and diffuser. Nonetheless, should deviations prove to be significant, Inlet should at least yield some qualitative guidelines to consider in the final design of the scramjet inlet.
INPUTS

Freestream Conditions

AMO  Free-stream Mach number

GAM  Ratio of specific heats (Use 1.4 for ideal gas)

Ramp Geometry

THETC  Initial ramp angle, radians

BETAE  Estimation of ramp shock angle

ALPHA  Deflection angle of second ramp (measured from tangent of first ramp)

Cowl Specifications

COWLA  Initial interior cowl angle, radians

THETAL  Location of cowl lip relative to spike tip measured from centerline of ramp in radians (Note: THETAL is used only when the cowl contour is unspecified. Under this condition the cowl lip radius is 1.)

SPILL  Set SPILL = 0 to locate the cowl at along initial shock wave. Set SPILL = 1.0 to locate the cowl at THETAL.

XCOWL  x-coordinate of cowl lip position (Used when you want to place the cowl lip somewhere besides at the location where the ramp shock wave would intersect it. Use when inlet geometry is known and off-design performance is to be studied. Set XCOWL = 0 to place the cowl lip along the ramp shock wave.)

YCOWL  y-coordinate of cowl lip position (Used when you want to place the cowl lip somewhere besides at the location where the ramp shock wave would intersect it. Use when inlet geometry is known and off-design performance is to be studied. Set YCOWL = 0 to place the cowl lip along the ramp shock wave.)
OUTPUTS

Conditions in the Vicinity of Ramp Oblique Shock

Program outputs:
1. position of point for calculations
2. Mach number
3. angle of flow
4. angle of shock wave
5. ratio of static pressure behind shock to static pressure in front of shock
6. ratio of total pressure behind shock to total pressure in front of shock

Conditions on Ramp Contour
1. x-coordinate of point
2. y-coordinate of point
3. magnitude of velocity vector
4. flow angle
5. x-direction component of the velocity
6. y-direction component of the velocity
7. Mach number
8. ratio of static pressure to freestream total pressure
9. ratio of static pressure to total pressure behind the shock
SPECIFYING COWL GEOMETRY

Cowl contour is specified by a third-order polynomial of form

\[ Y = R(I,1) + R(I,2)X + R(I,3)X^2 + R(I,4)X^3 \]

The coefficients are specified in sets from 1 to \( NR \) where \( NR \) is an input variable specifying the number of sets of cowl contour coefficients to be read in. Also the starting position where the above equation is in effect must also be specified by the variable \( COWL(I) \) which specifies the \( X \)-coordinate to impose the specified set of coefficients.

Centerbody Specifications

Similar to cowl specification, centerbody is expressed as a third-order polynomial of the form

\[ Y = S(I,1) + S(I,2)X + S(I,3)X^2 + S(I,4)X^3 \]

The coefficients are specified in sets from 1 to \( NS \) where \( NS \) is an input variable specifying the number of sets of centerbody contour coefficients to be read in. Also the starting position where the above equation is in effect must also be specified by the variable \( BODY(I) \) which specifies the \( X \)-coordinate to impose the specified set of coefficients.

Calculation of Cowl or Centerbody Coefficients

When the Cowl or Centerbody coefficients are to be calculated set the corresponding coefficients equal to 0 and instead specify the flow conditions within the inlet by specifying the following input variables:

- \( NTHR \) Number of internal isentropic compression sections.
- \( AMT(J) \) Final Mach number specified for the end of the \( J^{th} \) section, \( J = 1,NTHR \).
- \( THR(J) \) Length of \( J^{th} \) section.
- \( ANG(J) \) Final surface angle (radians) of \( J^{th} \) section.
- \( NIS(J) \) Number of surface net points within \( THR(J) \).
Flowfield Specifications

START  Initial net point spacing parameter for point locations. Flow properties are calculated at each point. (A good initial value for start is 0.050)

DELP  Ratio of angular location of last initial data point to initial shock angle. The program constructs the initial datum line from the point on the surface, BODY (2), to a point in the field whose angular location is DELP times the initial shock angle.

PRINT  Set PRINT = 0 to bypass the output printout for the forebody flow field calculations. Set PRINT = 1.0 to print flow field calculations.

M  Number of points in flowfield calculations.

Internal Flow

NTHR  Number of internal isentropic compression sections.

ISHK  Used for off-design calculations only.
For on-design calculations set ISHK = 1.
For off-design ISHK is index which specifies at which centerbody location BODY (ISHK) the shock wave is canceled.

NSHK  Number of internal shock waves to be computed.
Set NSHK = 1 to cancel cowl lip shock.
NSHK - 1 specifies number of internal shock reflections.

Input Parameters

DELB  Initial increment used in calculation of ramp shock angle (set DELB = 1.4 x 10^{-4})

DELU1  Integration increment for conic flow field calculation (Set DELU1 = 1.0 x 10^{-5})

ERROR  Convergence parameter (Set ERROR = 1.0 x 10^{-4})

OFF  Set OFF = 0 for on-design calculations.
Set OFF = 1.0 for off-design calculations.

NDIM  Set NDIM = 2 for 2-D case.

NR  Number of sets of cowl contour coefficients.

ND  Number of increments in additive drag calculations.
Conditions in the flow field
1. x-coordinate of point
2. y-coordinate of point
3. magnitude of velocity vector
4. flow angle
5. x-direction component of the velocity
6. y-direction component of the velocity
7. Mach number
8. ratio of static pressure to freestream total pressure
9. ratio of static pressure to total pressure behind the shock

Conditions in the Vicinity of Cowl Oblique Shock
1. x-coordinate of point
2. y-coordinate of point
3. Mach number
4. flow angle
5. shock wave angle
6. ratio of pressure behind shock to pressure ahead of shock
7. ratio of total pressure behind shock to total pressure ahead of shock

Conditions along Cowl Contour Surface
1. x-coordinate
2. y-coordinate
3. magnitude of velocity vector
4. flow angle
5. x-direction component of velocity
6. y-direction component of velocity
7. Mach number
8. ratio of static pressure to freestream total pressure
9. ratio of static pressure to total pressure behind shock

Table of Cowl Coefficients
The program list cowl coefficients along with the x-coordinate position where the coefficients become effective.

Table of Centerbody coefficients
The program list centerbody coefficients along with the x-coordinate position where the coefficients become effective.
APPENDIX A6: CPIPE
### Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>cross section area</td>
</tr>
<tr>
<td>η</td>
<td>friction coefficient</td>
</tr>
<tr>
<td>f</td>
<td>force</td>
</tr>
<tr>
<td>h</td>
<td>enthalpy</td>
</tr>
<tr>
<td>M</td>
<td>molecular weight</td>
</tr>
<tr>
<td>P</td>
<td>static pressure</td>
</tr>
<tr>
<td>q</td>
<td>dynamic pressure</td>
</tr>
<tr>
<td>q̇, q̇</td>
<td>heat flux into control volume</td>
</tr>
<tr>
<td>T</td>
<td>temperature</td>
</tr>
<tr>
<td>v</td>
<td>velocity</td>
</tr>
<tr>
<td>V</td>
<td>moles of vibrate</td>
</tr>
<tr>
<td>W</td>
<td>mass flux</td>
</tr>
<tr>
<td>x</td>
<td>axial distance</td>
</tr>
<tr>
<td>β</td>
<td>angle between wall and duct centerline</td>
</tr>
<tr>
<td>θ</td>
<td>angle</td>
</tr>
<tr>
<td>ρ</td>
<td>density</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>Pr</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>μ</td>
<td>viscosity</td>
</tr>
<tr>
<td>γ</td>
<td>ratio of specific heats</td>
</tr>
<tr>
<td>Subscript</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>------------------------</td>
</tr>
<tr>
<td>a</td>
<td>air</td>
</tr>
<tr>
<td>f</td>
<td>fuel</td>
</tr>
<tr>
<td>f</td>
<td>friction</td>
</tr>
<tr>
<td>e</td>
<td>end of axial step</td>
</tr>
<tr>
<td>v</td>
<td>vitiate</td>
</tr>
<tr>
<td>w</td>
<td>wall</td>
</tr>
<tr>
<td>T</td>
<td>isentropic stagnation</td>
</tr>
<tr>
<td>aw</td>
<td>adiabatic wall</td>
</tr>
<tr>
<td>\infty</td>
<td>ambient condition</td>
</tr>
</tbody>
</table>
APPENDIX CPIPE

Summary

CPIPE is a fortran program written in 1968 by G.Y. Anderson of NASA Langley Research Center. CPIPE is a 1-dimensional, real gas analysis of the combustion of hydrogen in air in a supersonic channel (see Figure A1). CPIPE was originally intended to perform a one step analysis of the entire underbody flowfield, but can be modified to only analyze the combustor region. Given the initial upstream conditions and other inputs (see Table A1), CPIPE outputs the downstream 1-dimensional flow properties. CPIPE allows for several fuel injection points and allows the user to specify the fuel Mach number, stagnation pressure and temperature, and angle at which it is injected. There are several other options available regarding the boundary layer transition point and mixing models used (see Options, Table A1). In addition, CPIPE contains some empirical relations (introduced by Chuck McClinton, NASA Langley Research Center) which approximate the likelihood of autoignition and flameholding.
Figure A1 - Idealized one-dimensional supersonic combustor.
Conservation Equations

CPJPE uses the mass, momentum, and energy conservation equations to perform a one-step analysis of the flow properties in a supersonic channel. The equations and the control volumes they are applied to are shown explicitly below:

**Mass:** \[ w + \Delta w_f - w_g = 0 \]

**Energy:** \[ w h_T + \Delta w_f h_f + dQ - w_g h_g = 0 \]

**Momentum:**
\[
(P_A + w V) + (P_f A_f w \sin \beta + \Delta w_f V_f \cos \theta) - (P_g A_g + w_g V_g) + \int_{A} p_w dA_w - dF_f = 0
\]
Temperature

CPIPE performs a 1-dimensional analysis of the wall heating in a supersonic channel. Given the temperature distribution along the wall of the channel, CPIPE will calculate the heat flux. The equations used to calculate the heat flux are shown explicitly below:

Wall heat flux found using modified Reynolds Analogy:

\[ \dot{Q}_w = \int \frac{C_f}{2} \rho V (h_{aw} - h_w) \left( \frac{A_w}{L} \right)^{1/2} dA_w \]  

(1)

\[ C_f = \text{Van Driest skin friction coefficient found from iterative solution of:} \]

\[ \frac{0.242}{A^2} \frac{(\sin^2 \theta + \sin^2 \phi)}{\left( \frac{T_w}{T_{aw}} \right)} = 0.41 \log_{10} \left( \frac{Re_{aw}}{C_f} \right) \]

\[ - \left( 1 + \frac{2n}{2} \log_{10} \left( \frac{T_w}{T_{aw}} \right) \right) \]

\[ C_f = \frac{2A^2 + B}{(B^2 + 4A^2)^{1/2}} \quad \beta = \frac{B}{(B^2 + 4A^2)^{1/2}} \quad n = 0.76 \]

\[ A^2 = \left( \frac{\gamma - 1}{2} \right) \frac{m_a^2}{T_{aw}/T_a} \quad B = \left[ 1 + \left( \frac{\gamma - 1}{2} \right) m_b^2 \right] \frac{m_a^2}{(T_w/T_a)} \]

\[ Re_{aw} = \frac{\rho V L}{\mu} \]

Species Viscosity (Southerlands formula):

\[ \mu_i = \frac{C_i T_i^{1/2}}{T + C_i} \times 10^{a_i} \frac{16.5}{f_i^2} \quad \mu = \mu_i \left\{ \sum_{i=1}^{N} \left( \frac{x_i}{\mu_i} \right) \right\} \]

\[ \phi_{ij} = \frac{\left[ 1 + \left( \frac{M_i}{M_j} \right)^{1/2} \left( \frac{\mu_{ij}}{\mu_i} \right)^{1/2} \right]^{1/2}}{\gamma_{ij}^{1/2} (1 + \frac{M_i}{M_j})^{1/2}} \]

Species considered \( (H_2O, O_2, H_2, N_2) \)
Adiabatic Wall Enthalpy: 

\[ h_{aw} = h + (Pr)^{1/3} \frac{v^2}{2} \]

Where \( (Pr)^{1/3} = \frac{C_p \mu}{K} = \frac{4.8}{98.5} \)

ORIGINAL PAGE IS OF POOR QUALITY
**Geometry:**

- \( X_0 \) = beginning axial location (m)
- \( X_{mx} \) = end axial location (m)
- \( \Delta x \) = step size (m)
- \( XTRAN \) = X position where B.C transitions (m)
- \( WID \) = width of engine (m)
- \( \& AP \) = non-dimensionalising height of combustor = 1.0 (m)

**Distributions:**

- \( \{ T_A, X_A \} = \{ \text{flow area (m}^2\), axial location (m)\} \)
- \( \{ TWP, XWP \} = \{ \text{wetted perimeter (m), axial location (m)}\} \)

**Upstream Conditions** (1-D) 9

- \( P_0 \) = static pressure (\( \text{N/m}^2\))
- \( T_0 \) = static temp. (\( \text{K} \))
- \( M_0 \) = velocity (\( \text{m/s} \))

**Fuel Injection Parameters:** (specify for each injection pt.)

- \( \{ TPHE, XPHIE \} = \{ \text{injected equivalence ratio, axial location (m)\} \}
- \( \{ TPHE, XPHIE \} = \{ \text{reacted equivalence ratio, axial location (m)\} \}
- \( TTF \) = fuel stagnation temp (\( \text{K} \)) (1000)
- \( PTF \) = fuel stagnation press (\( \text{N/m}^2\)) (4758000)
- \( MF \) = fuel mach number (1.0)
- \( THETA \) = fuel injection angle (rad. to horizontal) (0.8)
- \( BETA \) = wall angle at injection (rad.) (0.0)
- \( KF \) = end of fuel inj. region (cm)
Temperature:

\{ T_w, T_{wx} \} = \{ Temperature \text{, wall (\%)} \text{, axial position (m)} \}

- $T_{tank} =$ Temp of fuel in tank (200K)

Options:

<table>
<thead>
<tr>
<th>Initial B.L. status:</th>
<th>0</th>
<th>Turbulent B.L. at $x_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>(ALAM1)</td>
<td>1</td>
<td>Laminar B.L. at $x_0$</td>
</tr>
<tr>
<td></td>
<td>-1</td>
<td>$C_f = q_w = 0$</td>
</tr>
</tbody>
</table>

- B.L. Transition cooling:

  (ALAMTRN)

<table>
<thead>
<tr>
<th>1</th>
<th>Laminar flow</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Turbulent flow</td>
</tr>
</tbody>
</table>

- Type of C.M. model:

  (EQUIL)

<table>
<thead>
<tr>
<th>1</th>
<th>Only mixing Limits Run</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Chemical Kinetics Modeled</td>
</tr>
</tbody>
</table>

ORIGINAL PAGE IS OF POOR QUALITY
APPENDIX B: HAND CALCULATIONS
Drag Calculation:

\[ D = \frac{1}{2} \rho_\infty V_a^2 C_D S_{ref} \]

\[ S_{ref} = 335 \text{ ft}^2 \quad \text{(based on \( H_2 \))} \]
\[ C_D = 0.05 \]
\[ \rho_\infty = \frac{1}{2} \rho_\infty V_a^2 \]

at \( M = 10 \) \( alt = 100,000 \text{ ft} \)
\[ \rho_\infty = 0.0010439 \frac{\text{lbm}}{\text{ft}^3} \]
\[ V_\infty = 991.68 \frac{\text{ft}}{\text{s}} \]
\[ V_\infty = 9916.8 \frac{\text{ft}}{\text{s}} \]

\[ \frac{\rho_\infty}{2} V_a^2 = 1588.7 \frac{\text{lbm}}{\text{ft}^2} \quad \text{(reasonably structurally)} \]

\[ D = \frac{1}{2} \rho_\infty C_D S_{ref} = 26,610.7 \text{ lbf} = \text{Drag} \]

Thrust Calculation:

* need to get newest estimates from SCRAM (Loro)

* Thrust equation: (momentum balance on e.w.)

\[ T = m_e \mu_e - m_i \mu_i + \rho_e A_e - \rho_i A_i \]
\[ m_e = \rho_e \mu_e A_e \]
\[ m_i = \rho_i \mu_i A_i \]

\[ T_{\text{effective}} = T - D_{\text{spill}} \]
\[ D_{\text{spill}} = \int p \, dA \]

103
Overall Geometry:

forebody:

<table>
<thead>
<tr>
<th>Forebody</th>
<th>( \frac{P_{c2}}{P_{a0}} )</th>
<th>( M_2 )</th>
<th>( \frac{P_2}{P_{a0}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1</td>
<td>0.7829</td>
<td>7.51</td>
<td>8.97</td>
</tr>
<tr>
<td>#3</td>
<td>0.8798</td>
<td>7.85</td>
<td>9.17</td>
</tr>
<tr>
<td>#11</td>
<td>0.7780</td>
<td>7.33</td>
<td>14.63</td>
</tr>
</tbody>
</table>

Design #1:

<table>
<thead>
<tr>
<th>Ramp angle</th>
<th>Location</th>
<th>5°</th>
<th>5°</th>
<th>3°</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>15'</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>24'</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Design #3:

<table>
<thead>
<tr>
<th>Ramp angle</th>
<th>Location</th>
<th>5°</th>
<th>5°</th>
<th>3°</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>18'</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27'</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

By moving ramps aft, (1) decrease in stagnation pressure losses (i.e. more efficient), (2) decreased magnitude of Mach decrease, (3) increased static pressure gain.

Design #11:

<table>
<thead>
<tr>
<th>Ramp angle</th>
<th>Location</th>
<th>5°</th>
<th>5°</th>
<th>3°</th>
<th>3°</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>18'</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>27'</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31'</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Additional ramp decreases much more than #1 (which simply moved ramps forward), (2) stagnation pressure losses are only slightly more than #1 (3) major increase in static pressure rise.
Final configuration => # 11
Values of properties at exit plane (36')

\[
\frac{P}{P_\infty} = 14.63 \quad \frac{P_e}{P_\infty} = 0.778 \quad (\text{mass averaged value for total pressure only})
\]

\[
\begin{align*}
T & = 21.1588 \quad T_\infty = 408^\circ R \\
\frac{T}{T_\infty} & = 2.1107 \\
\eta & = 7.4 \\
\end{align*}
\]

Inlet: Used Mack 7-4 inlet design (see attached sheets)

From Pete's sheet:

\[
\begin{align*}
A_e & = 0.133062 \quad \frac{P_e}{P_\infty} = 1.77570 \\
A_\infty & = 0.360845 \quad \frac{P_e}{P_\infty} = 21.1588 \quad m_e = 4.0 \\
\frac{L_{comb}}{A_\infty} & = 6.4
\end{align*}
\]

Setting \( A_e = 5\) inches = .42' \( \Rightarrow A_\infty = 3.13' \)

\( A_i = 1.13' \) (dist small lip from \( \Rightarrow L_{comb} = 20' \) (body)

Use exit conditions from forebody design:

\[
\begin{align*}
\frac{P_i}{P_\infty} & = \frac{14.63 (23.27 \frac{\text{lb}}{\text{ft}^2})}{340.4} = 3.40.4 \frac{\text{lb}}{\text{ft}^2} \\
\frac{P_e}{P_i} & = 21.1588 \quad \Rightarrow P_e = 340.4 (21.1588) = 7203.3 \frac{\text{lb}}{\text{ft}^2} \\
\frac{P_i}{P_\infty} & = 0.778 \quad \frac{P_e}{P_\infty} = 1.77570 \quad \Rightarrow \frac{P_i}{P_\infty} = 0.604 \\
\frac{T_i}{T_\infty} & = 2.1107 \times 408^\circ R = 861.16^\circ R
\end{align*}
\]

Since INLET does not estimate temp rise in inlet, we estimated it using oblique shock tables from: J.O. Anderson
Temperature Estimation in INLET:

1st 5° ramp: \( \theta = \) wedge angle = 5° \( \beta = \) wave angle = 12° 

\( \frac{P_2}{P_1} = 2.232 \quad M_{n_2} = .7157 \quad \frac{T_2}{T_1} = 1.294 \quad \frac{P_2}{P_1} = 1.793 \)

\[ M_2 = \frac{M_{n_2}}{\sin(\beta - \theta)} = 5.87 \]

\[ T_2 = \frac{T_2}{T_1} = 1114.3^\circ R \quad P_2 = \frac{P_2}{P_1} = 759.86 \text{ lb}_f/\text{ft}^2 \]

\[ P_2 = \frac{V_2}{\sqrt{T_1}P_1} = 0.010399 \text{ lb}_f/\text{ft}^3 \]

2nd 5° ramp: \( \theta = 5° \quad \beta = 13° \)

\[ M_{n_2} = M_2 \sin(\beta - \theta) = 0.32 \quad \frac{P_2}{P_2} = 1.866 \quad M_{n_3} = 0.776 \]

\[ \frac{T_2}{T_1} = 1.204 \quad \frac{P_2}{P_1} = 1.551 \quad M_3 = \frac{M_{n_3}}{\sin(\beta - \theta)} = 5.57 \]

\[ T_3 = 1341.7^\circ R \quad P_3 = 1417.9 \text{ lb}_f/\text{ft}^2 \]

Rest of inlet: We know that the rest of the inlet compresses the flow from 5.6 \( \rightarrow \) 4.0, approx. This by one more oblique shock.


\( M_2 = 4.0 \Rightarrow \) wedge angle \( \approx \) 16° 
\( M_1 = 5.6 \Rightarrow \) wave angle \( \approx \) 26°

\[ M_{n_11} = M_1 \sin 26^\circ = 2.45 \quad \frac{T_2}{T_1} = 2.088 \quad \frac{P_3}{P_2} = 3.273 \]

\[ P_3 = 0.0579 \text{ lb}_f/\text{ft}^2 \]

To check find \( P \) to combustor: \( \frac{P_3}{P_2} = 6.836 \)

\[ P_3 = 9692.8 \text{ lb}_f/\text{ft}^2 \Rightarrow \text{compare to } P = 7203 \text{ lb}_f/\text{ft}^2 \]

\( \Rightarrow \) Temp guess 13 not bad for rough estimate.
Summary INLET:

Exit Conditions:

\[ P_e = 7203.3 \text{ lb/ft}^2 \]
\[ P_{te} = 0.604 \]
\[ R_e = \frac{2801.5}{60} \text{ ft} \]
\[ M_e = 3.0 \]
\[ P_e = 0.579 \text{ lbm/ft}^3 \]
\[ A_e = 5 \text{ inches} = .42' \]

Use these as inputs for CPIPE.

Summary of CPIPE Outputs:

\[ M_e = 2.2986 \]
\[ P_e = 17380.0 \text{ lb/ft}^2 \]
\[ T_e = 5596.4 \text{ K} \]

Assumes no stagnation pressure loss across combustor: \[ P_{te} = 0.604 \]

\[ A_e = 6 \text{ inches} = .5' \]

Nozzle: Does not work (Seagull). If use ideal expansion (i.e., \( P_e = P_{te} = 23.27 \text{ lb/ft}^2 \)) should be able to find (from texts) the exit velocity. If ideally expanded, streamline defining \( A_e \) will extend straight from the exit of combustor (see diagram in Thrust Calc). Can find \( A_i \) by tracing the streamline from the cowl lip, maintaining the same distance from the forebody (you should not cross any shock waves).

For thrust eqn: \[ T = \dot{m}_e e - \dot{m}_i u_i + p_e A_e - p_i A_i \]
\[ \dot{m}_e = 1001.04 \text{ lbm/sec} \]
\[ u_i = 9916.8 \text{ ft/s} \]
\[ A_i = 4 \times 13' = 210' \]
\[ A_{cowl} = 21 - 12.3 = 8.7' \]
\[ p_i = 23.27 \text{ lb/ft}^2 = P_e \]
Mass removal required to completely remove bromine layer on 5000

consideration mass removal sufficient to completely remove displacement

thickness at 35':

\[ \delta = 0.136 \text{ ft} \]

\[ U_e = 9806 \text{ ft/sec} \]

\[ \rho_e = 0.0000744 \text{ slug/ft}^3 \]

The mass removal rate per unit depth is simply:

\[ \delta \rho_e U_e = 0.9992 \text{ slug/ft sec} \]

Over the width of the salinor, we have:

\[ \rho_e = 0.709 \text{ slug/sec} \]

Over the entire area, the velocity of the removed mass would need to be:

\[ V = \frac{\rho_e}{\rho_{\text{in}}} = \frac{0.708}{208} \approx 400 \text{ ft/sec} \]
Mass removed on 5000: boundary layer suction

Approximate the mass removed as, \( m = \frac{2}{3} \frac{[\text{slug/m}^2/\text{s}]}{\text{ft}^2} \)

\[
\begin{align*}
\theta & = 2 \\
0 & \text{ to } 30' \\
-0.005 & \text{ to } 33' - 40'
\end{align*}
\]

with a linear approximation between 30 - 33'.

