Optimization of Air-breathing Hypersonic Aircraft Design using Euler Codes and Genetic Algorithms

Vivek Ahuja * and Dr. Roy J. Hartfield, Jr. **

Department of Aerospace Engineering, Auburn University, Auburn, Alabama-36830

An advanced, modular paneling scheme has been developed for atmospheric flight vehicles with an aim of predicting the drag during hypersonic flight. A generic, three-dimensional grid generation algorithm has been developed for this effort that allows the construction of detailed aircraft geometries using the concept of geometric overlap in three-dimensional spaces. The use of Bernstein polynomials is used to defining the vehicle outer mold line. An automated grid refinement system has been developed that is fully autonomous in operation and converts the user defined grid into the refined grid by trimming the geometrical internal overlaps and generation of high fidelity surface grids over the exposed outer surfaces of the design airframe. A hybrid group of aerodynamic predictive codes using Euler Methodology for Finite Difference Based fluid dynamic systems has been created to determine the aerodynamic loads on the three dimensional airframe. The use of this aero-predictive system allows for the evaluation of detailed distributions of pressure and velocity over the airframe surfaces. Models have also been added for the determination of skin friction over the exposed surfaces using panel interaction schemes. The results for this preliminary design level drag prediction code have been found to be extremely encouraging in their compatibility to the experimental and flight test data.

Nomenclature

\( u \) = Local velocity at any point in the grid
\( \rho \) = Local density at any point in the grid
\( p \) = Local pressure at any point in the grid
\( E \) = Total energy per unit volume at any point in the grid
\( C_p \) = Specific Heat of the fluid
\( T \) = Local temperature at any point in the grid
\( \mu \) = Viscosity of the fluid at any point in the grid
\( k \) = Constant of Thermal Conductivity for the fluid at any point in the grid
\( M \) = Mach Number of the fluid flow at any point in the grid
\( Re_x \) = Reynolds Number of the fluid flow at any station ‘x’ along the length of the fuselage
\( Pr \) = Prandtl Number of the fluid flow at any station ‘x’ along the length of the fuselage
\( St \) = Stanton Number of the fluid flow at any station ‘x’ along the length of the fuselage
\( q_w \) = Heat Transfer rate of the fluid flow at any point in the grid
\( L_i \) = Mean length of a surface panel in the ‘i\textsuperscript{th}’ numerical direction
\( L_j \) = Mean length of a surface panel in the ‘j\textsuperscript{th}’ numerical direction
\( ipts \) = Number of grid points in the ‘i\textsuperscript{th}’ numerical direction
\( jpts \) = Number of grid points in the ‘j\textsuperscript{th}’ numerical direction
\( V_x \) = Velocity in the airframe reference ‘X’ system
\( V_y \) = Velocity in the airframe reference ‘Y’ system
\( V_z \) = Velocity in the airframe reference ‘Z’ system
\( \alpha \) = Angle of attack
\( U \) = Free stream velocity
\( \rho \) = Free stream atmospheric density
\( \Delta y \) = Panel width in the spanwise direction

* Graduate Research Assistant, Aerospace Engineering, and AIAA Member
** Professor, Aerospace Engineering, and AIAA Associate Fellow
\[ \Delta C_{L,i,j} = \text{Elemental lift coefficient over a surface panel} \]
\[ \Delta C_{D,IND} = \text{Elemental induced drag coefficient over a surface panel} \]
\[ m = \text{Correlation constant between the friction and drag coefficient with a value in the range of 6 to 7} \]
\[ v = \text{Local tangential component of the velocity to the flat panel (at any random orientation)} \]
\[ l_x = \text{Length from the exposed panel at the leading edge to the current panel under consideration} \]
\[ \theta = \text{ramp angle} \]
\[ \beta = \text{Shock angle created by the oblique shock} \]
\[ M_1 = \text{Incoming Mach number of the flow (= 2.5 for the given case study)} \]
\[ n_x = \text{Total number of nodes on the grid inclusive of ghost cells} \]
\[ n_g = \text{Number of Ghost cells on either side of the grid} \]

I. Introduction

Hypersonic flight continues to be the elusive next logical step in aeronautical engineering. However, with the realization of the basics of Scramjet technology via unmanned technology demonstrator vehicles over the last two decades and the advancement of technology in the fields of metallurgy and high enthalpy flow engines, the idea of re-usable hypersonic air-breathing vehicles has begun to reach critical mass. Nevertheless, development of hypersonic speed vehicles continues to be an exercise in multidisciplinary optimization and multi-goal compromises in order to be viable.

Advanced Computational Fluid Dynamics (CFD) software can produce very accurate results but the CFD results come at the cost of time efficiency. Most of today’s CFD software systems are extremely time intensive and require massive computer hardware and networking to effectively and completely model large airframes. As a result, CFD solutions are usually not suitable in the preliminary design stages as a means of predicting the aerodynamic performance of the vehicle, especially when optimization is employed.

The current aero-predictive development effort is aimed at bridging the gap between the classic theoretical drag models and empirical relations with the grid based numerical calculations of the CFD solvers to provide a time efficient, yet accurate model for vehicle aerodynamics. A fully arbitrary three dimensional grid generator has been created to allow the user to pursue highly detailed models of an airframe. These grids resemble CFD grids but the methods employed maintain the computational speed required of preliminary design tools. Paneling methods combined with random three dimensional grids makes the current model highly versatile, fast and more accurate than other similar preliminary design algorithms.

With the advancement of highly accurate fast predictor codes for the aerodynamics and chemistry of high enthalpy flows and the use of robust optimization codes such as Genetic Algorithms, it is now possible to conduct an optimization on hypersonic airframe designs with full three dimensional modeling of prospective vehicles and determine their performance characteristics.

This effort has yielded an extensive Integrated System Architecture of Algorithms (ISAA) making use of three dimensional Euler codes, integrated surface panel skin drag and boundary layer models and advanced grid refinement codes under the aegis of a system management control algorithm. The ISAA is designed to provide numerically integrated solutions for overall skin friction drag, pressure drag and induced drag coefficients references to standard geometry parameters, alongside surface distribution data for the same coefficients to allow visualization of high drag regions etc. In addition, complete aircraft wakes can now be modeled to provide a three dimensional cross section of the flow system around an airframe.

This effort builds on the previous work done by the authors in the field of multidisciplinary optimization and design of hypersonic air vehicles Binary encoded Genetic Algorithms in conjunction with a highly robust three dimensional automated gridding techniques for hypersonic airframes to now include more advanced aerodynamic codes in the form of three-dimensional Euler codes to allow for multiple goal optimizations that include maximizing the thrust, minimizing the drag, increasing the lift, reducing the weight, improving stability characteristics, reducing loads and improving flight time.

A generalized three-dimensional Euler code has been created that combines with the automated three-dimensional generic mold grid generator code also created for this effort. The combined system of codes allows for the evaluation of the aerodynamic properties of any generic hypersonic vehicle. In addition, an advanced propulsion code has been created that builds up on the use of the Euler code solutions inside scramjet combustor ducts to
provide the propulsion performance evaluations of the vehicle. The vehicle inert weight and mission weights have been carefully modeled using structural analysis techniques such as Buckling and Beam theory to allow a realistic value for the required lift. The airframe design has been created to allow transition from all-body designs to wing-body designs under the direction of the Genetic Algorithm control authority.

II. The 3-Dimensional Generic Mold Grid Generator and Hypersonic Designs

For maximum flexibility, a robust three dimensional generic fuselage design model capable of evolving from one type of hypersonic flight configuration to another based on the design point optimization is created and employed in this investigation. Additionally, the engine inlet capture area is crucial to