Thus, in \( m \) per unit depth is

\[
\frac{d}{3} (-0.005)(3') + (-0.005)(7')
\]

\[
\Rightarrow m = 0.00425 \text{ slugs/ft sec ft}
\]

If suction is to be performed at this rate over the entire width of the forebody prior to the first inlet ramp angle, then

\[
\frac{d}{3} = 0.00425 \times w \quad w = \text{width of forebody } \leq 7'
\]

\[
\Rightarrow m = 0.00425 \times 7(0.00425) = \frac{d}{3}
\]

\[
\Rightarrow m_{001} = 0.0296 \text{ slugs/ft sec}
\]

\[
\Rightarrow v = \frac{m_{001}}{w} \frac{1}{p E} = 53.2 \text{ ft/sec}
\]

Original page is of poor quality.
COMPUTER CODES — 19.4 ft DOWNSPOUT HEAD 5.96

ASSUME POISON STANDEA CONSTANT

\[ P_{\text{exit}} = \left( \frac{17380 \left[ 1 + \frac{x-1}{3} (2.3)^2 \right]}{1 - \frac{x-1}{3} (5.96)^2} \right) = 141.74 \text{ lb/ft}^2 \]

WHERE \( M_e = 2.3 \) & \( P_e = 17380 \text{ lb/ft}^2 \) AT COMBUSTOR EXIT

CONSEQUENTLY, THE TOTAL TEMPERATURE ALSO STAYS
CONSTANT, HENCE FROM EUTROPIAN RELATIONSHIPS

\[ T_{\text{exit}} = \left[ \frac{5596.4 \cdot R}{140.2 (5.96)^2} \right] = 1415.6^\circ \]

\[ A = \sqrt{\frac{RT}{M_e}} = \left[ \frac{5.4 (1716) \cdot (1415.6)}{140.2 (5.96)^2} \right]^{1/2} = 1844 \text{ ft}^2/\text{sec} \]

\[ U = M_e A = 5.96 \times 1844 = 10990 \text{ ft}^2/\text{sec} \]

EFFECTIVE NOZZLE EXIT AREA IS DETERMINED
BY APPLYING THE CONTINUITY EQUATION.

MASS FLOW AT COMBUSTOR EXIT:

\[ m = \int U A = \left( \frac{17380}{1716 (5596.4)} \right) (2.9) \sqrt{1.4 (1716) (5596.4)} \left( 0.5 \right) \]

\[ = 7.729 \text{ slugs/ft}^2 \text{ ft}^2/\text{sec} \]

\[ A = \frac{m}{U} = \left( \frac{7.729 (1716) (1415.6)}{10990 \times 141.74} \right) = 12.05 \text{ ft}^2 \text{ per foot span} \]

FINALLY

\[ T = m_c U_e - m_o U_o + P e A - P_o \]

\[ = (7.729 x 1090) 00 (10) + 141.74 x 12.05 - 23.27 (7.13) \]

\[ - (7.19 (9916) 1.00 / 044) / 2 \]

\[ = 7646 \text{ lb per foot span} \]
References

1. Penland, J.A., Hallissy, J.B., and Dillon, J.L. "Aerodynamic Characteristics of a Hypersonic Research Airplane Concept having a 70 degree swept double-delta wing at mach numbers from 0.80 to 1.20, with summary of data from 0.20 to 6.0", NASA TP-1552, December 1979.


13. Penland, Jim A., Hallisey, James B., Piloon, James L., "Aerodynamic Characteristics of a Hypersonic Research Airplane Having a 70 Degree Swept Double Delta Wing at Mach Numbers From 0.8 to 12.0, With Summary of Data From 0.6 to 6.0", NASATP, 552, Langley Research Center, Hampton, Va, 1979.


THERMAL MANAGEMENT GROUP

Design of a Combined Impingement and Actively Cooled Panel Concept for a Hypersonic Drone

Group Members:
Virginia Koch
James Marcolesco
Gregory Noffz
Jerry Yen

Faculty Advisor:
Professor A.F. Mills
Abstract

The aim of this paper is to design and compare active cooling systems that met the requirements of a hypersonic drone. The design procedure involved the determination of a flight condition to be studied; justification of the use of an active regenerative cooling system; and, finally, the optimization of the design of such a system. Several computer codes were written to facilitate parametric studies of the design. Two designs are proposed: one utilizing the hydrogen fuel as a coolant, and the other utilizing ethylene glycol as the secondary fluid. Due to weight considerations, the hydrogen design was determined to be preferable.
# Table of Contents

1. List of symbols

2. Introduction

3. Determination of Flight Profile and Nose Geometry


5. Mission Selection

6. Analysis and Design of an Active Cooling System
   6.1 Impingement Cooling
   6.2 Panel Cooling
      6.2.1 Fin Analysis
      6.2.2 Convection
      6.2.3 Conduction

7. Results/Discussion
   7.1 Final Designs
   7.2 Comparison of Final Designs

8. Summary

9. Bibliography

10. Figures
Appendix 1: Sample Calculations
  Appendix 1.1: Stagnation Point Heating
  Appendix 1.2: Heating Rate Away from Stagnation Point
  Appendix 1.3: Heating Rates for Noses of Various Radii
  Appendix 1.4: Equilibrium Temperature Calculation
  Appendix 1.5: Preliminary Heat Exchanger Analysis

Appendix 2: Computer Codes
  Appendix 2.1: HETAQ
  Appendix 2.2: ACSES

Appendix 3: Design Results
  Appendix 3.1: Hydrogen Design
  Appendix 3.2: Ethylene Glycol Design

Appendix 4: Hydrogen Properties

Appendix 5: Hypersonic Cruise Vehicle Configuration
1 List of Symbols

a Speed of Sound

c Specific Heat (Btu/lb R)

g Acceleration due to Gravity

h Altitude (Ft.) or Enthalpy(ft$^2$/sec$^2$)

k Conductivity

q Total Heat Transfer Rate (Btu/ft$^2$)

t Time (Sec)

A Area (ft$^2$)

D Drag = $0.5\rho_\infty V^2 C_D A$

L Lift = $0.5\rho_\infty V^2 C_L A$

M Mach Number = $V/a$

P Pressure

R Radius

T Temperature

V Flight Velocity (ft/s)
SUBSCRIPTS
B  body
d  diameter
eq  equilibrium
s  stagnation
sh  shock value
SL  sea level value
w  wall value
\infty  free stream
\delta  inviscid edge of the boundary layer

SUPERSCRIPTS
-  mean value
.  derivative with respect to time

SPECIAL SYMBOLS AND CONSTANTS
\( g \)  gravitational constant at sea level
\( Le \)  Lewis Number = \( Pr/Sc \)
\( L/D \)  Ratio of lift to drag
\( Pr \)  Prandtl number
\( Re \)  Reynolds number
\( \rho_{sl} \)  sea level density
\( \sigma \)  Stefan-Boltzmann constant
GREEK SYMBOLS

$\epsilon$  Emissivity

$\mu$  Viscosity

$\rho$  Density
2 Introduction

The phenomenon of aerodynamic heating was first experienced on re-entry vehicles, where heating rates on the order of tens of BTU/ft²s were encountered for relatively short periods of time. In spite of these high heating rates, the short time interval that the vehicles were in the critical regime allowed the problem to be managed with passive systems, such as ablative surfaces or heat sink structures.

Air breathing vehicles, generally experiencing less intense heating rates, tend to be in the critical regimes for much longer periods of time. The aircraft skin does not possess sufficient thermal capacity to act as a heat sink. In addition, requirements for reusability and turn-around time rule out ablative surfaces.

With these considerations in mind, the aim of the Thermal Management Group was to design a hypersonic drone cooling system suitable for testing candidate National Aerospace Plane (NASP) cooling systems. The hypersonic drone vehicle configuration was the same as that studied by the 1987-88 UCLA Advanced Aeronautics Design Class. The analysis was focused upon the nose region of the drone since the computation of high stagnation point heating rates is simpler and better understood than the heating rates occurring within the engine.

Due to the relatively low heating rates experienced at the drone nose, the use of a heat pipe was unwarranted. Instead, the designed system was based on an actively cooled panel concept. Although the actual thermal design of the NASP nose region would probably make
use of the heat pipe concept, there will exist panels elsewhere that will require an active cooling system. The design presented here, then, is simply a test of that concept. It would be judicious to point out that this thermal design is by no means meant to be directly applicable to the nose of the NASP. Rather, it is the concept of actively cooled panels that is currently under study; the system design just happens to be located at the nose region of the hypersonic drone.

The design procedure involved the determination of a flight condition to be studied; justification of the use of an active regenerative cooling system; and finally, the analysis and optimization of the design of such a system. The aircraft configuration is presented first, followed by a detailed description of the design procedure and results.

3 Determination of Flight Profile and Nose Geometry

To obtain useful data from flight tests, a suitable flight profile was established. In general, the flight condition is governed by two basic parameters: altitude and Mach number. Decreasing altitude will increase air density, resulting in increased heating rates and higher equilibrium temperatures, particularly at high Mach numbers. Also, with an increase in Mach number, specific impulse (ISP) and engine efficiency tend to decrease. Thus, given a range of Mach numbers for scramjet operation (5-12) and a rough estimate of the fuel tank size (8 feet diameter, 40 feet long), an effort was made to determine a flight condition where active cooling was required (due to material limitations), while maximizing the flight time for data collection and experiments.
A code called Hypersonic Equilibrium Temperature and Heat Flux (HETAQ) [See Appendix 2.1] was created to develop a data base of flight conditions. HETAQ varied Mach numbers from 5 to 12, altitudes from 70,000 feet to 115,000 feet, and nose radii from 1 to 6 inches. Lees Approximate Method was used to determine stagnation point heating rates, and an energy balance of the convective heat transfer with radiative cooling produced the skin equilibrium temperature. A simple routine incorporating the thrust equation was used to determine the total flight time available given the geometry of the proposed fuel tank, Mach number, and altitude. The code calculated time to thermal equilibrium for aluminum and titanium noses at each given flight condition. This was necessary to prove that the time required for the craft to reach thermal equilibrium would be less than the cruise flight time.

Once the data base was developed from HETAQ, the group selected a flight condition suitable for the entire design team. Specifically, though a higher Mach number was desirable for the propulsion group, the resulting shorter flight time proved to be incompatible with the Thermal Management Group's aim of achieving steady-state conditions. From the HETAQ data, any speed faster than Mach 9 or 10 would result in the use of too much fuel and thus create undesirable heating rates. Therefore, the speed was set at Mach 10, and the altitude set at 100,000 feet. In addition, the nose radius was set at one inch, since a smaller nose radius results in a higher equilibrium temperature (the temperature that would arise without cooling). The higher temperature was desired to justify the use of an active cooling system. Indeed, the equilibrium temperature for such a configuration is $4000^\circ R$, high enough
to require an active cooling system.

In summary, with a flight profile of Mach 10 at 100,000 feet, and with a nose radius of one inch, results from HETAQ determined that the stagnation point and points aft required cooling.

4 Methods for Computing Aerodynamic Heating

In the hypersonic regime, the aircraft endures the most severe heating rates at the stagnation region of the body. A stagnation point is defined as the point on the body where a streamline terminates. The stagnation region differs from the rest of the flow field in that the boundary layer is at its thinnest. A large change in velocity near the stagnation point means a large change in kinetic energy. This lost kinetic energy is converted to thermal energy in the form of stagnation point heating. A thin boundary layer means the flow is at free stream conditions very close to the body. Heat transfer is thus at a maximum where the boundary layer thickness is at a minimum. The larger velocity gradient and the thin boundary layer combine to make the stagnation region the area of critical aerodynamic heating.

It is important to note that the boundary layer, defined as the region dominated by viscous effects, has a finite thickness at the stagnation point. Thus, the flow properties at the edge of the boundary layer, required by all of the methods discussed below, are the properties behind the bow shock, NOT the stagnation properties.

Heat transfer rates from flat plates subjected to convection are given by equations of
the form

\[ \dot{q}_s = h(T_\infty - T_w), \]  

(1)

where \( h \) is the convective heat transfer coefficient. This quantity depends on the Reynold’s Number

\[ Re = \frac{\rho v L}{\mu} \]

The length from the leading edge, \( L \), is zero at the stagnation point. Therefore, instead of relying on \( Re \) as a parameter, the methods discussed use the velocity gradient at the stagnation point.

A method used extensively is the Fay and Riddell Method. Developed in 1959 at Avco, it is accepted as the most accurate formula available, but is also one of the most difficult to use (See Ref. 11). The heat transfer rate is given by

\[ \dot{q}_s = \frac{1}{1020 Pr^{0.6} \sqrt{\alpha \rho \mu}} (\frac{\rho_w h_w}{\rho_s h_s})^{0.1} (\rho_s h_s V_\infty)^{0.5} (h_s - h_W) (\frac{dV_s}{V_\infty d\theta})^{0.5} [1 + (Le^m - 1) \frac{h_D}{h_s}] \]

(2)

where

\[ Le = \text{Lewis Number} = 1.4 \]

\[ m = 0.52 \text{ for Equilibrium Boundary Layer Flow} \]

\[ = 0.63 \text{ for Frozen Boundary Layer Flow} \]

It is apparent that eqn.(2) is quite complex, particularly since terms such as the enthalpies and the velocity gradient are themselves derived from long equations.
Lees Approximate Method, presented in 1958 (See Ref. 11), resulted in a reasonably accurate (+ or - 12 percent) value for stagnation point heat transfer rate.

\[ \dot{q}_s = \frac{1}{\sqrt{R_o}}(15.5)^{2.5n}(\rho_\infty)^{0.5}\left(\frac{V_\infty}{1000}\right)^3 \]  

For pencil and paper preliminary design, Lees Approximate Method provided the necessary accuracy with the minimum amount of computation and the least chance for error. Lees Approximate Method was also chosen as the standard method by Hankey (Ref. 11).

Fortunately, the group had access to MINIVER, a software tool that calculates stagnation point heating rates as well as heating rates at other regions. MINIVER utilizes the Fay-Riddell Method among others, depending on the region and flow regime. Between Fay-Riddell, Lees and MINIVER, the group was able to verify results quickly. This practice not only served to increase the group's confidence in the results but also to increase the group's understanding of the physical phenomenon.

So far, stagnation POINT heat transfer rates \( \left( \frac{Btu}{Ft\cdot Scw} \right) \) have been discussed. Before one can speak of heat loads \( \left( \frac{Btu}{Scw} \right) \), the heat transfer rates away from the stagnation point must be determined and the frontal area must be known. The heat transfer rate away from the stagnation point is given as a function of theta by

\[ \dot{q} = \dot{q}_s(COS\theta)^{3/2} \]
for \( \theta \) up to 60°. This correlation is accurate to within seven percent (See Ref.11). Since five degree half angle nose cones are considered appropriate for hypersonic aircraft (See Ref.4), \( \theta \) varies from 0 to 85 degrees. This range is beyond the sixty degree limit of accuracy, but is adequate for a first order approximation. Using the incremental area as shown Figure 6, the above equation is integrated over \( \theta \) to give heat transfer \( \left( \frac{Rtu}{sec} \right) \) through the nose. Recall that heat transfer rate PER AREA decreases with an increase in nose radius. However, total heat transfer increases with an increase in nose radius. This is due to the increase in frontal area with larger radii. An optimum would have to be found between the size of the nose radius and and the size of the conical portion of the nose. Correlations exist that give heat transfer rates for flat plates at angles of attack. Using the proper incremental area, these correlations can be integrated to find the total heat transfer to the conical portion of the nose.

5 Mission Selection

The first step involved the derivation of stagnation point heat transfer rates \( \left( \frac{Rtu}{ft^2sec} \right) \) using Lees Approximate Method for various flight conditions. Heat transfer rates were computed in terms of nose radius and wall enthalpy, \( h_w \), given by

\[
h_w = 0.234(T_w) + (10)^{-5}(T_w)^2
\]  

This allowed designers to vary the important parameters of nose radius and wall tem-
perature at a given flight condition and determine the heat transfer rate. The following is a list of some of the flight conditions examined. Stagnation point heat transfer is given by

\[ \dot{q}_s = \frac{X}{\sqrt{R_o}} \left( 1 - \frac{U_w}{Y} \right) \]

where \( X \) and \( Y \) are constants for that flight condition.

<table>
<thead>
<tr>
<th>MACH</th>
<th>ALTITUDE</th>
<th>X</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>50000</td>
<td>74.07</td>
<td>777.40</td>
</tr>
<tr>
<td></td>
<td>70000</td>
<td>49.02</td>
<td>782.70</td>
</tr>
<tr>
<td>8</td>
<td>50000</td>
<td>175.75</td>
<td>1307.02</td>
</tr>
<tr>
<td></td>
<td>70000</td>
<td>116.19</td>
<td>1315.11</td>
</tr>
<tr>
<td>9</td>
<td>100000</td>
<td>93.24</td>
<td>1704.68</td>
</tr>
<tr>
<td>10</td>
<td>50000</td>
<td>343.30</td>
<td>1987.30</td>
</tr>
<tr>
<td></td>
<td>70000</td>
<td>226.95</td>
<td>2000.45</td>
</tr>
</tbody>
</table>

A graphical summary of this data is given in Figure 2 for various nose radii and flight conditions. In addition, sample calculations can be found in Appendix 1.

At this point, the worst case must be examined. Provided there exists no active cooling scheme and the structure of the vehicle does not act as a heat sink, the final equilibrium temperature of the nose must be determined. Even if the heat from aerothermal heating is not bled off by a coolant or absorbed by the structure, energy is still lost to the surroundings via radiation. There exists some temperature such that the amount of heat coming in due to convection is exactly equal to the amount of heat leaving due to radiation (See Figure
3). This temperature is called the equilibrium temperature and can be solved for by using the following equation.

\[ q_s = \epsilon \sigma (T_{w,eq}^4 - T_\infty^4) \]  

(7)

\( T_\infty \) can be taken to be zero and as a first estimate, \( \epsilon \), the emissivity, can be taken to be 0.85. For the same flight conditions listed above and a nose radius of four feet, the corresponding equilibrium temperatures are given below.

<table>
<thead>
<tr>
<th>MACH</th>
<th>ALTITUDE [ft]</th>
<th>EQUILIBRIUM TEMP. [R]</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>50000</td>
<td>2225</td>
</tr>
<tr>
<td></td>
<td>70000</td>
<td>2090</td>
</tr>
<tr>
<td>8</td>
<td>50000</td>
<td>3025</td>
</tr>
<tr>
<td></td>
<td>70000</td>
<td>2810</td>
</tr>
<tr>
<td>10</td>
<td>50000</td>
<td>3777</td>
</tr>
<tr>
<td></td>
<td>70000</td>
<td>3525</td>
</tr>
</tbody>
</table>

Note that even with such a large and impractical nose radius, the equilibrium temperatures become too high for a conventional material such as aluminum. Therefore, an active cooling system must be designed that can maintain the nose at a reasonable temperature. Figure 4 shows equilibrium temperatures at 100,000 feet as a function of Mach number and nose radius. Note that at our designated flight condition of Mach 10, 100,000 feet the equilibrium wall temperature is about 4500\( R \).
The next parameter that must be examined is the time required for the nose to reach equilibrium, provided no heat is bled off by a coolant or absorbed by the structure. An approximate energy balance that gives an estimate of time required to reach equilibrium skin temperature is given by

\[ \rho C_p V(T_{\text{w,eq}} - T_{\text{initial}}) = \dot{q}_s A \Delta t - \sigma \epsilon A (T_{\text{w,eq}} - T_{\infty}) \Delta t \]  

(8)

Assuming an “eggshell” type structure of uniform thickness, times on the order of seconds are required to reach equilibrium (See Figure 5). Aluminum, 1/16” thick takes the longest to reach steady state. In reality, there would exist some support structure behind the shell (spars, ribs, etc.) that would act as a heat sink and lengthen time to equilibrium. However, the craft would have to be accelerated to its target flight condition. As it passes through the lower Mach numbers, it would experience aerothermal heating. Therefore, when the clock “starts”, \( T_{\text{initial}} \) is already greater than the temperature at launch. Despite these complications, it is obvious that a drone at the designated flight condition would easily reach thermal equilibrium within the estimated 5.32 minute burn time.

In summary,

1) There exists several methods, including the code HETAQ, that allow rapid and accurate calculation of stagnation point heat transfer rates.

2) The stagnation point is the area of critical aerodynamic heating.

3) Correlations for heat transfer rates can be integrated over frontal areas to determine
total heat transfer \( (\text{Btu/Sec}) \) for a reasonable first estimate.

4) Heat transfer rates and heat loads at hypersonic speeds are large enough to warrant design of an active cooling system for any vehicle that spends more than even a few seconds in these regimes.

5) Due to propulsion and drag considerations, Mach 10 and 100,000 feet is chosen as the target flight condition.

6) With an estimated flight time of 5.32 minutes at Mach 10, 100,000 feet, the nose of the aircraft should reach thermal equilibrium long before the hydrogen fuel runs out.
6 Analysis and Design of an Active Cooling System

High heating rates at the nose of the drone dictate that the coolant be routed from the tank to the stagnation region and then back along the cone before being routed to the scramjet (See Figure 7). The design is broken up into two parts; impingement cooling and panel cooling. Impingement cooling takes place in the manifold immediately behind the stagnation region where a single jet of coolant impinges on the inside surface of the nose and is then routed into numerous tubes. Panel cooling takes place after the coolant enters the tubes and is modelled by a fin analysis.

Both impingement cooling and panel cooling are modelled in the Actively Cooled Structure Evaluation System (ACSES), a FORTRAN computer program written by the authors.

6.1 Impingement Cooling

A method commonly used to cool the leading edge of a gas-turbine airfoil involves the impingement of a jet of coolant against the leading edge's inside surface. Impingement cooling can also be applied to the nose of the hypersonic research vehicle currently under study by the UCLA Advanced Aeronautics Design Class.

In Ref. 13, Livingood presents Nusselt number correlations determined experimentally for jet impingement upon a concave, semi-hemispherical surface. Since the nose of the hypersonic test vehicle was modelled as a semi-hemisphere, a subroutine program called upon by ACSES utilized these correlations in determining the final fluid bulk temperature
exiting the semi-hemisphere manifold.

The geometrical parameters of the manifold (see Figure 8), including nozzle diameter, $d$, nose radius, $R_n$, and distance between the nozzle and the nose inner surface $Z_n$, were input into the program. These parameters could be varied to optimize the manifold design. The fluid bulk temperature entering the manifold from the nozzle ($T_{b,\text{noz}}$), and the coolant mass flow rate through the nozzle, $MF_{R,\text{noz}}$, were also input and optimized. Finally, fluid properties for two different coolants were used. The program outputted the fluid bulk temperature exiting the impingement cooling apparatus, $T_{b,\text{out}}$. This $T_{b,\text{out}}$ value was then returned to the program as the initial coolant temperature entering the active cooling tubes extending aft of the nose.

The procedure used to find $T_{b,\text{out}}$ was an iterative one. First, an initial guess was made for $T_{b,\text{out}}$. Fluid properties were then calculated based on the film temperature $T_{film}$:

$$T_{film} = 0.5[T_{w,\text{max}} + 0.5(T_{b,\text{noz}} + T_{out})],$$

where $T_{w,\text{max}}$ was the maximum allowable wall temperature. For the design considered, the nose wall was assumed to be at $T_{w,\text{max}}$ for the maximum efficiency of the design. $T_{w,\text{max}}$ itself was considered a conservative value since it was taken as half of the material's melting temperature.

The average Nusselt number correlation, $Nu_{ave}$, for jet impingement on a concave semi-hemisphere, as presented by Livingood (Ref. 13), was used to determine an average heat transfer coefficient $h_{ave}$. The correlation and the resulting average heat transfer coefficient
are presented below.

\[ N_{uave} = 2.98Re^{0.585} \left( \frac{d}{D} \right)^{1.10} \left( \frac{Zn}{d} \right)^{-0.007}; \]  

\[ h_{ave} = \frac{N_{uave}k}{d} \]  

where,

- \( Re \) = Reynolds number based on nozzle diameter, \( d \)
- \( D = 2 R_n \) = nose diameter
- \( k \) = thermal conductivity of the coolant

Once the average heat transfer coefficient was obtained, the total heat transfer into the coolant was evaluated. The coolant heat transfer was equivalent to the aerodynamic stagnation point heating of the nose minus the radiative cooling. Thermal resistivity through the thin-skinned, highly conductive nose was considered negligible in this analysis. Thus, the coolant heat transfer was:

\[ q_{coolant} = q_{nose} - q_{radiation} \]  

where \( q_{nose} \) was a function of the stagnation heat flux as determined by Lees' approximate method of stagnation heating in Hankey's report (Ref. 11), and outlined in Appendix 1. Given the surface area of the nose, the temperature exiting the manifold could be easily determined:

\[ q_{coolant} = h_{ave}A_{surf}(T_{b, out} - T_{b, nor}) \]
or,

\[ T_{b,\text{out}} = \frac{q_{\text{coolant}}}{h_{\text{ave, surf}}} + T_{b,\text{noz}} \]  

(14)

This \( T_{b,\text{out}} \) was then compared with the initial guess, and the entire procedure was repeated until the values for \( T_{b,\text{out}} \) converged.

6.2 Panel Cooling

The coolant tubes were placed equidistant apart on the cone. The distance between the tubes \( L \), was therefore only a function of \( x \), the distance from the manifold. This symmetry of the geometry and the assumed symmetry of the aerodynamic heat load dictated that only one of the coolant panes need be considered. These panels were then divided into slices of width and analyzed accordingly.

![Diagram of panel cooling](image)

The analysis was divided into three modes of heat transfer: convection from the coolant to the inside wall of the tube; conduction from the inside wall to the outside wall of the tube;
and conduction from the base of the fin through the nose cone wall (which was modeled as a fin). Given an initial coolant fluid temperature, the goal of this analysis was to calculate the maximum temperature of the panel.

6.2.1 Fin Analysis  Due to symmetry, coolant temperature at a given x location was independent of the particular tube. This fact implied a temperature profile shown below, with \( \frac{dT}{dy} = 0 \) at the midpoint section of the two adjacent tubes.

The analysis was then simplified, in that only half of the panel need be modeled by the fin. The fin endpoints were the base of the tube and the midline of the two adjacent tubes. A detailed derivation follows.
\[ q(x) = \text{Heat flux} \]

\[ X_{\text{skin}} = \text{Thickness of skin.} \]

\[ T_o = \text{Base of "fin" temperature.} \]

\[ X_1, X_2 = \text{Station locations: } \Delta x = x_2 - x_1 \]

Performing an energy balance in the \( y \) direction:

- Assume Top surface adiabatic: Temperature and heat flux constant across width \( \Delta x \)

\[
q A_c|_y - q A_c|_{y+dy} + g_s P dy = 0 \tag{15}
\]

\[ A_c = \text{cross sectional area} = t \cdot \Delta x \]

\[ P = \text{exposed perimeter} = \Delta x \]

- Dividing by \( dy \) and using the definition of a derivative and Fourier's Law: \( q = -k \frac{\partial T}{\partial y} \)

the above equation becomes:

\[
k t \frac{\partial^2 T}{\partial y^2} + q_s(x) = 0 \tag{16}\]

- However, it was assumed the temperature did not vary across the width \( \Delta x \) so an ordinary differential equation was obtained (Eqn 17):

\[
k t \frac{d^2 T}{dy^2} + q_s(x) = 0 \tag{17}\]
• Boundary conditions:

\[
\frac{dT}{dy} = 0 \quad \ldots \quad y = \frac{L}{2} \\
T = T_o \quad \ldots \quad y = 0
\]  
(18)

(19)

• Integrating twice:

\[
T' = -\frac{q_s(x)}{2kt} y^2 + C_1 y + C_2
\]  
(20)

• Applying the boundary conditions, and solving for the constants, Eqn 20 becomes:

\[
T = \frac{q_s(x)}{2kt} (y^2 - yL) + T_o
\]  
(21)

Equation 21 applies anywhere along the fin; however, the temperature of interest was the midline temperature of the panel, which was the maximum temperature the panel of skin would reach: \(T = \frac{T_o}{2} = T_{\text{crit}}\)

\[
T_{\text{crit}} = q_s(x) \left(\frac{L^2}{8kt}\right) + T_o
\]  
(22)

\(T_o\), the temperature at the base of the fin which was the same as the temperature on the outside wall of the tube, was calculated from the conduction and convection analysis.

6.2.2 Convection The total heat that must be absorbed by the coolant fluid was equal to twice the amount of heat absorbed over the surface area of the fin, as heat was entering the tube from both sides

\[
Q = L(x)q_s \Delta x
\]  
(23)
The flow in the tube was turbulent with a high temperature gradient at the wall. Thus a convection correlation given by Incopera and DeWitt (Ref. 9) was used to best fit the conditions.

\[ Nu_d = 0.027 Re_d^{4/5} Pr^{1/3} \left( \frac{H}{\mu_s} \right)^{0.14} \]

\[ m_u = \text{dynamic viscosity at the mean temperature of the fluid} \]

\[ m_{u_s} = \text{dynamic viscosity at the wall temperature} \]

\[ Re_d = \frac{u_d}{\nu} \]

valid if:

\[ 0.7 \leq Pr \leq 16,700, \]

\[ Re_d \geq 10,000, \]

\[ 4d \geq 10, \]

- The heat transfer coefficient can be calculated from the Nusselt number:

\[ h = \frac{Nu_d k}{d} \quad (25) \]

- From Newton’s Law of Cooling, the temperature of the wall can be calculated:

\[ T_{wall} = T_m + \frac{Q}{hA_{xc}} \quad (26) \]

The properties were evaluated at \( T_m \), the bulk mean temperature of the fluid. \( T_m \) was calculated by using a heat balance. Knowing an initial \( T_{m,in} \), \( T(m, out) \) can be calculated:

\[ T_{m,out} = \frac{Q}{mC_p} + T_{m,in} \quad (27) \]
Therefore, the temperature of the inside wall was now known and could be used in the conduction analysis to calculate the temperature on the outside of the wall of the tube, or $T_o$, as it was referred to in the fin analysis.

6.2.3 Conduction

The equation for conduction heat transfer through a plane was used to calculate the outside wall temperature, $T_o$. The heat was assumed to flow through the bottom of the tube only. A fin analysis of the circumference of the tube demonstrated a rapid decrease in wall temperature after approximately 30 percent of the diameter. Because the tube was welded to the nose wall, additional heat transfer would occur. Thus, 50 percent
of the surface area was chosen as the effective area of conduction.

\[ A_{\text{cond}} = \frac{d\pi \Delta z}{2} \]  

(28)

Treating the tube as a flat plate of thickness \( x_{\text{tube}} \), and thermal conductivity \( k_{\text{tube}} \), the heat rate is:

\[ Q = \frac{k_{\text{tube}}}{x_{\text{tube}}} A_{\text{cond}} (T_o - T_{\text{wall}}) \]  

(29)

This equation can be solved for \( T_o \)

\[ T_o = \frac{Q x_{\text{tube}}}{k_{\text{tube}} A_{\text{cond}}} + T_{\text{wall}} \]  

(30)

The conduction heat transfer and the convection heat transfer equations were combined into a thermal resistance network. Conservation of energy implied the heat rates, \( Q \), used in equations (26) and (29) must be equal.

\[ R_1 = \frac{x_{\text{tube}}}{k_{\text{tube}} A_{\text{cond}}} \]  

(31)
Finally, $T_o$ can be calculated from eqn. (33) and $T_{crit}$ from eqn. (22). The objective of the design was to keep $T_{crit}$ less than the maximum allowable temperature for the nose cone material, $T_{max}$. 

$$R_2 = \frac{1}{hA_x}$$

$$T_o = Q\left(\frac{1}{hA_{cross}} + \frac{x_{tube}}{k_{tube}A_{cond}}\right) + T_m$$
Two design configurations were evaluated. One utilized liquid hydrogen as the coolant and the other a 30 percent aqueous glycol solution. The hydrogen configuration was composed of supercritical hydrogen flowing directly from a highly pressurized fuel tank to the active cooling system design. The hydrogen was then diverted back to the scramjet engine. The ethylene glycol design consisted of glycol circulating through a closed loop system including a pump, a heat exchanger, and the active cooling system design.

Both the designs were formulated with three common constraints in mind: the critical temperature was to remain less than the maximum allowable material temperature; the coolant velocity was to be less than the fluid speed of sound; and the pressure loss was to be kept as small as possible. A description of the specific constraints applied to each design is presented below.

The critical wall temperature for both designs was set at 1000 K. This corresponded to half the melting temperature of the wall material, molybdenum. This constraint was the most critical for both designs.

In the hydrogen design, the constraint placed upon fluid velocity was the most restrictive. The highest value of fluid velocity occurred in the nozzle entering the manifold. The fluid temperature in the nozzle was assumed to be equal to 20 K. The speed of sound of supercritical hydrogen was then found to be 1675 m/s (Appendix 4). The pressure of the hydrogen stored in the fuel tank was assumed to be approximately 3000 psi. It was decided to set the maximum allowable pressure to twenty-five percent or 750 psi.

Similarly, the glycol design constraints were determined. The speed of sound was 1644 m/s (Ref. 6), and the maximum pressure loss was assumed to be about 700 psi, to limit the size of the pump required. An additional constraint placed on the glycol design involved the temperature range of the coolant. The freezing point of a 30 percent aqueous
glycol solution is 257 K; this temperature was chosen as the minimum coolant temperature. The maximum coolant temperature was chosen to be 420 K (Ref. 6).

Once given the design constraints, various parametric studies were performed and several observations were made. In order to decrease the pressure loss, large tubes or smaller coolant mass flow rates were needed. However, with a smaller mass flow rate, the nose temperature increased and more tube branching was required. The larger the tube size, the less efficient the heat transfer in the tube. Hence the critical wall temperature increased with increasing tube size.

7.1 Final Designs

Liquid Hydrogen Coolant Final Design:

After parametrically varying mass flow rate and tube geometry, the liquid hydrogen design converged upon the following configuration (see Appendix 3.1):

- mass flow rate: 3.5 kg/s
- Tube diameter: 4.88 mm
- wall thickness: 2 mm
- initial number of tubes: 24
- Branching locations: 0.225 m, 0.875 m, 2.1 m, 3.55 m

Coolant velocities are kept well within the sonic range in both the impingement nozzle (806 m/s) and the tubes (281 m/s). System pressure loss is also kept to a minimum of about 150 psi. Outlet coolant temperature is 703 K. Therefore, all of the previously mentioned constraints were satisfied.

Increasing tube size would lower the pressure drop even more for the same mass flow rate. However, the fewer number of tubes allowed would result in the critical temperature being exceeded. Hence, 4.88 mm diameter tubes are the largest standard tubing that can be used with out fear of the craft exceeding the critical temperature.
30 Percent Aqueous Glycol Solution Final Design:

The critical aspect about the glycol coolant design was minimizing temperatures near the stagnation region. This was accomplished by using small tube sizes and as many tubes as allowed by the geometry. Since the density of this coolant is much larger than that of liquid hydrogen, and since the glycol flows in a closed system with a heat exchanger, high mass flow rates could also be used to minimize initial wall temperatures. This, however, created large pressure losses which would have to be overcome with a powerful pumping system.

The following configuration was determined to best satisfy the design constraints on wall critical temperature and coolant pressure loss (see Appendix 3.2):

- mass flow rate: 150 kg/s
- Tube diameter: 6.5mm
- wall thickness: 2.5mm
- initial number of tubes: 20
- Branching locations: 0.1m, 0.75m, 1.9m, 4.35m

The velocity of the coolant remains within the sonic limit both in the impingement nozzle (953 m/s) and the tubes (224 m/s). The outlet temperature of the glycol solution is 418 K. The pressure loss in the system is relatively high (1422 psi). Therefore, all of the constraints were satisfied save that of pressure loss.

This high pressure loss was found to be unavoidable. In order to keep the critical wall temperature below 1000 K at all stations, a high mass flow rate and a thick skin were required. Since weight is a critical issue in any aircraft design, the 2.5 mm wall thickness was judged to be as thick as desired. Taking all of the above into consideration, a compromise on the pressure loss was required. This resulted in a pressure loss 722 psi greater than the imposed design constraint.
Because the ethylene glycol design required a heat exchanger, a preliminary heat exchanger analysis was performed in which heat transfer coefficients of liquid hydrogen and glycol were compared. Given a tubular counter-flow heat exchanger, calculations were performed (listed in Appendix 1.5), and it was determined that the heat transfer coefficient for the liquid hydrogen was only about two to three times smaller than that of the ethylene glycol, given similar conditions. Thus, a configuration of bare concentric tubes could be used without fins on the inner tube extending out into the liquid hydrogen. If the proposed ethylene glycol design was chosen as the preferable design, the next step then would be to utilize software capable of designing such a heat exchanger.

7.2 Comparison of the Two Designs

The liquid hydrogen design met all of the design constraints with a satisfactory margin of safety, whereas the ethylene glycol design failed to meet the pressure constraint, and met the temperature constraint with a very low margin of safety. In addition, the ethylene glycol design required a heat exchanger and a large pump, as well as a greater skin thickness, all of which add considerably to the weight of the drone vehicle. Therefore, the liquid hydrogen design was deemed the better of the two.

One concern about the liquid hydrogen design is safety. In particular, a leak may prove to be disastrous for such a highly pressurized system of hydrogen. Because this will be placed on an unmanned drone vehicle, weight considerations dictated this small compromise in safety. In application to the NASP, however, appropriate safety measures could be taken to insure against such an occurrence.
8 Summary

Given an existing configuration, the authors determined a suitable flight condition for a hypersonic drone that would require some form of active cooling scheme and provide the propulsion group with an adequate challenge. The flight condition chosen was Mach 10 at 100,000 feet. The authors wrote a FORTRAN program called HETAQ (Hypersonic Equilibrium Temperature and Heat Flux) that parametrically varied Mach and altitude while computing equilibrium temperatures, heat fluxes and flight times for each flight condition.

Heating rates for regions aft of the stagnation point were derived for the chosen flight condition. An active cooling configuration was determined and consisted of coolant impinging on the inside of the skin at the stagnation region and then being routed through numerous tubes running back along the inside of the nose cone.

The authors wrote ACSES (Actively Cooled Structure Evaluation System), a FORTRAN program that allows the user to vary tube and manifold geometry, mass flow rate, and coolant to observe what effect these parameters have on critical temperature, coolant temperature, and pressure loss.

A design for liquid hydrogen coolant was formulated that optimized pressure loss and mass flow rate, given imposed design constraints. A design for ethylene glycol was also formulated. The hydrogen design was deemed the better of the two for the drone vehicle, based on weight considerations.
9 The Bibliography

References


17. Penland, et al. "Aerodynamic Characteristics of a Hypersonic Research Airplane Con-
cept Having a 70 Degree Swept Double-Delta Wing at Mach Numbers from 0.80 to 1.20 with Summary of Data from 0.2 to 6.0." NASA Langley Research Center, VA. NASA TP-1552 (1979).


10 Figures
Flight Time varies from 55.42 sec. for Mach = 1.00 to 13.76 sec. for Mach = 12.00.

Altitude = 12,000 feet.

K = normalizing constant
L = length
W = weight
R = rate
V = velocity

Fig 2
**Skin Equilibrium Temperature Calculation**

Energy balance on skin assuming no heat transfer through back-face:

\[
\text{convective heat transfer} = \text{radiative heat transfer}
\]

\[
q_s A = q_r A
\]

\[
q_s A = \varepsilon \sigma A \left( T_{v,eq}^4 - T_\infty^4 \right)
\]

- \(q_s\) = heat flux at stagnation point from Lees approximate method
- \(A\) = area
- \(\varepsilon\) = emissivity of material
- \(\sigma\) = Boltzmann constant
- \(T_{v,eq}\) = equilibrium wall temperature
- \(T_\infty\) = ambient air temperature

\(T_{v,eq}\) was calculated as a function of altitude, Mach number and nose radius to obtain a data base

*Fig 3*
Flight time varies from 2.75 min for Mach = 8 to 0.68 min for Mach = 12. Vol. fuel = 2010.61 ft³. Drag Coefficient = 0.20

Altitude = 100,000 feet

R = nose radius (in.)
R = 6, R = 5, R = 4, R = 3, R = 2, R = 1

This condition is possibly the best that can be achieved using a drone. However, the NASA encounters severe heating conditions.

Fig 4
Time to Skin Equilibrium
Temperature Calculation

Approximate energy balance to give an estimate of the flight time required to achieve the equilibrium skin temp.

\[ \rho C V(T_{v,eq} - T_{initial}) = \]

\[ q_s A \Delta t - \sigma \varepsilon A (T_{v,eq} - T_\infty) \Delta t \]

Some sample materials
at 100,000 ft; Mach 9; radius = 1"
flight time = 1.8 min

<table>
<thead>
<tr>
<th></th>
<th>Skin Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1 mm</td>
</tr>
<tr>
<td>Aluminum:</td>
<td>3.8 sec</td>
</tr>
<tr>
<td>Titanium:</td>
<td>2.7 sec</td>
</tr>
</tbody>
</table>

\[ q_s \rightarrow \text{skin thickness} \]
\[ q_r \leftarrow \text{no heat sink} \]

\[ q_s = q_r \]

Fig 5
Heating Rates For Noses of Various Radii:

\[ 0 \leq \theta \leq 80^\circ \]

\[ \dot{q} = \dot{q}_s (\cos \theta)^{3/2} \]

\[ q = q_s (\text{Area}) \]

Incremental Area \( dA = 2\pi R_0 \sin \theta \, d\theta \)

\[ \dot{q}_{\text{nose}} = \int_0^{80^\circ} \dot{q}_s (\cos \theta)^{3/2} \cdot 2\pi R_0 \sin \theta \, d\theta \]

\[ \dot{q}_{\text{nose}} = \dot{q}_s \cdot R_0 \cdot (2.4816) \]

\[ q \text{ (\emph{W} per sec)} \]

\[ \begin{array}{c}
300 \\
200 \\
100 \\
\end{array} \]

\[ R_0 \text{ (in.)} \]

\[ m = 12.90 \text{K} \]

\[ m = 10.70 \text{K} \]

\[ m = 10.90 \text{K} \]

\[ \text{Fig. 6} \]

ORIGINAL PAGE IS OF POOR QUALITY
Combined Impingement and Actively Cooled Panel Concept for the Drone Nose:

- Side view
  - $T_{in}$
  - Impingement Cooling
  - Manifold
  - Small tubes
  - $q(x)$
  - $LH_2$ to scramjet
  - $LH_2$ from tank

- End view
  - $T_{max}$ due to "fin" effect

* Maximum skin temperature is midway between coolant tubes
* Tubes welded to skin chosen to simplify fabrication
Impingement cooling at nose

\[ \text{Average heat transfer coefficient, } \bar{h} = \frac{\bar{Nu} \times k}{d} \]

\[ (\text{Tin} - \text{Tnoz}) = \frac{\dot{q}_{\text{nose}} - \dot{q}_{\text{rad}}}{\text{area}} \times \frac{1}{\bar{h}} \]

- Livingood's correlation for \( \bar{Nu} = f(\text{Re}, \text{geometry}) \)

\[ \bar{Nu} = 2.98 \text{(Re)}^{0.585} \left( \frac{d}{D} \right)^{1.10} \left( \frac{\text{zn}}{d} \right)^{-0.007} \]

- \( D \) = hemisphere diameter
- \( \text{zn} \) = nozzle exit-to-target separation
- \( d \) = nozzle diameter

Fig 8
Appendix 1: Sample calculations
Appendix 1.1: Stagnation Point Heating

(Lees' Approximate Method)
\[ V_0 = 6 \left( 9.76 \right) = 58.56 \text{ ft/sec} \]

\[ E_0 = E_{sl} \exp \left( -h/23700 \right) = 2.378 \times 10^{-4} \exp \left( -20000/2370 \right) \]

\[ = 1.24 \times 10^{-4} \text{ sec}^2/\text{ft} \]

\[ h_s = h_c = 0.25 T_0 + \frac{V_0^2}{50100} \]

\[ = 0.25 \left( 460 - 67 \right) + \frac{5856^2}{50100} \]

\[ = 782.73 \text{ ft} \text{/sec} \]

\[ \dot{g} = 21.92 \left( e_{x0}^{5}\right) \frac{1}{\sqrt{R_0}} \left( \frac{V_0}{1000} \right)^3 \left( 1 + \frac{h_u}{h_s} \right) \]

\[ \dot{g} = 21.92 \left( 1.24 \times 10^{-4} \right)^5 \frac{1}{\sqrt{R_0}} \left( \frac{5856}{1000} \right)^3 \left( 1 + \frac{h_u}{782.73} \right) \]

\[ \dot{g} = 49.02 \frac{1}{\sqrt{R_0}} \left( 1 + \frac{h_u}{782.73} \right) \text{ ft/sec} \]

**Equilibrium Temp for** \( R_0 = 4 \)

\[ h_u = 0.234 T_w + 1 \times 10^{-5} T_w^2 \]

\[ \delta\text{onc} = \delta\text{red} \]

\[ 24.5 = 0.03131 h_u + 0.85 \left( 4.81 \times 10^{-5} \right) T_w^4 \]

\[ 24.5 = 0.007327 T_w + 3.13 \times 10^{-5} T_w^4 + 4.0885 \times 10^{-13} T_w^4 \]

Say \( T_w = 1500^\circ R \)

24.5 \( \approx 13.76 \)

Say \( T_w = 2000^\circ R \)

24.5 \( \approx 22.95 \)

Say \( T_w = 2500^\circ R \)

24.5 \( \approx 23.55 \)

Say \( T_w = 2100^\circ R \)

24.5 \( \approx 24.71 \)

**\( T_w \) @ Equilibrium \( \approx 2090^\circ R \)**
\[ V_0 = 8 \text{ (976) } = 7808 \text{ ft/sec} \]

\[ C_{\infty} = C_{SL} \exp(-4/33700) \]
\[ = (2 \times 10^{-5} \text{ lb sec}^3/\text{ft}^4) \exp(-70000/23700) \]
\[ = 1.24 \times 10^{-4} \text{ lb sec}^3/\text{ft}^4 \]

\[ \dot{q} = 21.92 \sqrt{C_{\infty}} \cdot \left( \frac{V_0}{1000} \right)^3 \left( 1 - \frac{h_w}{h_s} \right) \]
\[ = 21.92 \sqrt{1.24 \times 10^{-4}} \cdot \left( \frac{7808}{1000} \right)^3 \left( 1 - \frac{h_w}{h_s} \right) \]

\[ h_s = h_e = 0.25 T_w + \frac{V_0^2}{50000} \]
\[ h_s = 0.25(460 - 62) + \frac{7808^2}{50000} \]
\[ h_s = 1315.11 \text{ ft} \]

\[ \dot{q} = \frac{116.19}{\sqrt{R_o}} \left( 1 - \frac{h_w}{1315.11} \right) \text{ Btu/ft}^2 \text{ sec} \]

**Equilibrium Well Temp**

\[ \dot{q}_{ent} = \dot{q}_{rad} \quad \text{for } R_o = 4 \]

\[ 58.09 = 0.04417 h_w + 0.25 \left( 4.81 \times 10^{-12} \right) T_w^4 \quad h_w = 0.234 T_w + 110^\circ F \]

\[ 58.09 = 0.010337 T_w + 4.417 \times 10^{-7} T_w^3 + 4.0885 \times 10^{-4} T_w^4 \]

Say \( T_w = 2000^\circ R \quad 58.09 \approx 28.98 \)

Say \( T_w = 2800^\circ R \quad 58.09 \approx 57.53 \)

\[ T_w \approx 2810^\circ R \]
\[ M = 10, \quad A/\rho = 70 \text{k} \] (Lee's Approx)

\[ V_\infty = 10 (976) = 9760 \text{ ft/s} \]

\[ E_\infty = \left(2.378 \times 10^{-3} \text{ lb sec}^2/\text{ft}^4\right) \exp \left(-70000/23700\right) \]

\[ = 1.24 \times 10^{-9} \text{ lb sec}^2/\text{ft}^4 \]

\[ \dot{q} = \frac{21.92 \left(1.24 \times 10^{-9}\right)^{5/4} \left(9760\right)^3 \left(1 - \frac{h_w}{h_s}\right)}{\sqrt{R_o}} \]

\[ h_s = 0.25 T_\infty + U_{\infty}^2/50100 \]

\[ = 0.25 (960 - 6) + 9760^2/50100 \]

\[ = 2000 \text{ Btu/lb} \]

\[ \dot{q} = \frac{226.95}{\sqrt{R_o}} \left(1 - \frac{h_w}{2000}\right) \]

Equilibrium Temperature

\[ \dot{q}_{\text{conv}} = \dot{q}_{\text{rad}} \]

\[ R = 4.16 \]

\[ \frac{226.95}{\sqrt{4.16}} \left(1 - \frac{h_w}{2000}\right) = 0.85 \left(4.81 \times 10^{-14}\right) T_w^4 \]

\[ 113.47 = 0.05673 h_w + 4.0885 \times 10^{-13} T_w^4 \]

\[ 113.47 = 0.01327 T_w + 5.673 \times 10^{-7} T_w^2 + 4.0885 \times 10^{-10} T_w^4 \]

Say \[ T_w = 2000^\circ \text{R} \]

\[ 113.47 \neq 35.35 \]

Say \[ T_w = 3000^\circ \text{R} \]

\[ 113.47 \neq 78.03 \]

Say \[ T_w = 4500^\circ \text{R} \]

\[ 113.47 \neq 238.8 \]

Say \[ T_w = 3560^\circ \text{R} \]

\[ 113 \approx 108 \]

Say \[ T_w = 3550^\circ \text{R} \]

\[ 113 \approx 119 \]

Equilibrium:

\[ T_{\text{will}} \approx 3525^\circ \text{R} \]
\[
L = \sqrt{\frac{\mu R}{T}} = \sqrt{0.4 \left(173.1 \cdot \left(460 - 69.7\right)\right)} = 973 \text{ ft/sec}
\]

\[
V_{\infty} = 6(973) = 5836 \text{ ft/sec}
\]

\[
E_0 = C_s \exp \left(-\frac{h}{2c_{v0}}\right) = 2.378 \times 10^{-2} \exp\left(-\frac{50000}{23700}\right)
\]

\[
= 2.89 \times 10^{-4} \text{ lb sec}^2/\text{ft}^2
\]

\[
h_s = h_c = 0.25 T_w + \frac{V_{\infty}^2}{50100}
\]

\[
= 0.25(460 - 69.7) + \frac{5836^2}{50100}
\]

\[
= 777.4 \text{ Btu/h ft}
\]

\[
\dot{q} = 21.92 \frac{1}{\sqrt{R_o}} \cdot (E_0)^5 \cdot \left(\frac{V_{\infty}}{1000}\right)^3 \left(1 - \frac{h_w}{h_s}\right)
\]

\[
\dot{q} = 21.92 \frac{1}{\sqrt{R_o}} \cdot (2.89 \times 10^{-4})^5 \cdot \left(\frac{5836}{1000}\right)^3 \left(1 - \frac{h_w}{777.4}\right)
\]

\[
\dot{q} = \frac{74.07}{\sqrt{R_o}} \left(1 - \frac{h_w}{777.4}\right) \text{ Btu/h ft}^2 \text{ sec}
\]

**Equilibrium Wall Temp**

\[
h_w = 0.234 T_w + 1 \times 10^{-5} T_w^2
\]

\[
R_o = 4
\]

\[
37.03 = 0.047638 h_w + 0.85 \left(4.81 \times 10^{-4}\right) T_w^4
\]

\[
37.03 = 0.01119 T_w + 4.764 \times 10^{-2} T_w^2 + 4.0885 \times 10^{-13} T_w^4
\]

Say \( T_w = 2000 \text{ °F} \) \( \Rightarrow 37.03 \neq 30.72 \)

Say \( T_w = 2300 \text{ °F} \) \( \Rightarrow 37.03 \neq 39.58 \)

Say \( T_w = 2250 \text{ °F} \) \( \Rightarrow 37.03 \approx 37.95 \)

\[
T_w \approx 2225 \text{ °F}
\]
\[ M = 8 \quad A_{1+} = 50000 \quad A \]

\[ v_o = 5 \times 9.73 = 77.84 \quad \text{ft/sec} \]

\[ e_o = 2.84 \times 10^{-4} \]

\[ h_s = 0.25 T_w + \frac{v_o^2}{50} \]

\[ = 0.25 \left( 460 - 69.7 \right) + \frac{77.84^2}{50000} \]

\[ = 1307 \quad \text{Btu/hr} \]

\[ q = 21.92 \cdot e_o^{-0.5} \cdot \frac{v_o}{R_o^{-3}} \cdot \left( \frac{v_o}{1000} \right)^3 \left( 1 - \frac{h_w}{h_s} \right) \]

\[ q = 21.92 \left( 2.84 \times 10^{-4} \right)^{-0.5} \frac{1}{R_o^{-3}} \left( \frac{77.84}{1000} \right)^3 \left( 1 - \frac{1307}{1307} \right) \]

\[ q = \frac{175.75}{R_o^{-3}} \left( 1 - \frac{h_w}{1307} \right) \quad B_{+}^{+0.5} \cdot \text{sec} \]

**Equilibrium Wall Temp for \( R_o = 4 \)**

\[ h_w = 0.234 T_w + 1 \times 10^{-5} T_w^2 \]

87.87 = 0.06723 \( h_w \) + 0.85 (4.01 \times 10^{-12}) \( T_w^2 \)

87.87 = 0.01573 \( T_w \) + 6.723 \times 10^{-7} \( T_w^2 \) + 4.0885 \times 10^{-15} \( T_w^4 \)

Say \( T_w = 3000^\circ \text{R} \) \( \approx 87.87 \neq 85.27 \)

Say \( T_w = 3050^\circ \text{R} \) \( \approx 87.87 \neq 89.61 \)

\[ T_w \approx 3025^\circ \text{R} \]

*ORIGINAL PAGE IS OF POOR QUALITY*
\[ V_0 = 9730 \]

ORIGINAL RACE IS
OF POOR QUALITY
\[ h_s = 0.25 \left( 460 - 64.7 \right) + 9730^2 / 50100 \]
\[ = 1987.3 \quad \text{B}+\text{u}/\text{b} \]

\[ q = 21.92 \left( 2.89 \times 10^{-4} \right)^{5/3} \frac{1}{\sqrt{R_0}} \left( \frac{9730}{1000} \right)^3 \left( 1 - \frac{h_s}{1987.3} \right) \]

\[ q = \frac{343.3}{\sqrt{R_0}} \left( 1 - \frac{h_s}{1987.3} \right) B^{+u/2} \text{sec} \]

**Equilibrium Temp**

\[ R = 4.44 \]

\[ h_s = 0.234 T_w + 1 \times 10^{-5} T_w^3 \]

\[ q_{\text{conv}} = q_{\text{rad}} \]

\[ 171.63 = 0.0202 T_w + (8.6363 \times 10^{-7}) T_w^2 + (4.0885 \times 10^{-13}) T_w^4 \]

Say \( T_w = 3000^\circ \text{R} \)
\[ 171.63 \approx 101.48 \]

Say \( T_w = 4000^\circ \text{R} \)
\[ 171.63 \approx 199.3 \]

Say \( T_w = 3600^\circ \text{R} \)
\[ 171.63 \approx 152.6 \]

Say \( T_w = 3800^\circ \text{R} \)
\[ 171.63 \approx 174.98 \]

Say \( T_w = 3780^\circ \text{R} \)
\[ 171.63 \approx 172.1 \]

\[ T_w \approx 3777^\circ \text{R} \]
\[ T \text{ in } \degree F = -51.098 \]
\[ \rho = 0.0109971 \text{ atm} \]
\[ \text{Density} \quad \rho_{\text{atm}} = 0.13960 \]
\[ \sigma = \sqrt{8RT} = \sqrt{1.4(1731.3)(460 - 51.098)} \]
\[ = 995 \text{ ft/sec} \]
\[ V_0 = 9(995) = 8960 \text{ ft/sec} \]
\[ C_0 = C_s \exp \left( -\frac{1}{23700} \right) = 2.378 \times 10^{-3} \exp \left( \frac{-100000}{23700} \right) \]
\[ = 3.497 \times 10^{-5} \text{ in sec}^2 \]
\[ h_0 = h_0 = 0.25 \frac{V_0}{T_0} + \frac{V_0^2}{50100} \]
\[ = 0.25(409) + 8960^2/50100 \]
\[ = 1704.677 \]
\[ \bar{g} = 21.92 \frac{1}{\sqrt{h_0}} \cdot e^{-5} \cdot \left( \frac{V_0}{1000} \right)^2 \left(1 - \frac{h_0}{h_5}\right) \]
\[ = \frac{21.92}{\sqrt{1704.677}} \cdot \left(3.497 \times 10^{-5}\right)^{-5} \cdot 719.323 \left(1 - \frac{1704.677}{h_5}\right) \]
\[ \bar{g} = 93.242 \left(1 - \frac{h_5}{1704.677}\right) \]
Appendix 1.2: Heating Rate Away From Stagnation Point
Calculations For Heating Rate Away

From Stagnation Point:

\[ q = \dot{q}_s (\cos \theta)^{3/2} \quad \text{for } \theta \uparrow \text{ to } 60^\circ \]

\( (7\% \text{ accuracy}) \)

\[
\theta
\]

For this aircraft, nose will be a 10° half angle cone. Therefore, \( \theta \) will vary between 0 and 80°. Use the above formula anyway.

\[ M = 10 \quad 70 \text{K} \quad T_{\text{eqn}} = 3525 \text{°K} \]

Lee's Method yields

\[ \dot{q}_s = \frac{226.95}{\sqrt{R_0}} \left(1 - \frac{hw}{2000}\right) \]

Melting Point of Steel: 1810°K = 3258°R

\( \Rightarrow \) Assume Wall Temp is \( = 1200\text{°R} \)

\[ hw = 0.234 Tw + 1 \times 10^{-5} Tw^2 \]

\[ = 0.234(1200) + 1 \times 10^{-5}(1200^2) = 295.2 \quad \text{Btu/lb} \]

\( \Rightarrow \)

\[ \dot{q}_s = \frac{226.95}{\sqrt{R_0}} \left(1 - \frac{295.2}{2000}\right) \]

\[ \dot{q}_s = 193.45 / \sqrt{R_0} \quad \text{Btu/ft}^2\cdot\text{sec} \]
Compute heat loads for various nose radii.

\[ k_0 = 1" = \frac{1}{12} \text{ ft} \]

\[ q_c = 193.45 \sqrt{1/2} = 670 \text{ Btu/ft}^2/\text{sec} \]

\[ q_r = q \cdot \sin \theta \]

**Recommended Area**

\[ dA = 2\pi R_0 \sin \theta \cdot d\theta \]

\[ q_{\text{nose}} = \int_0^{90^\circ} \frac{q_c}{(\cos \theta)^{3/2}} \cdot 2\pi R_0 \sin \theta \cdot d\theta \]

\[ q_{\text{nose}} = 350.81 \int_0^{90^\circ} (\cos \theta)^{-3/2} \sin \theta \cdot d\theta \]

\[ u = \cos \theta \quad du = -\sin \theta \cdot d\theta \]

\[ \int (\cos \theta)^{-3/2} \sin \theta \cdot d\theta = -u^{-1/2} \cdot du \]

\[ = -\frac{2u^{1/2}}{3} + c \]

\[ q_{\text{nose}} = -350.81 \left( \frac{2}{3} \right) \left( \cos \theta \right)^{5/2} \bigg|_0^{90^\circ} \]

\[ q_{\text{nose}} = -140.32 \left[ (\cos 90^\circ)^{5/2} - (\cos 0^\circ)^{5/2} \right] \]

\[ q_{\text{nose}} = \frac{138.55}{Btu/\text{sec}} \]

**Fig 6**
\[
R_o = 2'' = \frac{\sqrt{6}}{2} \mu
\]

\[
\dot{\theta}_s = \frac{193.45}{\sqrt{6}} = 473.8 \, Btu/\mu^2 \cdot \text{sec}
\]

\[
\dot{\theta} = \dot{\theta}_s \cdot \text{Area}
\]

\[
\dot{\theta}_\text{noise} = \int_0^\theta \dot{\theta}_s (\cos \theta)^{3/2} \cdot 2\pi \cdot R_o \sin \theta \, d\theta
\]

\[
\dot{\theta}_\text{noise} = -\dot{\theta}_s \cdot 2\pi \cdot R_o \frac{3}{5} (\cos \theta)^{5/2} \bigg|_0^{\theta_0}
\]

\[
\dot{\theta}_\text{noise} = -198.48 \left[ (\cos 90^\circ)^{5/2} - 1 \right]
\]

\[
\dot{\theta}_\text{noise} = 196 \, Btu/\text{sec}
\]

\[
R_o = 3'' = \frac{\sqrt{4}}{2} \mu
\]

\[
\dot{\theta}_s = \dot{\theta}_s \cdot 2\pi \cdot R_o \frac{3}{5} (0.98743)
\]

\[
\dot{\theta}_s = (193.45/\sqrt{4}) \cdot 2\pi \cdot (\frac{3}{5}) (0.98743)
\]

\[
\dot{\theta}_s = 240 \, Btu/\text{sec}
\]

ORIGINAL PAGE IS OF POOR QUALITY
\[ M = 10 \quad 90 \, \text{k} \]

\[ a = \sqrt{\frac{R}{T}} = \sqrt{(1.4)(173.3)(460 - 56.53)} \]

\[ a = 988.9 \, \text{ft/sec} \]

\[ V_{\infty} = 9889 \, \text{ft/sec} \]

\[ e_\infty = e_{\infty} \cdot \exp\left(-t/23700\right) = 2.378 \times 10^{-3} \cdot \exp\left(-9000/23700\right) \]

\[ = 5.333 \times 10^{-5} \, \text{bs ft/sec} \]

\[ h_e = \frac{h_e}{25} = \frac{V_{\infty}^2}{50100} \]

\[ = \frac{2.25(460 - 56.53)}{9889} \cdot \frac{9889}{50100} \]

\[ = 2053 \, \text{bs ft} \]

\[ \dot{q}_s = 21.92 \cdot \frac{1}{\sqrt{R_o}} \cdot (e_\infty)^5 \cdot \left(\frac{V_{\infty}}{1000}\right)^3 \left(1 - \frac{h_w}{h_e}\right) \]

\[ \dot{q}_s = 21.92 \cdot \left(5.333 \times 10^{-5}\right)^5 \cdot \left(\frac{9889}{1000}\right)^3 \left(1 - \frac{h_w}{2053}\right) \]

\[ \dot{q}_s = \frac{154.9}{\sqrt{R_o}} \left(1 - \frac{h_w}{2053}\right) \]
Well temp = 1200° R

\[ q_s = \frac{154.8}{\sqrt{R_0}} \left(1 - \frac{295.2}{2053}\right) \]

\[ q_s = 132.54/\sqrt{R_0} \]

\[ R_0 = 1'' = \frac{1}{12} \pi \]

\[ q_nose = 94.95 \text{ Btu/sec} \]

\[ R_0 = 2'' = \frac{1}{6} \pi \]

\[ q_nose = 134 \text{ Btu/sec} \]

\[ R_0 = 3'' = \frac{1}{4} \pi \]

\[ q_nose = 167 \text{ Btu/sec} \]
\[
\begin{align*}
V_\infty &= 12 \, (488.9) = 11867 \, \text{ft/sec} \\
C_\infty &= 5.333 \times 10^{-5} \\
\frac{h_c}{h_e} &= 0.25 \left( \frac{V_\infty}{50} \right)^2 \\
&= 0.25 \left( \frac{11867}{50} \right)^2 \\
&= 2912 \, \text{ft/sec} \\
\dot{g}_s &= \frac{2192}{\sqrt{R_0}} \left( 5.333 \times 10^{-5} \right)^5 \left( \frac{11867}{1000} \right)^3 \left( 1 - \frac{h_c}{h_e} \right) \\
&= \frac{267.5}{\sqrt{R_0}} \left( 1 - \frac{h_c}{h_e} \right) \\
\dot{g}_s &= \frac{267.5}{\sqrt{R_0}} \left( 1 - \frac{295.2}{2912} \right) \\
\dot{g}_s &= \frac{210}{\sqrt{R_0}} \\
R_0 &= 1'' = \frac{1}{12} \, \text{ft} \\
\dot{g}_s &= 210/\sqrt{12} = 831 \, \text{ft/hr}^2 \text{sec} \\
\dot{g}_{ose} &= 831 \, R_0 \left( \frac{2.4816}{2} \right) \\
\dot{g}_{ose} &= 172 \, \text{ft/hr}^2 \text{sec} \\
R_0 &= 2'' = \frac{1}{6} \, \text{ft} \\
\dot{g}_s &= 210/\sqrt{6} = 588 \, \text{ft/hr}^2 \text{sec} \\
\dot{g}_{ose} &= 588 \times R_0 \left( \frac{2.4816}{2} \right) = 243 \, \text{ft/hr}^2 \text{sec}
\end{align*}
\]
\[ R_o = 3'' = \frac{1}{4} \text{ ft} \]

\[ q_s = \frac{290}{\sqrt{25}} = 480 \frac{\text{lb}}{\text{ft}^2 \text{sec}} \]

\[ q_{nose} = 480 R_o (2.4816) = 298 \frac{\text{lb}}{\text{sec}} \]
Appendix 1.3: Heating rates for Noses of Various Radii
Heating Rates For Noses of Various Radii

\[ 0 \leq \theta \leq 80^\circ \]

\[ \dot{q} = q_s (\cos \theta)^{\frac{3}{2}} \]

\[ q = \dot{q} \cdot \text{Area} \]

Incremental Area \( dA = 2\pi R_o \sin \theta \, d\theta \)

\[ q_{\text{nose}} = \int_0^{80^\circ} \dot{q}_s (\cos \theta)^{\frac{3}{2}} \cdot 2\pi R_o \sin \theta \, d\theta \]

\[ q_{\text{nose}} = q_s \cdot R_o \cdot (2.4816) \]

\( q \) (Btu/sec)

300

200

100

m=12, 70K

m=10, 70K

m=10, 90K

Ro (inches)

Original page is of poor quality
Appendix 1.4: Equilibrium Temperature Calculation
Lee's Approx Method.

\[
\dot{q}_s \sqrt{R_0} = \left[ 1 - \frac{h_x}{h_y} \right] \left[ 15.5 \cdot 2 \left( \frac{\rho \cdot U_\infty}{1000} \right)^{0.5} \right] \left[ T_w - \frac{1}{2} \left( \frac{\rho \cdot U_\infty}{1000} \right)^{0.5} \right] \left[ \frac{h_x}{h_y} \right] \left[ \frac{T_w}{T_x} \right] \left[ \frac{V_x}{(V_x)^{0.5}} \right]
\]

\[
\dot{q}_s = \varepsilon_0 \left[ T_w^4 - T_x^4 \right]
\]

\[
\dot{q}_s \sqrt{R_0} = \left[ 1 - \frac{T_w}{T_x} \left( \frac{\rho \cdot U_\infty}{1000} \right)^{0.5} \right] \left[ 2.192 \varepsilon_0 \left( \frac{V_x}{(V_x)^{0.5}} \right)^{0.5} \right]
\]

\[
\dot{q}_s = 120.545 \text{ Btu/s} - 9.252 \times 10^{-7} T_x^2
\]

\[
\dot{q}_s = 46.33949\text{ Btu/s} - 7.65793 \times 10^{-3} T_w^2 - 3.27267 \times 10^{-7} T_w^2
\]

\[
\epsilon = 0.85
\]

\[
\sigma = 4.81 \times 10^{-13} \left( \frac{8.1 \times 10^{-17}}{\varepsilon_0 \cdot 2.86 \text{ cS}} \right)
\]
$$120.5454 - 0.026265(T_w)$$
$$- 9.2538 \times 16^{-2} (T_w)^2 = \sqrt{R_0 \epsilon \sigma (T_w^{-4} - T_o^{-4})}$$

$$\frac{x^2}{R_0} (T_w - T_o) = 2.9484, \; C^{14} - 5.29534 \times 10^{-2} T_w$$
$$= 2.263 \times 10^{-2}, \; C^{14} T_w$$

$$T_w^4 = \frac{1}{R_0} \left[ 2.9484 \times 5^{14} - 5.29534 \times 10^{-2} T_w \right]$$
$$- 2.263 \times 10^{-2}, \; C^{14} T_w$$

$$T_w = 222.2362 \; R = 234.23612 \; \text{deg}$$
$$R_0 = 4', \; C \; \text{deg}$$
$$f_3 = \epsilon \sigma (T_w^{4} - T_o^{4}) = 26 \; \frac{B + 1}{B - 1.5 \epsilon} \; \text{sec}$$
\[ 46.38949 - 7.65793 \times 10^{-3} T_w - 3.27262 \times 10^{-7} T_w^2 = \sqrt{R_0} \cdot \sigma \left( T_w^4 - T_a^4 \right) \]

\[ T_w^4 = \left[ 1.13463 \times 10^4 - 1.87324 \times 10^9 T_a \right] \left( \frac{1}{\sqrt{R_0}} \right) \]

\[ \text{Heating} \Rightarrow T_w = 2381.35459 \quad ^{\circ}R = 1921.4 \text{ F} \]

\[ R_0 = 4' ; \quad 115 \text{ kV} \geq \text{ V}_{	ext{max}} \]

\[ Q_{\text{e}} = 6 \sigma (T_w^4 - T_a^4) = 13.13 \cdot \frac{6^2}{4 \pi \cdot 150^2} \]
Appendix 1.5: Preliminary Heat Exchanger Analysis
HEAT EXCHANGER

CONFIGURATION

{ \{ T_{h,i} \} }

HOT FLUID, IN

COLD FLUID, OUT

{ \{ T_{c,0} \} }

HOT FLUID OUT

COLD FLUID, IN

{ \{ T_{h,0} \} }

{ \{ T_{c,i} \} }

HOT FLUID: GLYCOL

COLD FLUID: HYDROGEN

QUESTION: ARE FINS REQUIRED ON TUBE 1?

ORIGINAL PAGE IS OF POOR QUALITY
**Properties**

<table>
<thead>
<tr>
<th></th>
<th>H₂ (≈ 250 K)</th>
<th>30% Glycol (200 K)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( C_p )</td>
<td>15200 J/kgK</td>
<td>4041 J/kgK</td>
</tr>
<tr>
<td>( \mu )</td>
<td>77 \times 10^{-7} kg/m·s</td>
<td>940.6 \times 10^{-7} kg/m·s</td>
</tr>
<tr>
<td>( k )</td>
<td>180 \times 10^{-3} W/m·K</td>
<td>509.9 \times 10^{-3} W/m·K</td>
</tr>
<tr>
<td>( P_r )</td>
<td>0.69</td>
<td>0.745</td>
</tr>
</tbody>
</table>

**Relations**

\[
R_e = \frac{4 m}{\rho \pi D},
\]

\[
P_r = \frac{\mu c_p}{k},
\]

\begin{equation}
N_u = 0.023 R_e^{0.8} P_r^{0.33}, \quad R_e > 2300, \quad P_r > 0.5 \quad \text{(turb. flow)}
\end{equation}

\[
N_u = 3.65 + \frac{0.065 (D/L) R_e P_r}{1 + 0.04[(D/L) R_e P_r]^{2/3}}, \quad \text{(laminar flow)}
\]

\[
N = \frac{N_u k}{D}
\]
First, an energy balance is performed, where the heat exchanger is assumed to be 100% effective — that is, \( T_{c,0} = T_{hi} \):

\[
\dot{m}_H C_{p_H} (T_{H,i} - T_{H,o}) = \dot{m}_c C_{p_c} (T_{c,0} - T_{c,i})
\]

\[
\frac{\dot{m}_H}{\dot{m}_c} = \frac{C_{p_H} (T_{c,0} - T_{c,i})}{C_{p_H} (T_{H,i} - T_{H,o})}
\]

Substituting values:

\[
\frac{\dot{m}_H}{\dot{m}_c} = \frac{15300 (500 - 20)}{4041 (500 - 300)} = 9.025
\]

Thus, this ratio of mass flow rates is required for 100% effectiveness. If this heat exchanger is required to be quite effective, say 80% effective, then

\[
\frac{\dot{m}_H}{\dot{m}_c} = .80 (9.025) = 7.22
\]

Tube Flow \& Lam. Flow

If \( \dot{m}_c \) is chosen, then

\[
\dot{m}_c = .001 \text{ kg/s} \quad \dot{m}_H = .00732 \text{ kg/s}
\]

Also, \( D = .010 \text{ m} \), \( D_n = \frac{4 \text{(Area)}}{\text{Perimeter}} = D' - D \);

if \( D' = .020 \text{ m} \), then \( D_n = .010 \text{ m} \) also, simplifying this analysis.
Results are listed, in which Nusselt number correlations were used to calculate heat transfer coefficients.

**Tub**

\[ \dot{m}_c = 0.001 \text{ kg/s} \]
\[ \dot{m}_w = 0.00737 \text{ kg/s} \]
\[ Re_c = 16535.6 \]
\[ Re_w = 9800 \]
\[ Nu_{c,0} = 48.103 \]
\[ Nu_w = 32.46 \]
\[ h_c = 865.85 \text{ W/m}^2\text{k} \]
\[ h_w = 1654.5 \text{ W/m}^2\text{k} \]
\[ \frac{h_w}{h_c} = 1.9 \]

\[ \dot{m}_c = 0.01 \text{ kg/s} \]
\[ \dot{m}_w = 0.0722 \text{ kg/s} \]
\[ Re_c = 165350.0 \]
\[ Re_w = 97731 \]
\[ Nu_{c,0} = 303.85 \]
\[ Nu_w = 204.9 \]
\[ h_c = 5469.3 \]
\[ h_w = 10443 \]
\[ \frac{h_w}{h_c} = 1.9 \]

**Lam**

\[ \dot{m}_c = 0.0001 \text{ kg/s} \]
\[ \dot{m}_w = 0.0007 \text{ kg/s} \]
\[ Re_c = 1653.55 \]
\[ Re_w = 947.59 \]
\[ Nu_{c,0} = 41.364 \]
\[ Nu_w = 4.05 \]
\[ h_c = 76.76 \]
\[ h_w = 306.5 \]
\[ \frac{h_w}{h_c} = 2.7 \]

Thus, \( \frac{h_w}{h_c} \) is only about 2 to 3, which means that \( h_c \) (of hydrogen) is only 2 to 3 times smaller than \( h_w \) (of glycol). Thus, fins are probably not necessary for Tube 1.
Appendix 2: Computer Codes
Appendix 2.1.1: HETAQ Description
We used Lees Approximate Method in HETAQ because it quickly and accurately determined stagnation point heating rates. From there, obtaining the skin equilibrium temperatures for varying flight conditions was just a matter of equating two equations and solving for the unknown.

Using the simple energy balance equation:

\[ q_c = q_r \]

the convective heat transfer \( q_c \) is identical to the radiative heat transfer \( q_r \). So we can now equate \( q_c \) with the radiative heat expression:

\[ q_c = q_r = \varepsilon \sigma (T_{wall}^4 - T_{\infty}^4) \]

Now, equating the expression of \( q_c \) from Lees Approximate Method with that from the radiative expression and solving for the unknown wall temperature \( T_{wall} \) gives a fourth order equation with which Newton's Method was used, iterating to solve \( T_{wall} \) with the HETAQ code.

Due to the limitations of fuel it was necessary to determine how long it would take for the nose to reach its equilibrium temperature. If it is longer than the calculated flight time, then the data obtained is in the transient region, and possibly not as valuable as data
taken at a cruise equilibrium temperature. Therefore, this time calculation was a valuable step in determining what the proposed flight condition would be.

In calculating the time it took for the skin to reach this equilibrium temperature, an approximate energy balance was used to give an estimate of the flight time required:

\[ pCV(T_{w,eq} - T_{initial}) = \dot{q}_s A \Delta t - \sigma A (T_{w,eq} - T_{\infty}) \Delta t \]

This equation assumes that the nose piece is a simple thin skin structure (no heat sink) which is actively cooled to the desired wall temperature—to be determined by the material's thermal properties. The time needed to reach equilibrium is quite short for the materials we analyzed. Using a thin titanium skin around 1 mm thick would only require a time of 1.8 minutes. This should leave enough time for equilibrium data to be taken.

Here is a summary of the process used. Calculate what the stagnation point heating rates will be using Lees Approximate Method. Then, determine what the skin equilibrium temperature will be, using the \( q_s = q_r \) energy balance equation. Now, in order to show that there will be a heating problem, if the equilibrium temperature is greater than the limiting temperature of the nose material, active cooling must be used. To determine how much cooling will be necessary, set the wall temperature at the limiting material temperature and use Lees Approximate Method to determine

\[ q_s = q_r + q_{cool} \]

where \( q_{cool} \) is the cooling rate to be determined by the active cooling system.
Appendix 2.1.2: HETAQ Data
**HELAQ . DATA**

**Altitude = 100000 ft**

<table>
<thead>
<tr>
<th>Time</th>
<th>Radius</th>
<th>Mass equ. term</th>
<th>a</th>
<th>v</th>
<th>Flight cond. term</th>
<th>Thrust</th>
<th>1 min.</th>
<th>2 min.</th>
<th>3 min.</th>
<th>4 min.</th>
<th>5 min.</th>
<th>6 min.</th>
<th>7 min.</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1</td>
<td>3342.2</td>
<td>0.048</td>
<td>0.048</td>
<td>1250</td>
<td>12027.5</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>3342.2</td>
<td>0.048</td>
<td>0.048</td>
<td>1250</td>
<td>12027.5</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>3342.2</td>
<td>0.048</td>
<td>0.048</td>
<td>1250</td>
<td>12027.5</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>4</td>
<td>3342.2</td>
<td>0.048</td>
<td>0.048</td>
<td>1250</td>
<td>12027.5</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>5</td>
<td>3342.2</td>
<td>0.048</td>
<td>0.048</td>
<td>1250</td>
<td>12027.5</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>6</td>
<td>3342.2</td>
<td>0.048</td>
<td>0.048</td>
<td>1250</td>
<td>12027.5</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>7</td>
<td>3342.2</td>
<td>0.048</td>
<td>0.048</td>
<td>1250</td>
<td>12027.5</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>8</td>
<td>3342.2</td>
<td>0.048</td>
<td>0.048</td>
<td>1250</td>
<td>12027.5</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td>5.93</td>
<td></td>
</tr>
</tbody>
</table>

**Final Time = 105.15 sec / 1.75 min**
<table>
<thead>
<tr>
<th>State Reached</th>
<th>1 min</th>
<th>1/16&quot; (1.59 mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum</td>
<td>8.24 sec</td>
<td>3.08 sec</td>
</tr>
<tr>
<td>Titanium</td>
<td>5.67 sec</td>
<td>2.41 sec</td>
</tr>
</tbody>
</table>

**Flight Conditions:**
- Thrust = 160,297.8 lbs
- Fuel flow rate = 53.74 lbs/sec
- Flight time = 165.35 sec

**Proposed Flight Condition:**
- Thrust = 186,45.73 lbs
- Fuel flow rate = 80.4 lbs/sec
- Flight time = 150.62 sec

**Alternatives:**
- Thrust = 160,297.8 lbs
- Fuel flow rate = 53.74 lbs/sec
- Flight time = 165.35 sec

**Alternatives:**
- Thrust = 186,45.73 lbs
- Fuel flow rate = 80.4 lbs/sec
- Flight time = 150.62 sec

**Alternatives:**
- Thrust = 160,297.8 lbs
- Fuel flow rate = 53.74 lbs/sec
- Flight time = 165.35 sec
<table>
<thead>
<tr>
<th>Flight conditions:</th>
<th>ISP = 1380 sec</th>
<th>Thrust = 159845.73 lbs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fuel flow rate = 80.4 l/s</td>
<td>Flight time = 1.64 min</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Engine Performance</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Nose Equivalent</td>
<td>3700.04 ft-lb</td>
<td>-</td>
</tr>
<tr>
<td>Specific Thrust</td>
<td>57.64 lb/in.*</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material</th>
<th>Length</th>
<th>ISP</th>
<th>J</th>
<th>Thrust</th>
<th>Flow Rate</th>
<th>Flight Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum</td>
<td>4.03 sec</td>
<td>1.13</td>
<td>1.25</td>
<td>8.95 sec</td>
<td>0.44 sec</td>
<td></td>
</tr>
<tr>
<td>Titanium</td>
<td>4.03 sec</td>
<td>1.13</td>
<td>1.25</td>
<td>8.95 sec</td>
<td>0.44 sec</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Flight conditions:</th>
<th>ISP = 1380 sec</th>
<th>Thrust = 159845.73 lbs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fuel flow rate = 80.4 l/s</td>
<td>Flight time = 1.64 min</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Engine Performance</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Nose Equivalent</td>
<td>341.41 A</td>
<td>-</td>
</tr>
<tr>
<td>Specific Thrust</td>
<td>57.64 lb/in.*</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material</th>
<th>Length</th>
<th>ISP</th>
<th>J</th>
<th>Thrust</th>
<th>Flow Rate</th>
<th>Flight Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum</td>
<td>4.03 sec</td>
<td>1.13</td>
<td>1.25</td>
<td>8.95 sec</td>
<td>0.44 sec</td>
<td></td>
</tr>
<tr>
<td>Titanium</td>
<td>4.03 sec</td>
<td>1.13</td>
<td>1.25</td>
<td>8.95 sec</td>
<td>0.44 sec</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Flight conditions:</th>
<th>ISP = 1380 sec</th>
<th>Thrust = 159845.73 lbs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fuel flow rate = 80.4 l/s</td>
<td>Flight time = 1.64 min</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Engine Performance</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Nose Equivalent</td>
<td>441.24 A</td>
<td>-</td>
</tr>
<tr>
<td>Specific Thrust</td>
<td>156.5 lb/in.*</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material</th>
<th>Length</th>
<th>ISP</th>
<th>J</th>
<th>Thrust</th>
<th>Flow Rate</th>
<th>Flight Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum</td>
<td>2.76 sec</td>
<td>1.13</td>
<td>1.25</td>
<td>4.38 sec</td>
<td>0.38 sec</td>
<td></td>
</tr>
<tr>
<td>Titanium</td>
<td>2.76 sec</td>
<td>1.13</td>
<td>1.25</td>
<td>4.38 sec</td>
<td>0.38 sec</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Flight conditions:</th>
<th>ISP = 1380 sec</th>
<th>Thrust = 159845.73 lbs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fuel flow rate = 80.4 l/s</td>
<td>Flight time = 1.64 min</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Engine Performance</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Nose Equivalent</td>
<td>4125.42 A</td>
<td>-</td>
</tr>
<tr>
<td>Specific Thrust</td>
<td>138.5 lb/in.*</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material</th>
<th>Length</th>
<th>ISP</th>
<th>J</th>
<th>Thrust</th>
<th>Flow Rate</th>
<th>Flight Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum</td>
<td>1.77 sec</td>
<td>1.13</td>
<td>1.25</td>
<td>5.92 sec</td>
<td>0.38 sec</td>
<td></td>
</tr>
<tr>
<td>Titanium</td>
<td>1.77 sec</td>
<td>1.13</td>
<td>1.25</td>
<td>5.92 sec</td>
<td>0.38 sec</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Flight conditions:</th>
<th>ISP = 1380 sec</th>
<th>Thrust = 159845.73 lbs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fuel flow rate = 80.4 l/s</td>
<td>Flight time = 1.64 min</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Engine Performance</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Nose Equivalent</td>
<td>4278.12 A</td>
<td>-</td>
</tr>
<tr>
<td>Specific Thrust</td>
<td>122.7 lb/in.*</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material</th>
<th>Length</th>
<th>ISP</th>
<th>J</th>
<th>Thrust</th>
<th>Flow Rate</th>
<th>Flight Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum</td>
<td>1.77 sec</td>
<td>1.13</td>
<td>1.25</td>
<td>5.92 sec</td>
<td>0.38 sec</td>
<td></td>
</tr>
<tr>
<td>Titanium</td>
<td>1.77 sec</td>
<td>1.13</td>
<td>1.25</td>
<td>5.92 sec</td>
<td>0.38 sec</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Flight conditions:</th>
<th>ISP = 1380 sec</th>
<th>Thrust = 159845.73 lbs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fuel flow rate = 80.4 l/s</td>
<td>Flight time = 1.64 min</td>
<td></td>
</tr>
</tbody>
</table>

**ORIGINAL PAGE IS OF POOR QUALITY**
<table>
<thead>
<tr>
<th>Name</th>
<th>Initials</th>
<th>Time 1</th>
<th>Time 2</th>
<th>Time 3</th>
<th>Time 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test 1</td>
<td>JH</td>
<td>1.28</td>
<td>1.17</td>
<td>1.85</td>
<td>1.65</td>
</tr>
<tr>
<td>Test 2</td>
<td>LM</td>
<td>1.44</td>
<td>1.33</td>
<td>1.85</td>
<td>1.65</td>
</tr>
<tr>
<td>Test 3</td>
<td>JK</td>
<td>1.44</td>
<td>1.33</td>
<td>1.85</td>
<td>1.65</td>
</tr>
<tr>
<td>Test 4</td>
<td>GH</td>
<td>1.44</td>
<td>1.33</td>
<td>1.85</td>
<td>1.65</td>
</tr>
</tbody>
</table>
Flight conditions: 120 sec  
Thrust = 28,4170.2 lbf
Fuel burn rate = 2.84 lb/sec
Flight time = 41.56 sec, 0.68 min

1 sec  

Table  

<table>
<thead>
<tr>
<th>Condition</th>
<th>1 sec</th>
<th>2 sec</th>
<th>3 sec</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum</td>
<td>1.41</td>
<td>2.98</td>
<td></td>
</tr>
<tr>
<td>Titanium</td>
<td>1.73</td>
<td>4.02</td>
<td>5.75</td>
</tr>
</tbody>
</table>

ORIGIINAL PAGE IS OF POOR QUALITY
Appendix 2.1.3: HETAQ Results
Stagnation Conditions

Nose

Note: air breathing
Scurve not
valid for h>100
C = 0.85

$70,000 \text{ ft}$

$E = 0.85$

HETAQ

Iteration equation for general case

$$T_{n+1} = T_n - \frac{R_0 (T_n)^4 + C(T_n)^2 + D(T_n - A)}{4R_0 (T_n)^3 + 2C(T_n)^2 + D}$$

$V_o = \text{Mach} \cdot V_{o}$

$A = \left[ \frac{2.192 \times 10^{-0.5}}{E \sigma} \right] P \left( V_o \right)^3$

$B = \left( \frac{T_o}{4} \right) + \left( V_o \right)^2 / 50,000$

$C = \left( A/B \right) \times 10^{-5}$

$D = \left( A/B \right) \times 0.234$

$\dot{q}_s = E \sigma (T_w)^4$

$\dot{q}_w = \sigma (T_w)^4$

Equilibrium Tend 4 Heating (C. W.)

$9.5 = 505.25 \times 0.1$

$9 = 1000$

$7 = 200$

$7 = 300$

$7 = 400$

$8E = 0.4 \times 0.4$

$E = 0.2$

$E = 0.25$

$E = 0.472.8$

$E = 0.585.8$

$E = 0.593.3$

$E = 0.72$

$E = 1.74$

$E = 2.9$

$E = 3.9$

$E = 4.9$

$E = 5.9$

$E = 6.9$

$E = 7.9$

$E = 8.9$

$E = 9.9$

$E = 10.9$
Stagnation Conditions

C Nose

Fig. 4

100,000 ft
$e = 0.85$

ORIGINAL PAGE IS
OF POOR QUALITY
Stagnation Conditions

\( e = 0.85 \)

Nose

Valid for scramjets

\( 5 \leq M \leq 15 \)
Appendix 2.1.4: HETAQ Code Listing
1 DIM T(4,12,6),Q(4,12,6),TR(4,12,6),Ti(4,12,6),T2(4,12,6),ISp(12),TS(4,12,6,4)
2 TEXT : HOME : READ N: FOR I = 1 TO N: READ Z(I),RO(I),TI(I),CS(I): NEXT I: REM READ IN DATA...ALT eFT). DENSITY (SLUGS/FT'3), TEMP INFINITY (R). SPEED OF SOUND (FT/SEC)
3 READ NUM: FOR I = 1 TO NUM: READ M$(I),CP(I),DNS(I),TH(I): NEXT: REM READ IN MATERIAL PROPERTIES
4 FOR I = 8 TO 12: READ ISP(I): NEXT : REM ISP FOR MACH =
5 E = 0.85:S = 4.91E - 13:PI = 3.141592654:S0 = 605: REM SET SIGMA, EMMI SSIVITY, PI, PLAN FORM AREA (FT'2)
6 Z = 0.01: REM ACCURACY SETTING
7 Di = 8:L = 40:CD = 0.20:HD = 4.42: REM set dimensions of tank, drag co eff. density of liq hydrogen
8 VOL = PI * ((DI / 2) * 2) * L:WH = HD * VOL: REM vol, weight of H2 in ft'3, Lo
9 PRINT : PRINT "(A)utomatic calculation or (M)anual?": INPUT I$: IF I$ = "M" OR I$ = "m" THEN 2000
97 REM
98 REM VARY PARAMETERS
99 REM
1^ FOR I = 4 TO 4
1. FOR M = 8 TO 12: REM MACH NUMBER
1. FOR K = 1 TO 6: REM RADIUS IN INCHES
1. R = SOP (K / 12)
210 CS = CS(1);
220 Ti = Ti(1)
230 RO = RO(1)
240 ISP = ISP(M)
500 GOSUB 5000: GOSUB 10000: GOSUB 30000
600 T(I,M,K) = INT (100 * T) / 100:Q(I,M,K) = INT (100 * Q) / 100
610 FOR L = 1 TO NUM:TS(I,M,K,L) = INT (100 * DNS(L) * CP(L) * TH(L) * (Ti(I,M,K) - 500) / Q(I,M,K)) / 100: NEXT L
620 FL(I,M,K) = INT (100 * FL) / 100:FR(I,M,K) = INT (100 * TR) / 100:
700 PRINT FL= FUEL FLOW. TR = THRUST
710 T(I,M,K) = Ti(T2(I,M,K) = T2
997 REM
998 REM SAVE DATA TO DISK
999 REM
1000 PRINT CHR$(4);"OPEN HETAG.DATA"
1010 PRINT CHR$(4);"WRITE HETAG.DATA"
1050 PRINT "Initial conditions": PRINT : PRINT "Density of hydrogen = "HD:"lb/ft'3": SPC( 3):"Tank: length = "L:" ft Diameter = "DI:" ft"
1060 PRINT "Fuel: Volume = "VOL:" ft'3": SPC( 5):"Weight = "WH
1070 PRINT "Cd = "CD: SPC( 3):"Plan form area = "S0:" ft'2"
1 ) PRINT
1100 FOR I = 4 TO 4
1. PRINT : PRINT : PRINT "Altitude = "Z(I):" ft"
1200 FOR M = 8 TO 12
REM DO MANUAL CALCULATIONS
2000 PRINT "Mach number = " :M
2050 INPUT "Density = " :R
2100 PRINT "Speed of sound = " :CS
2150 INPUT "T(infinity) = " :TI
2200 PRINT "Nose radius = (in ft)" :R = SQR (K)
2250 ISP = ISP(M)
2300 GOSUB 5000: GOSUB 10000: GOSUB 30000
2350 PRINT "Density of hydrogen = " :HD: " lb/ft^3" : SPC(3):"Tank length = " :L: " ft Diameter = " :DI: " ft"
2400 PRINT "Fuel volume = " :VOL: " ft^3" : SPC(5):"Fuel weight = " :WH
2450 PRINT "Co = (CD) = SPC(5):"Plan form area = " :SO: " ft^2" : PRINT ...
2500 FOR L = 1 TO NUM:TM(L) = INT (100 * DNS(L) * CP(L) * TH(L) * (T - 500) / 0) / 100: NEXT L
2550 PRINT "Equilibrium temp = " :T: " R" : PRINT "Heating rate = " :0: " btu/ft^2 sec"
2600 PRINT "Steady state reached 1 mm 1/16: CHR$ (34): " (1.59 mm)
2800 REM FLIGHT CONDITIONS
2850 PRINT "Flight conditions: ISP = " :ISP(M);" sec Thrust = " :TR(I,M,K): " lbs"
2950 NEXT K,M,I
3000 PRINT "CLOSE HETAO.DATA"
3050 END
3100 GOSUB 5000
3200 B = (TI / 4) + (V * V) / 50100: C = (A / B) * 1E - 5
3250 D = (A / B) * 0.234
3300 REM M = MACH #: CS = SPEED OF SOUND (FT/SEC): V = VINFINITY (FT/SEC): RO = DENSITY AT ALTITUDE: T1 = TEMP INFINITY (R): S [#] BTU/FT^2
2 SEC R'4
5040 RETURN
9998 REM
9999 REM DO ITERATION CALCULATION
10000 REM
10100 T = 2500
10500 X = T - ((R * T * T * T * T) + (C * T * T) + (D * T) - A) / ((4 * 
R * T * T * T) + (2 * C * T * T) + D)
10800 IF (ABS (T - X) > Z) THEN T = X: GOTO 10500
10900 Q = E * S * T * T * T * T
11000 RETURN
29999 REM
30000 REM DO THRUST CALCULATIONS
30001 REM
30010 TR = CD * 0.5 * RO * V * V * SO: REM THRUST EQN
30020 FL = TR / ISP: REM FLOW RATE (FT^3 / SEC)
30050 T1 = INT (100 * WH / FL) / 100: T2 = INT (100 * T1 / 60) / 100: REM
FLIGHT TIME IN SEC AND MIN
31000 RETURN
49997 REM
49998 REM ATMOSPHERIC DATA
60000 REM
4-10 DATA 5.70000,1.3761E-4,4392.97,971.05
4-10 DATA 80000,8.445E-5,5398.47,977.81
5-20 DATA 90000,5.2173E-5,5403.95,984.54
50030 DATA 100000,3.320E-5,5409.17,991.2
50040 DATA 110000,2.01E-5,5419.87,1003.8
60000 DATA 4,"Aluminum",0.208,169.5,208E-3,"Aluminum",0.208,169.5,208E-3,-3
60010 DATA "Magnesium",0.232,109.3,281E-3,"Magnesium",0.232,109.5,208E-3,-3
61000 DATA 2350,1988.1775,1488.1313
Appendix 2.2.1: ACSES Description
Actively Cooled Structure Evaluation System (ACSES)

This program integrates the theories of conduction, convection, and fin analysis discussed already. Its main objective is to provide a first approximation in the design of a combined impingement and actively cooled panel concept for the drone nose. The code allows the user to vary tube/nose geometry and material/coolant properties with which it uses to calculate the critical skin temperature distribution, total pressure loss, and exiting coolant temperature.

The design constraint is that the skin of the nose never exceed a maximum allowable temperature. A branching routine is incorporated in the program to help avoid this, but at the expense of an appreciable pressure loss. Thus the task is to find a design that limits the amount of branching necessary to keep the skin temperature below this maximum value and the pressure losses at a minimum.

ACSES requires the following input parameters:

- Number of coolant tubes (tubes)
- Maximum distance back on nose to be analyzed (m)
- Number of stations (subdivisions) for iteration of properties (xnum)
- Coolant tube inner diameter (dtube [mm])
- Thickness of coolant tubes (xtube [mm])
- Friction factor of coolant tubes (f)
- Radius of the nose cone (rnose [mm])
- Skin thickness of nose cone (xskin [mm])
- Skin limiting temperature (tmax [K])
- Mass flow rate of coolant into the manifold (mfr0 [kg/s])
- Impingement nozzle inner diameter (nozd [mm])
- Distance from nozzle exit to impingement on back of nose wall (zn [mm])
- Choice of coolant (liquid hydrogen, water-glycol solution (30%), thermal oil)
- Initial coolant temperature (t_{\text{man}} [K])

ACSES gives the following as output:

- The location of each required branch
- At each station along the nose cone:
  - Surface heat flux (q_{\text{mm}} [W/m^2])
  - Bulk mixing temperature of coolant (t_{\text{mav}} [K])
  - Inside tube wall temperature (t_{\text{w}} [K])
  - Base of tube temperature (t_{0} [K])
  - Local Reynolds number
  - Pressure loss (press\_l [Pa])
- Final results
  - Exiting coolant temperature (t_{\text{mo}} [K])
  - Total pressure loss (\text{loss} [PA], [psi], [atm])
  - Outlet tube mass flow rate (m_{fr} [kg/s])
  - Total number of branches

Assumptions:

- The properties of the tube and skin materials were evaluated near their maximum allowable temperatures, t_{\text{max}}, and taken as constant along the entire nose cone region. Since the properties don't vary too greatly at these temperatures, a constant value serves as a good approximation.
- Nonaccelerating, level-cruise at Mach 10, 100,000 feet. This flight profile allows ACSES to calculate the stagnation point heating rate and use it to determine the heat flux at any point along the nose.
Each tube branch is perfect and occurs over a negligible distance. Pressure and frictional losses are neglected as well.

Coolant properties vary greatly along entire nose region but are taken to be constant over each differential station. They are evaluated at the mean bulk temperature \( (T_{mo} - T_{mi}) / 2 \) and iterated to within 1 degree accuracy.

Tubes start at hemisphere of nose cap and extend back radially.

The distance that the heat transfer across the coolant tubes occurs is taken to be 50% of the tube circumference. This approximation is valid due to the fin effect of the tube shell. Analysis shows that the wall temperature decreases to the coolant temperature within 15% of the circumference. The other 35% of the circumference approximates the heat transfer through the solder that welds the tube to the nose skin.

Radiation off the nose and away from the inside wall have been neglected. Calculations show that it contributes less than 1% of the total heat flux.

Equations:

After the input of the parameters, the program goes through a set of equations to calculate the output. The order is as follows:

- Find the coolant temperature exiting the manifold
- Calculate nose and wall enthalpy to be used in heat flux calculation

\[ h = \]

- Start at first station
- Calculate heat flux

\[ q = \]
-- Calculate distance between two tubes

\[ L = \]

-- Iterate on energy balance to find temperature leaving station

--- Evaluate \( C_p \) at \( T_m \)

--- Find temperature leaving station

\[ T_m_0 = \]

--- Recalculate mean bulk temperature

\[ T_m = \frac{T_m_1 + T_m_0}{2} \]

--- Check with first guess, if not within 1 degree, iterate again

-- Evaluate coolant properties at new mean bulk temperature

-- Calculate Prandtl, Reynolds, and Nusselt numbers:

\[ Pr = \]

\[ Re = \]

\[ Nu = \]

from turbulent inner flow correlation valid for \( Re \) greater than and \( 0.7 < Pr < 13000 \) and ..... Viscosity is evaluated at mean bulk temperature and at wall temperature.

-- Calculate heat transfer coefficient between inner wall and coolant

\[ h = \]

-- Calculate thermal resistance network
R1 =
R2 =

-- Wall temperature of tube from convection of coolant is used to evaluate viscosity of coolant--iterate to within 1 degree as before.

\[ t_{W} = t_{\text{mean}} + (q_{m} \times R_{1}) \]

-- Find base of fin temperature from conduction through tube wall

\[ t_{0} = t_{W} + (q_{m} \times R_{2}) \]

-- Find critical wall temperature using the base of fin temperature with the fin analysis

\[ T_{\text{crit}} = t_{0} + \left( \frac{q_{m} \times (L - 2)}{8 \times k_{\text{skin}} \times x_{\text{skin}}} \right) \]

-- If \( T_{\text{crit}} > t_{\text{max}} \) branch

--- double the number of tubes
--- half the mass flow rate

-- Evaluate pressure loss along the station. Total pressure loss is found from the sum of each station loss.

\[ \text{loss} = \]

-- Print out calculations to an output file

-- Continue the analysis with the next station

Print out results

-- Total pressure loss
-- Final coolant exiting temperature
Limitations of ACSES:

The program only permits the design of a welded tube system. This configuration was chosen because of its ease to manufacture. The analysis also assumes a uniform heat distribution along the skin which in reality, might have several "hot spots" due to surface irregularities. This could be potentially disastrous because the existing design has no way of knowing where these hot spots are or any way to divert coolant to cool them off. If the temperature exceeded critical limits, the skin loses its structural integrity. This in turn would cause the cooling system to fail and allow temperatures to reach equilibrium and burn through the skin, destroying the drone. Currently under investigation are ways of discouraging the formation of hot spots—possibly by applying a thin ceramic layer on the outside of the nose.

Advantages of ACSES:

The program is a quick and relatively simple way to initiate the design of an actively cooled nose cone. The assumptions and correlations used in the code are accurate for the purposes they are serving and were easily integrated into the analysis. ACSES provides the means for a first approximation on what might otherwise be a series of tedious calculations. Due to the interactive nature of the program, many different cases varying in geometry, coolant choice, and materials can be run and compared in a short amount of time.

In the future, ACSES can be modified to provide the thermal design of the leading edge of the drone as well. Correlations would be needed for the given wing geometry which express the heat flux along the leading edge. The code would also have to allow for variable mass flow rates with multiple impingements along the span.
Appendix 2.2.2: ACSES Listing
ACS E S: Actively Cooled Structure Evaluation System
Ginger Koch, James Marcolesco, Greg Noffz, Jerry Yen
Version: 5.24.88

**INPUT PARAMETERS:**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>tubes</td>
<td>number of tubes</td>
</tr>
<tr>
<td>dtube</td>
<td>tube inner diameter, input [mm], converted to [m]</td>
</tr>
<tr>
<td>xtube</td>
<td>thickness of tube, input [mm], converted to [m]</td>
</tr>
<tr>
<td>xskin</td>
<td>skin thickness, input [mm], converted to [m]</td>
</tr>
<tr>
<td>xnum</td>
<td>number of stations</td>
</tr>
<tr>
<td>mfrO</td>
<td>mass flow rate of coolant into manifold [kg/s]</td>
</tr>
<tr>
<td>nozd</td>
<td>impingement nozzle inner diameter [m]</td>
</tr>
<tr>
<td>choice</td>
<td>choice of coolant</td>
</tr>
<tr>
<td>print</td>
<td>1) short print 2) long print</td>
</tr>
</tbody>
</table>

**CONSTANT PARAMETERS:**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>rnose</td>
<td>radius of nose cone [m]</td>
</tr>
<tr>
<td>alpha</td>
<td>cone angle, convert from degrees to radians</td>
</tr>
<tr>
<td>h</td>
<td>maximum distance back from nose analyzed [m]</td>
</tr>
<tr>
<td>zn</td>
<td>impingement nozzle distance [m]</td>
</tr>
<tr>
<td>tmax</td>
<td>material limiting temperature [K]</td>
</tr>
<tr>
<td>tmaxr</td>
<td>1800 [R] for molybdenum</td>
</tr>
<tr>
<td>tman</td>
<td>temperature of coolant into manifold [K]</td>
</tr>
<tr>
<td>p</td>
<td>3.141592654</td>
</tr>
<tr>
<td>f</td>
<td>friction factor of coolant tubes</td>
</tr>
<tr>
<td>kskin</td>
<td>thermal conductivity of the tube wall [W/mK]</td>
</tr>
<tr>
<td>ktube</td>
<td>thermal conductivity of the tube wall [W/mK]</td>
</tr>
</tbody>
</table>

Commercial Tubing Sizes:

<table>
<thead>
<tr>
<th>ID</th>
<th>OD</th>
<th>Thickness</th>
<th>gage</th>
<th>max number for 1&quot; nose</th>
</tr>
</thead>
<tbody>
<tr>
<td>[mm]</td>
<td>[in]</td>
<td>[mm]</td>
<td></td>
<td></td>
</tr>
<tr>
<td>----------</td>
<td>--------</td>
<td>-----------</td>
<td>------</td>
<td>------------------------</td>
</tr>
<tr>
<td>1.68</td>
<td>1/8</td>
<td>0.76</td>
<td>A</td>
<td>50</td>
</tr>
<tr>
<td>3.24</td>
<td>3/16</td>
<td>0.76</td>
<td>A</td>
<td>33</td>
</tr>
<tr>
<td>4.88</td>
<td>1/4</td>
<td>0.76</td>
<td>A</td>
<td>25</td>
</tr>
<tr>
<td>3.92</td>
<td>1/4</td>
<td>1.24</td>
<td>B</td>
<td>25</td>
</tr>
<tr>
<td>6.32</td>
<td>5/16</td>
<td>0.81</td>
<td>21A</td>
<td>20</td>
</tr>
</tbody>
</table>

Cupronickel 70% Cu 30% Ni Wrought
yield strength: 57,000 [psi] 393 [MPa]
tensile strength: 60,000 [psi] 414 [MPa]
melting temperature: 1500 [K]
specific gravity: 8.95
Nozzle Tubing Sizes:

<table>
<thead>
<tr>
<th>ID</th>
<th>OD Thickness</th>
<th>gage</th>
</tr>
</thead>
<tbody>
<tr>
<td>[mm]</td>
<td>[in] [mm]</td>
<td></td>
</tr>
<tr>
<td>-------</td>
<td>--------------</td>
<td>------</td>
</tr>
<tr>
<td>11.08</td>
<td>1/2 0.81</td>
<td>21A</td>
</tr>
<tr>
<td>9.40</td>
<td>1/2 1.65</td>
<td>16</td>
</tr>
<tr>
<td>14.12</td>
<td>5/8 0.89</td>
<td>20A</td>
</tr>
<tr>
<td>17.32</td>
<td>3/4 0.89</td>
<td>20A</td>
</tr>
</tbody>
</table>

Dependent Parameters:

- $t_{cin}$ = coolant temperature exiting manifold and entering tubes = $tk_1$ [K]
- $t_{fcal}$ = temporary film calculation [K]
- $t_{max}$ = bulk mixing temperature of coolant, used to evaluate properties [K]
- $t_0$ = base of fin temperature [K]
- $t_{tw}$ = temperature inside tube wall [K]
- $t_{crit}$ = midline critical temperature [K]
- $m_{avr}$ = mean fluid temperature at station $k$, in and out [K]
- $t_{cuit}$ = final fluid exiting temperature [K]
- $m_f$ = mass flow rate in one tube [kg/s]
- $n_{vel}$ = nozzle velocity [m/s]
- $v_{vel}$ = tube velocity [m/s]
- $q$ = total heat at $dx$ surf in terms of enthalpy from free stream conditions [W]
- $q_{in}$ = $[W/m]$  
- $q_{mm}$ = $[W/m^2]$  
- $\text{choice} = \text{choice of coolant}$  
- $\rho$ = density of coolant from curve fit [kg/$m^3$]  
- $\mu$ = dynamic viscosity of coolant from curve fit [kg/$m^3$]  
- $\mu_{wall}$ = dynamic viscosity of coolant at wall temp  
- $\mu_p$ = specific heat of coolant from curve fit [J/kgK]  
- $\rho_{F}$ = Prandtl number of coolant from curve fit  
- $\rho_{F}$ = Reynolds number of coolant  
- $\nu$ = Nusselt number of coolant from correlation  
- $\rho_{F}$ = nose enthalpy  
- $\rho_{F}$ = wall enthalpy  
- $\mu_{F}$ = local heat transfer coefficient for turbulent flow correlation at station $x$ [W/m²K]  
- $k_{fluid}$ = thermal conductivity of fluid from curve fit [W/mK]  
- $\rho_{s}$ = system pressure loss [%]  
- $\rho_{p}$ = dummy variable, piecewise pressure loss  
- $L$ = distance between tubes at distance $x$ [m]  
- $k$ = position number, $k = 2, x_{num}$  
- $x_{1}, x_{2}$ = distance back from manifold of next and previous station = $(k-1) \times h / x_{num}$ [m]  
- $dx$ = distance between stations $(x_{2} - x_{1})$ [m]  
- $\text{area}_{ah}$ = area that heat transfer takes place across tube  
- $\text{areas}$ = area that conduction takes place across skin  
- $x_{br}$ = distance back of each branch  
- $\text{nbr}$ = number of branches  
- $\text{br}$ = flag that branching has occurred
COOLANT PROPERTY SUBROUTINES:

thcond(choice,tmav,kfluid) thermal cond at tfilm, kh [W/mK]
specht(choice,tmav,cp) specific heat at tfilm, cp [J/kgK]
dynv(choice,tmav,mu) dynamic viscosity at tfilm, tw mu [kg/(m)(sec)]
dense(choice,tmav,ro) density at tfilm, exponential fit, ro [kg/m^3]
fit(choice,tmav,e,answer) curve fit routines

MAIN PROGRAM:

READ IN INPUT DATA:

program main
integer nnum, ntab, flag, nntub, choice, nbr, br
real h,mfr0,mfr,kfluid,kfilm,ktube,hs, mu,L,hoose,hwall,f,
nozd,avcl,dtube, mu, mw,xbr, max

read * ' Coolants: [1] liquid hydrogen ','
read * ' [2] water-glycol solution (30%)' 
read * ' [3] Therminol 60'
read * ' Input coolant choice ---------------------' 
read *' choice
read *' Minimum coolant temperatures:
read *' [1] 12 K 
read *' [2] 270 K 
read *' [3] 250 K 
read *' Input initial coolant temp [K] ---->

100 format(/)
200 format(/)

read *, tmax
print *, 'Input tube diameter [mm] ---------> ',
read *, dtube
dtube = dtube / 1000
print *, 'Input tube thickness [mm] ---------> ',
read *, xtube
xtube = xtube / 1000
max = 2 * pi * rnoe / (dtube + (2*xtube))
print *, 'For given geometry, # tubes < ', max
print *, 'Input number of tubes ---------> ',
read *, tubes

read *, xskin
xskin = xskin / 1000
print *, 'Max distance back on cone [m] ------> ',
read *, h
print *, 'Input number of stations ---------> ',
read *, n

read *, rnos
print *, 'Input coolant mass flow rate [kg/s] --> ',
read *, rnos
print *, 'Input nozzle diameter [mm] ------> ',
read *, rnoe
rnoe = rnoe / 1000
print *, 'Output print --> ',
read *, print
write *, print

---------------------------------------------------------------------

COMPUTE NOSE/WALL ENTHALPY:
---------------------------------------------------------------------

hnoe = (0.234 * tmax) + (0.00001 * tmax ** 2,)
hwall = hnoe
rnoe1 = rnoe * 3.2808
c1=9.792*(rnoe1**(-.3))*alpha*(2080.-hnoe)
     /((2080.-3.6667*hwall))

---------------------------------------------------------------------

FIND COOLANT TEMP

EXITING IMPINGEMENT:
---------------------------------------------------------------------
邪I.

crc.

tnl = to:it7

tmav = 1.2 * tmi

tw = 10. * tmi

x1 = 0.

dx = h / xnum

br = 0

numtub = tubes

rad = alpha * 2 * pi / 360

f = 0.008

mfr = mfr0 / numtub

xh = pi * dtube / 2.

xs = pi * dtube / 2.

call dense(choice, tmi, ro)

nvel = (4*mfr0) / (ro * pi * (nozd ** 2))

tvel = (4*mfr) / (ro * pi * (dtube ** 2))

open(5, file = 'acses.out')

open(10, file = 'plot.qs')

open(11, file = 'plot.tm')

open(12, file = 'plot.tcr')

write(5,1008)

write(5,1009)

if (print.eq.1) then

    write (*,1008)

    write (*,1009)
endif


STATION CALCULATIONS:

main routine to calculate tmi, tmav at each (dx) station
to a distance h away from manifold, xnum times.
limits of integration are x1,x2

do 10 k = 2, xnum

    x2 = k * dx

    L = (2. * pi / numtub) * (x2 * sin(rad) + nose)

Conversion factor from Btu/s ftsqrd to Watts/msqrd is 11352.8855

qmm = 11352.8855* c1 * (x2 **) (-0.2)

qm = L * qmm

q = qm * dx

if (print.eq.0) write (*,1510) k, x2

    format(3X,13, 'x = ',F5.2)

    if (br.eq.1) then

        br = 0

        if (print.eq.0) write (*,1011) nbr
endif

endf
ITERATION:

600  call specht(choice, tmax, cp)
tmo = tmi + (q / (mfr * cp))
titer = (tmo + tmi) / 2
if (abs(titer - tmax) > .GT. Epsilon) then
   tmax = titer
   go to 600
endif

700  call thcond(choice, tmax, kfluid)
call dmvv(choice, tmax, mu)
call dmvv(choice, tw, muw)
call dense(choice, tmax, ro)
pr = mu * cp / kfluid
re = 4 * mfr / (pi * mu * dtube)
mu = (0.027 * e ** 0.8) * ((mu/muw)**.14) * (pr**1/3)
hx = kfluid + mu / dtube
r1 = 1 / (hx * xh)
titer = tmax + (qm * r1)
if (abs(titer - tw) > .GT. Epsilon) then
   tw = titer
   go to 700
endif
r2 = vtube / (ktube * xh)
t0 = tw + (qm * r2)
tcrit = t0 + (qm * (L * 2) / (8 * kskin * xskin))
pres1 = (pi**2) * ro * (dtube**S)
ploss = plost + pres1
if (tcrit .GE. (0.95 * tmax)) then
   br = 1
   nfr = nfr + 1
   xbr = x2
   numtub = tubes * (2 ** nfr)
mfr = mfr0 / numtub
endif

1008  format (’, #', * x: ', qs: ', tmax: ', press: ',
       'tw: ', t0: ', tcrit: ', Re: ’)
1009  format (’, [m], [W/m^2], [K], [Pa],’,
       ’[K]', [K], [K], ’/)
write (5,1010) k, x2, qmm, tmax, press1, tw, t0, tcrit, re
if (print.eq.1) then
  write (*,1010) k,x2,qmm,tma,press,tmav,press1,tw,t0,tcrit,re
endif

format (e15.4,1x,e15.4)
if (br.eq.1) then
  write (5,1011) nbr
endif
tmi = tmo
x1 = x2

---

PRINT OUT RESULTS:

---

write (5,100)
write (5,1020)
if (print.eq.1) then
write (*,100)
write (*,1020)
endif
format ('' Results and initial parameters:'',/)
if (choice.eq.1) then
write (5,1050)
if (print.eq.1) write (*,1050)
format ('' Coolant considered: Liquid hydrogen'',/)
elseif (choice.eq.2) then
write (5,1060)
if (print.eq.1) write (*,1060)
format ('' Coolant considered: Water/glycol solution. (30%)'',/)
else
write (5,1070)
if (print.eq.1) write (*,1070)
format ('' Coolant considered: Therminol 60'',/)
endif
write (5,1110) tcin
format ('' Temperature exiting manifold = '',e10.4,'' [K]'')
write (5,1120) tmo
write (5,1130) ploss, (ploss * 1.4504e-4), (ploss/101350)
format (' System pressure loss = ', e10.4, ' [psi]', f8.2, ' atm')
write (5,1140) tubes
format (' Initial number of tubes = ', i4)
write (5,1150) (dtube * 1000)
format (' Tube diameter = ', f5.2, ' [mm]')
write (5,1160) (xtube * 1000)
format (' Tube thickness = ', f5.2, ' [mm]')
write (5,1170) (rnose * 1000)
format (' Nose radius = ', f4.1, ' [mm]')
write (5,1180) (xsnet * 1000)
format (' Wall thickness = ', f4.1, ' [mm]')
write (5,1190) alpha
format (' Cone half angle = ', f4.1, ' [deg]')
write (5,1200) mfr0
format (' Nozzle mass flow rate = ', f9.4, ' [kg/s]')
write (5,1202) nvel
format (' Velocity of coolant in nozzle = ', f8.3, ' [m/s]')
write (5,1204) mfr
format (' Outlet tube mass flow rate = ', f7.4, ' [kg/s]')
write (5,1206) tvel
format (' Outlet vel of coolant in tubes = ', f8.3, ' [m/s]')
write (5,1208) (zn * 1000)
format (' Impingement target distance = ', f4.1, ' [mm]')
write (5,1210) (nozd * 1000)
format (' Impingement nozzle diameter = ', f5.2, ' [mm]')
write (5,1215) mfr
format (' Number of branches = ', i4)
write (5,1220) tmmax
format (' Limiting material temperature = ', f5.0, ' [K]')
close(5)
close(10)
close(11)
close(12)
if (print.eq.1) then
write (*,1110) tmax
write (*,1120) tmmax
write (*,1130) ploss, (ploss * 1.4504e-4), (ploss/101350)
write (*,1140) tubes
write (*,1150) (dtube * 1000)
write (*,1160) (xtube * 1000)
write (*,1170) (rnose * 1000)
write (*,1180) (xsnet * 1000)
write (*,1190) alpha
write (*,1200) mfr0
write (*,1202) nvel
write (*,1204) mfr
write (*,1206) tvel
write (*,1208) (zn * 1000)
write (*,1210) (nozd * 1000)
write (*,1215) mfr
write (*,1220) tmmax
endif
end

************************** SUBROUTINE MANI **************************

Jerry Yen

Version 5.26.88

Variable description

pi, sigma ... constants
theta ... angle from nose tip of semihemisphere to back end of manifold (deg)
area ... surface area of semi-hemisphere from \( \theta = 0 \) to 90 degrees (m²)

\( r_{nose} \) ... nose radius (m)

\( h_{wall} \) ... wall enthalpy (English units)

(All temperatures are in degrees Kelvin)

\( t_{bnoz} \) ... temperature of fluid entering manifold from nozzle

\( t_{wmax} \) ... maximum allowable wall temperature

\( t_{bin} \) ... fluid bulk temperature exiting manifold (entering tubes)

\( t_{bin1} \) ... guess for \( t_{bin} \) (used in iteration)

\( t_{film} \), \( t_{film} \) ... film temperature at which properties are evaluated

\( t_{tube} \) ... tube wall temperature

\( t_{nozd} \) ... nozzle diameter (m)

\( r_{nozd} \) ... Reynolds number based on nozzle diameter and nozzle exit velocity

\( r_{zn} \) ... nozzle-to-target separation distance (m)

\( r_{emiss} \) ... radiative emissivity of wall

\( r_{dtarg} \) ... hemisphere diameter (m)

\( q_{stag} \), \( q_{rad} \), \( q_{nose} \) ... stagnation, radiative, and nose heat fluxes (English units)

\( r_{nuavo} \) ... average Nusselt number based on correlation

\( k \) ... thermal conductivity of fluid (W/m/K)

\( h_{tgc} \) ... heat transfer coefficient of fluid ((W*m²)/(K))

\( diff \) ... used in iteration

\( r_{mfrnoz} \) ... mass flow rate at nozzle (kg/s)

Inputs required: \( t_{bnoz} \), \( t_{wmax} \), \( r_{nozd} \), \( r_{nose} \), \( m_{fnoz} \), \( ZN \), \( EMISS \)

Output: \( t_{bin} \)

```
Subroutine Wall(choice, tbooz, twmax, nozd, nose, mfrnoz, zn, emiss, ttube, tbin, epsilon)
Real tbooz, twmax, tmax, nozd, nose, mfrnoz, rnozd, rzn, emiss, ttube,
     tbin, pi, sigma, theta, area, hwall, qstag, grad, qnose, mfrnoz,
     nuavo, tbin1, tfilm, k, htc, diff, tfilm, epsilon

Integer: choice
```

Variables independent of fluid temperature are calculated and converted into SI units

\( \pi = 3.141592654 \)

\( \sigma = 5.67 \times 10^{-8} \)

\( \theta = 85.0 \times (\pi/180.0) \)

"...1" quantities represent English units in use; note that \( h_{wall} \)
and \( q_{stag} \) are both in English units. They are converted into
m/s units in \( q_{nose} \) by .2931.

\( r_{nose1} = r_{nose} \times (3.281) \)

\( d_{dtarg} = r_{nose} \times 2.0 \)

\( area = 2.0 \times \pi \times (r_{nose} \times 2) \times (1.0 - \cos(\theta)) \)

\( twmax = twmax \times 1.8 \)

\( h_{wall} = 0.235 \times twmax \times 1 + 1.0E-05 \times (twmax \times 1 \times 2) \)

\( q_{stag} = ((128.0/((SORT(r_{nose1})))) \times (1.0 - (h_{wall} / 2080))) \)

\( q_{stag} = q_{stag} \times 3600. \)

\( r_{grad} = emiss \times \pi \times (twmax \times 4) \times area \)

\( q_{nose} = q_{stag} \times r_{nose1} \times 2.4816 \times .2931 \)

Correlation is only good for ZN/NOzd = 1e . 7.0

\( t_{film} = 0.5 \times (ttube + tbooz) \)

```
call dynw(choice, tfilm, visc)
revn = (4.0 \times mfrnoz) / (pi \times visc \times nozd)

if ((zn/nozd) < 7.0) then
    nuavo = 2.98 * (revn**(0.585)) \times ((pozd/dtarg)**1.10) * ((zn/nozd)**(0.007))
```

ORIGINAL PAGE IS OF POOR QUALITY
Now we iterate for TBIN (temperature of the fluid exiting the manifold)

\[
tbin = 1.1 \times \text{tbnoz} \\
tbin = tbin \\
tfilmn = 0.5 \times (0.5 \times (tbin1 + \text{tbnoz}) + 0.5 \times \text{twmax}) \\
call \ \text{THCOND(choice, tfilmn, k)} \\
htc = (\text{nua} \times g) / \text{nozd} \\
tbin = \text{tbnoz} + ((\text{qnose} - \text{qadv}) / (\text{area} \times htc)) \\
\text{diff} = \text{ABS}(tbin - tbin1)
\]

Can change the next line for a more precise & time-consuming iteration

\[
\text{if (diff.gt.epsilon) go to 10} \\
\text{return} \\
\text{end}
\]

```
subroutine thcond(choice, tmav, k)
   real k,lk,a1(10),a2(2),a3(2),tmav
   integer choice
   date a2
   *  /3.498e-3,3.1929e-4/ \\
   date a3
   *  /0.16381,-6.329e-4/ \\
   if (choice.eq.1) then
      a = 1.6030e+02 \\
b = -0.64379 \\
c = 4.1102e-03 \\
d = -7.679e-05 \\
e = 6.277e-09 \\
f = -1.819e-12 \\
k = a+(b*tmav)+(c*(tmav**2))+(d*(tmav**3))+(e*(tmav**4))+(f*(tmav**5)) \\
k = k * 1e-03 
elseif (choice.eq.2) then 
   call fit4(tmav, a2, k) 
else 
   call fit4(tmav,a3,k)
endif
return
end
```

```
subroutine specht(choice, tmav,cp)
   real a1(10),tmav,cp 
```

if (choice.eq.1) then
    a = 5.9055e+03
    bb = 1.4564e+02
    c = -0.78868
    d = 1.8103e-03
    e = -1.835e-06
    f = 6.7750e-10
    
    Cp = a + (bb*tmav) + (c*(tmav**2)) + (d*(tmav**3)) + (e*(tmav**4)) +
        (f*(tmav**5))
elseif (choice.eq.2) then
    if (tmav.lt.280.) then
        cp = 3930.
        return
    else
        call fit4(tmav,a2,cp)
    endif
else
    CP = 35.2358 * (Tmav ** 0.6687)
endif
return
end

subroutine dynv(choice,tmav,mu)
    real mu,visc,a(10),a2(2),a3(2)
    integer choice
    data a2
    * /14.945,-0.02994/
    data a3
    * /1.01E5,-2.07e-2/
    if (choice.eq.1) then
        a = 2.0053e+02
        b = -1.30242
        c = 3.5588e-03
        d = -3.205e-06
        e = 9.3978e-10
        
        mu = a + (b*tmav) + (c*(tmav**2)) + (d*(tmav**3)) + (e*(tmav**4))
        mu = mu * 1.0E-07
        return
    elseif (choice.eq.2) then
        call fit4(tmav,a2,mu)
        return
    else
        if (tmav.le.300.) then
            mu = 10 * 2.7355e37 * (tmav ** (-16.5512))
        else
            call fit4(tmav,a3,mu)
            mu = 2.6749e9 * (tmav ** (-4.7321))
        endif
    endif
    return
end

subroutine dense(choice,tmav,ro)
    real a2(2), a3(8), ro, tmav
    integer choice
data a2
   /1.1719E3,-4.142E-04/

if (choice.eq.1) then
   a = 1.0374e+02
   b = -0.9265
   c = 3.7198e-03
   d = -7.189e-06
   e = 6.5313e-09
   f = -2.235e-12

   ro = a+(b*tmav)+(c*(tmav**2))+(d*(tmav**3))+(e*(tmav**4))+(f*(tmav**5))

   return

elseif (choice.eq.2) then
   call fit4(tmav,a2,ro)
else
   ro = 1173.0803 - (0.5093 * Tmav) - (2.795E-04 * (Tmav ** 2))
endif
return
end

subroutine fit4(tmav,a,answer)
   real tmav, a(2), answer
   answer = a(1) * (2.718281828 ** (a(2) * tmav))
return
end
Appendix 3: Design Results
Appendix 3.1: Hydrogen Design
<table>
<thead>
<tr>
<th>Branch #</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>150</td>
</tr>
<tr>
<td>101</td>
<td>151</td>
</tr>
<tr>
<td>102</td>
<td>152</td>
</tr>
<tr>
<td>103</td>
<td>153</td>
</tr>
<tr>
<td>104</td>
<td>154</td>
</tr>
<tr>
<td>105</td>
<td>155</td>
</tr>
<tr>
<td>106</td>
<td>156</td>
</tr>
<tr>
<td>107</td>
<td>157</td>
</tr>
<tr>
<td>108</td>
<td>158</td>
</tr>
<tr>
<td>109</td>
<td>159</td>
</tr>
<tr>
<td>110</td>
<td>160</td>
</tr>
<tr>
<td>111</td>
<td>161</td>
</tr>
<tr>
<td>112</td>
<td>162</td>
</tr>
<tr>
<td>113</td>
<td>163</td>
</tr>
<tr>
<td>114</td>
<td>164</td>
</tr>
<tr>
<td>115</td>
<td>165</td>
</tr>
<tr>
<td>116</td>
<td>166</td>
</tr>
<tr>
<td>117</td>
<td>167</td>
</tr>
<tr>
<td>118</td>
<td>168</td>
</tr>
<tr>
<td>119</td>
<td>169</td>
</tr>
<tr>
<td>120</td>
<td>170</td>
</tr>
<tr>
<td>121</td>
<td>171</td>
</tr>
<tr>
<td>122</td>
<td>172</td>
</tr>
<tr>
<td>123</td>
<td>173</td>
</tr>
<tr>
<td>124</td>
<td>174</td>
</tr>
<tr>
<td>125</td>
<td>175</td>
</tr>
<tr>
<td>126</td>
<td>176</td>
</tr>
</tbody>
</table>

**ORIGINAL PAGE IS OF POOR QUALITY**
Results and initial parameters:

Coolant considered: Liquid hydrogen

Temperature exiting manifold = 0.1482E+03 [K]
Final exiting temperature = 0.7039E+03 [K]
System pressure loss = 0.1038E+07 [Pa] 0.1505E+03 [psi] 10.24 atm
Initial number of tubes = 24
  Tube diameter = 4.88 [mm]
  Tube thickness = 0.76 [mm]
  Nose radius = 25.4 [mm]
  Wall thickness = 2.0 [mm]
  Cone half angle = 5.0 [deg]
  Nozzle mass flow rate = 3.5000 [kg/s]
Velocity of coolant in nozzle = 806.279 [m/s]
Outlet tube mass flow rate = 0.0046 [kg/s]
Outlet vel of coolant in tubes = 281.257 [m/s]
Impingement target distance = 25.4 [mm]
Impingement nozzle diameter = 14.12 [mm]
Number of branches = 5
Limiting material temperature = 1000. [K]
Density of Liquid Hydrogen

\[ y = a + bx + cx^2 + dx^3 + ex^4 + fx^5 \]

Values fit for 20 MPa Isobar

\[ a = 1.6374 \times 10^{-2} \]
\[ b = -8.9265 \]
\[ c = 3.7148 \times 10^{-3} \]
\[ d = -7.189 \times 10^{-6} \]
\[ e = 6.5313 \times 10^{-9} \]
\[ f = -2.235 \times 10^{-12} \]
Thermal Conductivity of Liquid Hydrogen

![Graph showing thermal conductivity vs. temperature](image)

Equation fit for 30 MPa Isobar:

\[ y = a + bx + cx^2 + dx^3 + ex^4 + fx^5 \]

- \( a = 1.603 \times 10^{-2} \)
- \( b = -0.64379 \)
- \( c = 4.1192 \times 10^{-3} \)
- \( d = -7.674 \times 10^{-6} \)
- \( e = 0.23779 \times 10^{-9} \)
- \( f = -1.819 \times 10^{-12} \)
Appendix 3.2: Ethylene Glycol Design
<table>
<thead>
<tr>
<th>#</th>
<th>Re:</th>
<th>tw:</th>
<th>to:</th>
<th>tw:</th>
<th>branch #</th>
<th>t crit:</th>
<th>p critical:</th>
<th>m: 150kg/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.150</td>
<td>0.63E+07</td>
<td>3131E+06</td>
<td>15E+07</td>
<td>39E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>2</td>
<td>0.150</td>
<td>0.63E+07</td>
<td>3131E+06</td>
<td>15E+07</td>
<td>39E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>3</td>
<td>0.250</td>
<td>0.69E+07</td>
<td>3111E+06</td>
<td>15E+07</td>
<td>38E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>4</td>
<td>0.250</td>
<td>0.69E+07</td>
<td>3111E+06</td>
<td>15E+07</td>
<td>38E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>5</td>
<td>0.250</td>
<td>0.69E+07</td>
<td>3111E+06</td>
<td>15E+07</td>
<td>38E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>6</td>
<td>0.250</td>
<td>0.69E+07</td>
<td>3111E+06</td>
<td>15E+07</td>
<td>38E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>7</td>
<td>0.250</td>
<td>0.69E+07</td>
<td>3111E+06</td>
<td>15E+07</td>
<td>38E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>8</td>
<td>0.250</td>
<td>0.69E+07</td>
<td>3111E+06</td>
<td>15E+07</td>
<td>38E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>9</td>
<td>0.250</td>
<td>0.69E+07</td>
<td>3111E+06</td>
<td>15E+07</td>
<td>38E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>10</td>
<td>0.250</td>
<td>0.69E+07</td>
<td>3111E+06</td>
<td>15E+07</td>
<td>38E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>11</td>
<td>0.250</td>
<td>0.69E+07</td>
<td>3111E+06</td>
<td>15E+07</td>
<td>38E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>12</td>
<td>0.250</td>
<td>0.69E+07</td>
<td>3111E+06</td>
<td>15E+07</td>
<td>38E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>13</td>
<td>0.250</td>
<td>0.69E+07</td>
<td>3111E+06</td>
<td>15E+07</td>
<td>38E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
<tr>
<td>14</td>
<td>0.250</td>
<td>0.69E+07</td>
<td>3111E+06</td>
<td>15E+07</td>
<td>38E+03</td>
<td>47E+03</td>
<td>0.39E+03</td>
<td>3.27E+07</td>
</tr>
</tbody>
</table>

**ORIGINAL PAGE IS OF POOR QUALITY**
Results and initial parameters:

Coolant considered: Water/glycol solution (30%)

Temperature exiting manifold = 0.3708E+03 [K]
Final exiting temperature = 0.4181E+03 [K]
System pressure loss = 0.9805E+07 [Pa] 0.1422E+04 [psi] 96.75 psi
Initial number of tubes = 20
  Tube diameter = 6.50 [mm]
  Tube thickness = 0.70 [mm]
  Nose radius = 25.4 [mm]
  Wall thickness = 2.5 [mm]
  Cone half angle = 5.6 [deg]
Nozzle mass flow rate = 150.0000 [kg/s]
Velocity of coolant in nozzle = 953.126 [m/s]
Outlet tube mass flow rate = 0.4687 [kg/s]
Outlet vel of coolant in tubes = 224.682 [m/s]
Impingement target distance = 25.4 [mm]
Impingement nozzle diameter = 14.12 [mm]
Number of branches = 4
Limiting material temperature = 1000. [K]
Appendix 4: Hydrogen properties
# Appendix 4: Hydrogen Properties

## Table 4.12 (continued)

### Thermodynamic Properties of Normal Hydrogen

<table>
<thead>
<tr>
<th>Temperature (°C)</th>
<th>Volume (cm³/kg)</th>
<th>Isothermal Derivative (cm³/kg/°C)</th>
<th>Isochoric Derivative (cm³/kg/°C)</th>
<th>Internal Energy (cal/kg)</th>
<th>Enthalpy (cal/kg)</th>
<th>Entropy (cal/kg °C)</th>
<th>CV (cal/kg °C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1.0115</td>
<td>1.0089</td>
<td>1.0264</td>
<td>0.004</td>
<td>0.005</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>1</td>
<td>1.0120</td>
<td>1.0103</td>
<td>1.0269</td>
<td>0.004</td>
<td>0.005</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>2</td>
<td>1.0125</td>
<td>1.0125</td>
<td>1.0273</td>
<td>0.004</td>
<td>0.005</td>
<td>0.004</td>
<td>0.004</td>
</tr>
</tbody>
</table>

*Note: The table continues with similar entries for different temperatures.*
<table>
<thead>
<tr>
<th>Table 4.12 (continued)</th>
</tr>
</thead>
<tbody>
<tr>
<td>THERMODYNAMIC PROPERTIES OF NORMAL HYDROGEN</td>
</tr>
<tr>
<td>TEMPERATURE °C</td>
</tr>
<tr>
<td>------------------</td>
</tr>
<tr>
<td>40.00</td>
</tr>
<tr>
<td>41.00</td>
</tr>
<tr>
<td>42.00</td>
</tr>
<tr>
<td>43.00</td>
</tr>
<tr>
<td>44.00</td>
</tr>
<tr>
<td>45.00</td>
</tr>
<tr>
<td>46.00</td>
</tr>
<tr>
<td>47.00</td>
</tr>
<tr>
<td>48.00</td>
</tr>
<tr>
<td>49.00</td>
</tr>
<tr>
<td>50.00</td>
</tr>
<tr>
<td>51.00</td>
</tr>
<tr>
<td>52.00</td>
</tr>
<tr>
<td>53.00</td>
</tr>
<tr>
<td>54.00</td>
</tr>
<tr>
<td>55.00</td>
</tr>
<tr>
<td>56.00</td>
</tr>
<tr>
<td>57.00</td>
</tr>
<tr>
<td>58.00</td>
</tr>
<tr>
<td>59.00</td>
</tr>
<tr>
<td>60.00</td>
</tr>
</tbody>
</table>

**ORIGINAL PAGE IS OF POOR QUALITY.**
Appendix 5: Hypersonic Cruise Vehicle Configuration
Appendix 5:

Hypersonic Cruise Vehicle (HCV) Configuration (written by Lura Kern)

It was determined early in Winter Quarter 1988 that the Advanced Aeronautics Design Group (AAD) needed a unifying concept for the design of the hypersonic drone. This was accomplished by choosing a configuration that had been researched extensively at NASA Langley Research Center in the late 1970's. The advantage of a single tested configuration for the entire class is clear: each group would base their designs on the same vehicle, with the same restrictions applying to all three groups, thereby forcing greater interaction among the groups.

The configuration chosen by the AAD students was a 70 degree swept delta wing design conceived by NASA Langley engineers. It was a hypersonic research airplane concept, and has been dubbed the Hypersonic Cruise Vehicle (HCV) by the AAD students. Figure 1 shows the HCV as it was originally conceived. The HCV was designed to be air-launched from a B-52 at Mach 0.8 and rocket-accelerated up to a cruise speed of Mach 7 by four liquid hydrogen rocket motors mounted in the base. At this point, the scramjets would be ignited and the vehicle would cruise until burn-out, at which point it would return to base for an unpowered landing. A detailed description of the HCV's configuration is given in Figure 2. In developing the HCV design, a wind tunnel model was built, and performance characteristics of the model were collected and tabulated in Reference 1. The geometric characteristics of the model are tabulated in Table 1. The scaling for the model was 0.021 of the size conceived for the actual HCV.

The first step in adapting the HCV for the purpose of a hypersonic drone was to size it according to the needs of the class. This included calculating the amount of fuel required to go from the air-launch speed to cruise speed. From that, size and weight necessary for the fuel requirements were determined. In order to do this, it became necessary to determine a means of air-launch. The B-52 has size and weight constraints consisting of a 24-ft. wing span limit, a 9-foot fuselage...
height limit, and a gross weight restriction of 70,000 pounds. This limits the size of the drone considerably and, in order to reach Mach 10, it was found that the weight limit was exceeded. Alternatives were then examined. The 747 and B-1 aircraft were determined to be viable options for the air-launch.

Therefore, the HCV is a basis upon which each group can build their design. It gives the Flight Systems group the geometry and spacing of the vehicle so that they can determine which sensors and how many are required in certain places. This would result in a more detailed and more exact design. The Thermal Management group can determine critical areas on the HCV that would require cooling. They can change the design within certain reasonable limits to satisfy their cooling system design. A definite configuration for the drone gives the Propulsion group a starting point on the design of the underbelly for the vehicle; that is, a first iteration could be performed from which they refined their design to achieve an optimum configuration.
FLIGHT SYSTEMS GROUP

Group Members:
Alan Beck
Erik Berg
Aileen H. Honka
Mike Tsirulnikov
Eugene Wang
S. Willet
Ching Yu

Faculty Advisor:
Professor N. Levan
Introduction

Composing one third of the UCLA AAD program is the Flight Systems Group. The purpose of this group is to design, find, and adapt the necessary sensors and determine methods of integrating these sensors with the structure of the proposed drone envisioned by the class. The sensors to be designed are to complement the unique needs of the Propulsion and Thermal Management Groups.

The hypersonic flight regime in which the drone is designed to be flown and tested makes the design of the vehicle and its systems particularly challenging. The extremely high speed environment will lead to a host of problems, including high temperatures, pressures, and static and dynamic loads. From the viewpoint of sensor systems, many of the classic methods are inadequate in this region. For example, because of the huge heat fluxes, external probes are not usually feasible. Instead, non-intrusive methods need to be examined.

The parameters to be measured that were studied by the group were determined by the needs of the other two groups. These include temperature, pressure, flow constituents and concentrations, velocity, and skin friction. Sensors were designed from such diverse fields as quantum electronics, semiconductors, and fiber optics. The sensors researched originated from a variety of environments, from medical laboratories, to experimental laboratories, and our own class members.

This report will now detail the various sensors researched, developed, and adapted by the Flight Systems Group for the hypersonic drone design. The report begins with the very flexible and powerful sensor system called CARS. Next are two conceptual designs for shock-wave and pressure sensors, followed by a fiber optic pressure sensor. A hydrogen-concentration sensor realized using a modified semiconductor device, and a device using a sapphire diaphragm. The final sensor detailed is a laboratory sensor for measuring skin friction such as that encountered in the combustor of the scramjet.


Coherent Anti-Stokes Raman Spectroscopy

Coherent Anti-Stokes spectroscopy, or CARS, is a laser technique which is capable of measuring velocity, temperature, and constituents of gaseous flow. This is done by measuring the returning characteristics of emitted laser light from the molecules. The range of these parameters (velocity, temperature, pressure, and species concentration) vary under specific conditions. Available data reveal as follows:

A. Temperature and Pressure (in N2 gas)
   In N2 gas, measurement error is estimated to be less than 1.5% in time-averaged measurements, and from 3% to 6% for instantaneous measurements, over a wide range of temperature and pressure.

B. Species Concentration
   Species concentration sensitivity is greater than 1% to 5%, over a limited range of concentration (0.5% to 20.% depending upon the species). Usually, this is sufficient for practical combustion systems.

C. Velocity
   Velocity of a flow constituent, or species, is determined from the Doppler shift of the returning beam, and hence, is subject to the limitations in range of detection possible for this type of frequency shift, if any.

D. Pressure
   Though pressure is probably better dealt with using Inverse Raman Spectroscopy (IRS), a CARS system may be capable of measuring variations of pressure in a flow.

The accuracy of the CARS measurements depends largely on the prevailing conditions, the type of CARS used, and the method of processing the CARS data. However, good accuracy of readings under conditions of interest to the Flight Systems Group have been obtained:
"The velocity in a supersonic jet flow has been measured using Continuous Wave CARS and agrees to within 4 percent of the calculated value." (ref. 8).

The dimensions of the CARS apparatus, i.e. its size and weight, depend upon the type of laser used, as well as the geometry of the optics technique used to generate and process the CARS signal. The laser can be made fairly small, thus, it would not have as much influence on the size of the CARS set-up as the rest of the apparatus. A reasonable estimate for the area of the entire set-up 3 ft by 2.5 ft. The entire system size does not seem to be a rigid factor, however, since laser technology is moving toward smaller, and smaller, optical equipment.

The weight should not be more than a few pounds for the laser, and a little more for the optics that direct the beam, excluding any anti-vibrational equipment necessary for the implementation of any sort of laser measurement system on board the drone.

Power consumption (excluding the computer analysis power drain) would depend upon the number and type of lasers used. The supersonic jet flow measurements mentioned previously were taken with a 100 mW laser.

The CARS measurement rate varies with the type of CARS, and the laser pulse frequency used. At least one type of CARS can be used for continuous measurement. This requires a continuous-wave laser instead of a pulsed laser.

Whatever type of laser is used would definitely require an active cooling system, as very high power would be used.

Finally, we consider the placement and distribution of the system. The laser would be inside of the drone in a cooled area. For measuring wide areas of flow, a rotating platform would enable one CARS set-up to suffice. Or, alternatively, more than one clear window could be installed, and the emerging CARS beam re-directed through any of these temperature-resistant quartz windows. (Fig. 1)

CARS has several advantages that make it a good sensor for the drone. For instance, CARS will perform a few types of measurements at once, is generally non-intrusive, and the beam is not damaged by intensive heat. It is possible that CARS need not
have any apparatus to reflect the beam back from the medium it measures. This is due to the fact that the molecules in the medium, themselves, generate a laser-like beam.

The limitations of the CARS technique depends on which type of CARS is used. Some types work well in high pressure, turbulent flow, while others will not work under those same conditions. Most of the major limitations of CARS are a result of pressure gradient problems (the beam is refracted by pressure gradients, resulting in "beam steering" problems).

CARS will satisfy the needs and specifications of the Propulsion and Thermal Management groups. For example, in the air flow boundary layer outside of the drone, refraction and reflection ("beam steering") could possibly be used to our advantage. This is because the return beam is reflected/refracted back exactly in the same path. CARS spectra, and some geometry, can give us temperature and boundary layer data.

CARS can also measure flow constituents in the combustor nozzle, the mixing rate and turbulence of the fuel, and the concentration of elements downstream and upstream of ignition.

The most important aspect of CARS is the Raman effect. Laser light has a single frequency and phase quality. When light of a particular frequency hits a molecule, it emits a laser-like beam with the same qualities of the original beam, except that the emitted light is shifted by a frequency that is unique to that particular molecule or atom. This phenomenon is known as the Raman effect.

When this light hits another molecule of the same type, the probability of the molecule emitting a Raman shifted beam, originally low, greatly increases. The effect is amplified as the beam traverses through a sample of these molecules.

The beam is returned either by a reflecting apparatus at the other end, or by some technique that directs the beam back towards the source. The intensity, phase, and rotational direction of the beam and its spectrum gives information regarding temperature, concentration, and pressure of the molecules hit by the laser. Most of this information is determined by comparing the beam spectrum
to some theoretical or reference spectrum. Velocity is measured by noting the Doppler shift in the returning beam frequency.

Coherent Anti-Stokes Raman Spectroscopy, or CARS, is a laser spectroscopy technique. Spectroscopy is the study of spectra obtained from a spectroscope. A spectroscope is an apparatus that takes incoming light and by using a prism or diffraction grating forms a spectrum by separating the various wavelengths of the light into bands of varying intensity. The wavelength of a particular band can be determined from calculations using the position where the band occurs in the spectrum. How well these bands are distinguished from one another determines the resolution of the spectrum.

In CARS, two or more lasers are combined using a method called Optical Wave-Mixing to generate a single laser beam. This beam has a frequency that molecules in a media will resonate when hit. The enhanced beam that results from the molecule is the CARS beam or signal. This resonant response is the result of the Raman effect discussed earlier. Spectral analysis of the returning CARS signal can reveal information about the temperature, species concentration, pressure, and velocity of the molecules that it originates from.

The two or more beams that are optically wave-mixed are subject to certain requirements. For example, in two-laser CARS one laser is chosen to be the "pump" laser. The other laser is known as the "Stokes" laser because its frequency is shifted in the "red" direction of the electromagnetic spectrum with respect to the pump laser. The Stokes frequency is not fixed, and can take on different values.

The wave-mixed beam goes through the medium to be examined and results in a CARS beam and frequency. This is the "Anti-Stokes" beam, because its frequency is twice the pump frequency less the Stokes frequency. It is shifted towards the blue end of the electromagnetic spectrum with respect to the pump beam.

There are two types of Stokes lasers used in CARS methods: narrowband and broadband. These two ways of generating the Stokes frequency separates CARS into two groups which are named
according to the laser used. Narrowband (also known as "scanning") CARS builds the CARS spectra piece by piece by "tuning" the laser to a small range of Stokes frequencies, taking a measurement for each different frequency. This type of CARS is generally restricted to steady-state flow.

Broadband CARS, in contrast, will generate a complete spectrum for a given range of the Stokes frequency in one measurement. While a narrowband laser will output only a very narrow range of frequencies, a broadband laser outputs a wide range of frequencies. This type of CARS is necessary when measuring "randomly time-varying flows", since the sample could change from one point in time to the next.

Both scanning and broadband CARS usually examine only one species in a sample at a given time. There is a method that is capable of measuring all of the major constituents in a flow simultaneously. It is called Dual Broadband CARS. This method seems to hold the most promise to fulfill the sensor requirements for the hypersonic drone.

Dual Broadband CARS is one of a group of CARS techniques that utilize more than one pump and/or Stokes lasers to create the CARS beam. These are multi-frequency (multi-color) methods, since each laser operates at its own frequency. This method also contains the most potential for high pressure measurements which are expected for the combustor of the drone.

Alan C. Eckbreth and Torger J. Anderson, of United Technologies Research Center developed Dual Broadband CARS. "The technique employs two independent broadband Stokes sources in combination with a pump laser. In addition to the two-color wave-mixing processes between the pump and Stokes lasers, spectrally resolved CARS is produced in a three-color process from species whose Raman resonances correspond to the frequency difference of the two broadband sources. CARS is thus derived from a large number of species simultaneously removing the nominal limitations of CARS to interrogate only one constituent at a time." (Ref. 4)

For the Flight Systems Group purposes, the process of two and three color wave mixing is not so important as is the fact that two
broadband Stokes sources are used with a pump laser. The result is the capability to measure more than one species in a sample at a time.

In the article "Multi-Color CARS for Simultaneous Measurements of Multiple Combustion Species", Eckbreth and Anderson outline three other multi-color techniques as well as rotational/vibrational CARS methods. They are Dual Broadband, Dual Stokes, Dual Pump, Dual Pump-Stokes CARS, Pure Rotational and Combined Rotational/vibrational CARS.

Dual Stokes and Dual Pump CARS are used primarily for measuring two flow constituents at once. Measuring three or more are theoretically possible, but the application is too complicated to be practical. Both of these approaches result in simpler CARS signal detection schemes.

Dual Pump-Stokes CARS is interesting because the Stokes bandwidth can be made very narrow. In hydrogen air combustion, this method can then "track the appearance of H2O product in the reaction". This means that it could keep track of how much fuel is burned in the drones' scramjets.

Pure Rotational CARS is known for its high accuracy in measuring temperatures below 1000 K, even at high pressures. This method is difficult to apply. It uses the rotational approach on Dual Broadband CARS, making it simpler to implement. The advantages of this is the elimination of a Stokes laser, no calibration for the pump laser, and the pump frequency can be arbitrarily chosen for maximum power output.

Combined Vibrational/Rotational CARS combines Vibrational CARS with Rotational CARS. The result is a method that will measure two constituents at once over a very wide range of temperatures.

After considering the assets of all the CARS techniques, Dual Broadband CARS looks like the most promising as a sensor for the hypersonic drone. The major drawback is that there will be a loss of signal intensity in the CARS beam as compared to "regular" CARS. "Due to the increase of the CARS signal with increasing pressure, Dual Broadband CARS should be capable of single pulse
measurements of the major constituents at elevated pressures typical of gas turbines, internal combustion engines, and burning propellants. Furthermore, in Dual Broadband CARS, a beneficial spectral averaging may occur resulting in improved single pulse spectral quality. It is also noted that "dual Broadband CARS may be restricted, particularly for dynamic measurements, to elevated pressure combustion applications. Signal decrease accompanying broadband sources is compensated by ...scaling... with density". (Ref. 5,6)

This suggests that the use of Dual Broadband CARS in the high pressure scramjet engines to monitor the temperature, velocity, and species concentration of the flow there. A remote sensing system is possible for measuring velocity, pressure, and temperature outside of the drone as well.

To summarize, several CARS diagrams have been included. One of these is a "typical" CARS system, and also a configuration for measuring the after-burn of jet engine exhausts. After primarily examining the needs of the Propulsion Group, Dual Broadband CARS was selected as the method most capable of fulfilling them.
CARS Supporting Optics System

Having decided upon which type of CARS to implement, the manner which this was to be done remained to be developed. The following optics arrangement was designed to make the most of the CARS laser spectroscopy system as applied in the scramjet combustor. (Fig 2.)

The first is a result of our desire to check flow patterns in the scramjet in several areas. The optics arrangement is capable of achieving this goal because it uses the idea of one CARS system directed out of many windows.

The second way the optics arrangement optimizes the CARS system is by making the most of the available laser power coming from the optically wave-mixed CARS beam. This beam is split up between these windows in terms of fractions of the original beam pulse(s), as opposed to dividing a single pulse of the beam between all of the windows simultaneously.

The third way is that the CARS system is physically connected to a single, outer structure. This minimizes the effect of the aircraft's vibrations on the data, since all of the components of the optics system will vibrate relative to the drone and not each other. Further, the system may operate at such high speeds (picosecond pulses, for example) that, in comparison, any aircraft-related vibration would be insignificant. More precisely, if pulses are generated fast enough, data could possibly be gathered between one vibration and the next.

Thus, we hope to achieve a system capable of mapping at least part of the scramjet combustor flow, with a minimum amount of power loss, and in the most accurate way possible.

The way the CARS supporting optics system (CARS S.O.S.) operates is rather complicated. The CARS beam is generated in a separate system that is housed inside the main structure of the CARS S.O.S. As this beam comes out of the generating system, it goes through an aperture (marked "A" in the figure) and into the Supporting Optics System. As it travels horizontally into the system, the beam is directed to the first rotating shaft beam
splitter (marked "C") by a totally reflecting mirror ("B"). On this shaft, and on the second, tilted axis shaft below it, are disks which both allow the transmission and reflection of the beam. These disks are designed such that the beam will reflect off each one, in succession, for a predetermined pulse length, $L$. This pulse length is determined by the amount of reflecting disk that is cut away in the disks between it and the laser aperture. For a cut away length of circumference, $d$, assuming the laser beam traces this circumference, a pulse will be generated of length,

$$L = \frac{c \cdot d}{\pi \cdot r \cdot f}$$

where:
- $r$ = distance of beam from disk's center
- $c$ = the speed of light in a vacuum
- $f$ = the frequency of rotation of the shaft about its axis.

Note that the pulse length is dependent on the distance of the beam from the center of disk and not necessarily on the radius of the disk. The radius of the disk only determines the maximum $r$. This makes the optical design quite a tricky geometry problem, since the radius of a disk depends on the position of the disk on its shaft relative to the incoming beam, the horizontal spacing of the windows, and the amount of room between the horizontal and tilted axis rotating shafts. Thus, the pulse length may be changed by change in axis frequency, but the other parameters are fixed by geometrical requirements.

Once the CARS beam is split into a pulse by the horizontal shaft arrangement of disks, the pulse travels to its reflecting horizontal disk which directs it to the appropriate second, tilted axis shaft disk. These tilted axis shaft disks, also, alternately reflect and pass the beam pulse -- only these reflect the pulse down through a collimating lens (of 300 mm in focal length) and through one of the temperature-resistant quartz windows into the scramjet combustor. There, a CARS signal is generated by the specific, excited molecules, and travels directly up to the tilted shaft disk which first reflected it down. As soon as the pulse is over and the CARS laser is blocked from the reflecting horizontal
disk's view, the tilted shaft disk allows the beam to travel straight up to an optical detector.

If the pulse is long, since the CARS stimulated molecule beam is approximately instantaneously generated, an additional detector may be placed around the aperture of the original beam to collect any overlapping signal that occurs during the pulse generation. Any overlap could then be routed to the appropriate spectral analysis channel. It is possible to use this same system with an entirely reflecting second shaft and one photodetector at the CARS beam aperture. However, having separate photodetectors for each pulse has the possible advantage of easier signal separation at high pulse frequencies and faster electronic processing of the almost continuous wave of CARS returning pulses.

Note that only one pulse will be reflected up from the scramjet at any moment, in either case. Hence, one spectrograph and computer, operating continuously, could receive all of the data. It may, however, take more than one computer to store, analyze, and process the information at the speed required.

Note, also, that other means could have been used to make the division of the CARS beam between the windows much simpler, but with a considerable power loss increase. Beam splitters which partially reflect and transmit simultaneously, could have been used to split the original beam. But such beam splitters, used twice in succession, would have reduced the outgoing beam intensity to 25% of the original CARS beam.

However, in return for conserving power and the other optimization inherent in its design, the proposed Supporting Optics System has its own particular drawbacks. First, the size of the optics system is very much dependent on the spacing of the windows and number of windows used. A fixed amount of space is necessary to contain the spinning disks and to make sure that the two spinning shafts do not collide with one another.

Also, more power is consumed than in a stationary optics set-up, since power must be supplied to the shaft rotary motors. Further, the very fast rotation of the shafts would generate heat, which would add to the overall problem of cooling the CARS.
Lastly, such a system would be more prone to mechanical failure than a stationary system using half-reflecting/half-transmitting mirrors.

In summary, however, even with these drawbacks (increased power consumption, greater cooling requirements, and possible mechanical failure), the proposed optics arrangement for the application of CARS in the drone's scramjet combustor should be able fulfill the requirements it was designed to to a satisfactory degree. It will allow the measurement of combustor flow characteristics in several different areas in the combustor, provide the maximum possible power to these measurements (and, hence, the more accurate results), and minimize the effects of any aircraft-related vibration.
The next two devices described are different than the other sensors. Both use novel pulse-laser techniques that were designed conceptually from scratch. Thus, there is considerable ambiguity as to the actual feasibility of the devices since neither the dollar cost nor any available substitutes have been considered. Design is strictly conceptual that can be used as a springboard for further research.

An Optical Pressure Sensor

This device (shown in Fig. 3) uses a Michelson-type interferometric technique to measure the displacement of a material, such as quartz or graphite, under the load of substantial pressure. The sensor does so by an arrangement of thin aluminum mirrors of proper reflectivities embedded in the bulk material through which a laser beam is launched. Fringes are formed as pressure displaces the upper mirror downward towards the laser source. These fringes are then counted and the measure of bulk material displacement under the external load is then calculated. Through accurate calibration or theory, the displacement of the mirror, which moves in parallel with the bulk material it is embedded in, is related to the pressure the bulk is exposed to. Therefore such a device is useful in measuring pressures exceeding ordinary pressure sensors, pressure as high as the bulk material can withstand while still retaining its elastic properties. Outside dynamic pressure anywhere on the surface of a hypersonic airplane can be measured with such a device. The data can be sampled at rates above 100 KHz (limited only by the propagation delays of the associated electronics).

This device suffers from noise that results from all mirrors being displaced simultaneously from pressure. One way of correcting this noise-like effect is by inserting a soft buffer layer between the upper mirror and the lower part of the device.
Multiple Shock-Wave Sensor

The operation of the device is dependent of the shock wave creating a boundary layer. The differing physical properties of abrupt pressure and temperature changes across the boundary layer leading to a spatial discontinuity in the index of refraction. A laser beam can be reflected off of the shock wave because of the index discontinuity.

Due to the high hypersonic speeds, the shock waves will "hug" the plane, being nearly horizontal with respect to the surface. A pulsed laser beam is launched upward at right angles to the surface of the drone into the air (as shown in Fig. 4). The laser beam will be partially reflected upon each impact with each shock wave that it encounters. By detecting the back-scattered beam, its intensity-time profile can be used to predict the locations of the shock waves away from the plane. Knowledge of the appropriate permeability coefficient is needed in the calculation of the shock wave distance. This can be easily obtained with a simple in-flight measurement.

Noise from various sources such as the beam oscillating between two shock waves and beam-steering may be misinterpreted for a shock wave. The high temperatures and the near-plasma environment outside make the device difficult to design. It is difficult to single out only the intensity peaks produced by scattering the beam from a shock wave front, effectively rejecting noise contributions. Beam steering effect can lead to significant errors in the measurements due to the turbulent environment expected.

The velocity of the drone can be conceivably predicted from the angle the shock wave makes with the fuselage. By scanning the shock wave with the arrangement described above, such an angle can be computed. Combined with the altitude of the drone, and other basic parameters, the velocity can be predicted.
Fiber Optic Pressure Sensor

Fiber optics are currently being used in the measurement of many quantities, such as temperature, acceleration, velocity, electric and/or magnetic fields, fluid levels and acoustic waves. In the measurement of these parameters, the fiber plays either an active or passive role; active when the fiber itself is the sensing element and passive when the fiber simply transmits information from the sensing element. The device considered here is a pressure sensor in which the fiber takes an active role in the measurement.

The sensor is a flattened hollow cylinder about which is wrapped an optical fiber. At one end of the fiber is a source, an AlGaAs LED with a power consumption of the order of milliwatts. At the other end is a detector, a Si PIN photodiode. The pressure differential between the inside of the cylindrical cavity and the outside of it causes the radius of curvature of the flattened section to decrease from infinity to some finite value. As this radius decreases further, there is a corresponding approximately exponential increase in the attenuation of the intensity of light that reaches the detector. The light attenuation at the curved ends is constant and is taken into account when calibrating the device. The amount of light reaching the detector is therefore a function of the pressure difference between the inside and outside of the cavity. The outside of the cavity could be another cavity at a reference pressure, or a cavity in which the pressure can be varied to increase the range of measurement of the device. See figure 5 for a picture of the fiber optic pressure sensor. (ref. 12)

This sensor can be adapted to a high temperature environment by constructing the cylindrical cavity from titanium and threading the approximately 100 micron diameter fibers through tubes welded to the outside of the cylinder. Water or the liquid hydrogen on board would be pumped through these tubes to keep the fibers at a relatively constant temperature. In this way the device can be located near the surface of the inside walls of the engine or the nose of the craft for a flush air data system to optimize the pressure propagation delay time. The source and detector would be located
away from the high temperatures, fibers of course being used to transmit light to the sensor. In this way, real time measurement of pressures on the surface of the aircraft or inside the engine could be made.

The theoretical sensitivity of this device is given by the following equation:

$$S = \frac{1}{I} \frac{dI}{dP} = S_{opt} a^3 \left( 1 - \frac{\mu^2}{\varepsilon h^3} \right) \beta^2 \left( \beta + \frac{12}{\pi} \right) / \left( \beta + \pi \right)$$

where

$I$ = intensity reaching detector

$p$ = pressure

$S_{opt}$ = normalized optical sensitivity for a constant length $L$ of fiber bent to radius $R$. (ref. 7)

This device will use single-mode fibers rather than multi-mode fibers which were used in the Wright experiment (ref. 20) because of the vibrations of the multi-mode fibers encountered during flight would effect the electromagnetic power propagation and thus pressure measurement. Two advantages are gained, however, with the use of single-mode fibers. First the sensitivity, $S_{opt}$ is an order of magnitude greater, 20 compared to 2. Secondly, with single mode-fibers is proportional to the number of half turns the fiber makes around the flattened cylinder. so that the use of titanium, with two orders of magnitude greater Young's modulus than the epoxy resin of the Wright experiment, causing a two orders of magnitude decrease in the sensitivity can be compensated for with increased half turns of the fiber. Since is already an order of magnitude larger, two extra half turns brings the sensitivity of this device into the range of the Wright experiment, $S = 8$*/bar (1 bar = 100 kPa).

The dynamic range of the epoxy resin cylinder was tested for 200 kPa. This range could, of course, be extended with the use of the titanium cylinder.
MOSFET Hydrogen Concentration Sensor

Hydrogen flow concentration inside the combustor of the SCRAMJET is a very important parameter to be measured because it is related to the rate of combustion, the energy consumption, and the engine efficiency. The sensors used in the are relatively big and intrusive. The MOSFET hydrogen concentration sensor (ref. 3) has many advantages such as small size, low power consumption, high sensitivity, fast response, and relatively non-intrusiveness.

A sensor is a sensor and a transducer combination which senses physical quantities and then converts them through chemical parameters into electrical signals. The acronym, MOSFET, stands for Metal-Oxide-Semiconductor Field-Effect Transistor. It is a four-terminal device as shown in Figure 6.

The operation of a MOSFET is controlled by a vertical action and a horizontal action. The gate-bulk potential difference $V_{GB}$ sets up a capacitive effect which adjusts the amount of carriers (in our case, electrons) in the conducting channel. There will be carriers in the channel if, and only if, the gate-bulk potential is greater than, or equal to, the threshold voltage $V_T$. This is the vertical action. The horizontal action is controlled by the potential difference applied between the drain and the source $V_{DS}$. Current flows from the drain to the source if vertical action causes carriers to be present in the conducting channel. Depending on the value of $V_{DS}$, there are two regions of operation—the linear region and the saturated region. The linear region occurs if $V_{DS} < V_{GS} - V_T$ and the saturated region occurs if $V_{DS} > V_{GS} - V_T$. The drain current equations are given by:

$$ I_D = \mu(W/L)C_{OX} [V_{DS} (V_{GS} - V_T) - V_{DS}^2/2] \quad \text{linear region} $$

$$ I_D = (1/2)(W/L)C_{OX} (V_{GS} - V_T)^2 \quad \text{saturated region} $$

Where

- $\mu$ = mobility of electrons
- $W/L$ = gate width / gate length
- $C_{OX}$ = permeability / thickness of SiO$_2$

Notice that the current in the linear region is less than the current in the saturated region and thus a MOSFET operating in the linear region consumes less power. Also, the fact that the current

- 17 -
in the linear region depends on \( V_{DS} \) makes the linear region more flexible for our design.

In our design, the gate material used is palladium (Pd) instead of polysilicon used in microelectronic devices. Palladium, when in contact with hydrogen flow, acts as a catalyst for \( H_2 \) ionization to give \( 2H^+ \). This \( H^+ \) diffuses into the thin SiO\(_2\) layer to give a charge difference \( \Delta Q \). A \( \Delta Q \) changes the threshold voltage \( V_T \). If at \( \Delta Q = 0 \), \( V_T = V_{TO} \), then \( V_T = V_{TO} - V_T = \Delta Q/\text{COX} \). Now, if we substitute \( V_T = V_{TO} - \Delta Q/\text{COX} \) into the linear region \( I_D \) equation, we have:

\[
I_D = \mu(W/L)\text{COX}[V_{DS}\Delta Q/\text{COX} + V_{DS}(V_{GS} - V_{TO} - V_{DS}/2)]
\]

Notice that for a particular \( V_{DS} \), \( I_D \) is linearly related to \( \Delta Q \) with a slope equal to \( \mu(W/L)V_{DS} \) and the plot is shown in Figure 7.

The senducers can be embedded in the walls of the combustor. The advantage is that although the sensor cannot stand high temperatures, we need not worry about heating effects because the combustor walls will be actively cooled regardless of the presence of the MOSFET senducers.

Note that we can use platinum instead of palladium as the gate material. However, since they act the same for our purpose and that platinum is much more expensive than palladium, palladium is obviously a better choice.

The senducer has the advantage of small size—approximately \( 10^{-15} \text{ m}^3 \), low power consumption on the order of \( 0.1 \text{ mW} \), fast response time on the order of a few nanoseconds, and high sensitivity to \( H_2 \) concentration variation because molecular relaxation takes only a fraction of a nanosecond.

To ensure accuracy, each senducer should be calibrated individually because of the doping variations and difference in threshold voltages. However, if all the senducers are made on the same silicon wafer, this calibration problem can be greatly simplified.
Sapphire Diaphragm for Pressure and Temperature Measurement

A sapphire diaphragm approximately an inch in diameter is placed flush with the aircraft's skin. Pressure measurement with this device is based on the photoelastic effect. Light from an LED source is linearly polarized 45 degrees relative to the surface of the diaphragm and passed across the diameter of it. As pressure is applied to the surface, the linearly polarized light becomes elliptically polarized due to the photo-elastic effect. A second polarizer, again placed at 45 degrees relative to the surface, filters out the elliptically polarized light at the opposite end of the diaphragm. The degree to which the linearly polarized light is attenuated is thus a function of pressure. The accuracy of this device can be increased by using two LED sources and alternately polarizing them +45 degrees and -45 degrees relative to the surface, therefore, obtaining four outputs. The resulting four outputs can then be averaged. Temperature measurement with this device is based on blackbody radiation. As the temperature of a substance increases the amount of radiation emitted increases and the wavelength at which it is emitted decreases. The relation that describes this phenomena is Planck's equation. (ref. 17,18)

Since sapphire is not an ideal blackbody, the above relation must be multiplied by the emissivity of sapphire, 0.15, which is an indication of the degree to which it approaches the ideal blackbody.

Sapphire was the material of choice for several reasons. First and foremost is its high temperature properties. Its melting point is 2030 degrees Celsius with its maximum normal use temperature at 1950 degrees Celsius. It is unaffected by oxidizing and reducing atmospheres at elevated temperatures. Its crystalline structure, trigonal, is identical to that of alpha-quartz which was used in the photoelastic experiments. The tested range of quartz was 10 MPa with a 0.07% accuracy. There is no reason to believe that sapphire would be any less. It is a low power consumption device; the LED's are 100mW sources. And finally, since the diaphragm is flush with
the surface there is no need to wait for the pressure wave to propagate down a tube to the pressure sensor located inside the aircraft.
The Dual-Laser-Beam Skin Friction Interferometer

The skin friction sensor, using a non-intrusive double-laser-beam oil viscosity balance technique, was developed by Monson and Higuchi (ref. 14). Two out-of-phase laser beams, reflected from an oil film, gives alternative constructive and destructive interferences. The number of fringes, and thus the rate of change of the oil film over a given time span, can be precisely computed in terms of the known laser wavelength. The skin friction is related to the rate of change of thickness of the flowing oil film. The oil film used is silicon oil since it has a wide range of viscosity, a relative insensitiveness to temperature, a low surface tension, and a very low vapor pressure. The following paragraphs give a brief outline of its operation and characteristics.

Figure 8 shows the configuration of this device. A He-Ne laser beam is generated. The laser beam is expanded by a telescope, linearly polarized, and passed through a neutral density filter to reduce the beam power level to a value that avoids excessive oil heating. The beam is then split into two, using a interferometric flat. One of the beams is rotated by 90° by the half-wave retardation plate, and both beams are focused on the oil film. The reflected beams are separated by a polarization beam splitter and focused on a photodiode after passing them through polarizers and narrow band interference filters at the detectors for additional noise reductions. The interference patterns are then analyzed to give skin friction.

This device is very accurate and inexpensive. It can be used in three-dimensional laminar or turbulent flow. It is only slightly intrusive and can be used at various points on the combustor surface.

The only problem occurs when there are dust particles in the oil. A dust particle exactly at a beam focal point could cause erratic interferometer behavior. A large dust particle just ahead of a beam focal point could locally perturb either the oil, or the oil flow,
enough to influence the skin friction measurement, but could not be
detected on the interferometer record.
Conclusion

This ends the report for the Flight Systems Group. The sensor system design was approached with an integration with the Propulsion and Thermal Management groups. The challenging environment and the variety of needed measurements called for a equally large variety of sensors from differing backgrounds and applications. These innovative designs should provide an interesting testbed for the sensor systems on the hypersonic drone.
Bibliography


Coherent Anti-Stokes Raman Spectroscopy (CARS)

Schematic CARS Layout

Figure 1
CARS Scanning Setup

Figure 2.

Spectrograph → Optical Multi-Ch. Analyzer → Computer

Detector

Airflow

Combustion Chamber
Interferometric Pressure Detector

Figure 3
Multiple Shock Wave Sensor

Figure 4

optical fiber from pulsed laser source

mirror

optical detector

to reduce noise

A/D

CPU

(1) (2)

(1) (2)

time
Optical Fiber Pressure Sensor

Sealed rear surface of pressure cavity

To detector

From source

(Cooling system not shown)

Optical Fiber

Pressure Port

Wall of combustor or aircraft skin

Figure 5
The structure of a MOSFET

Figure 6
Hydrogen MOSFET Characteristics

\[ I = \mu \left( \frac{W}{L} \right) \left[ (V_{DS} \Delta Q) + V_{DS} C_{OX} (V_{GS} - V_{TO} - V_{DS}) / 2 \right] \]

Figure 7
Dual-Laser-Beam Skin Friction Interferometer

Figure 8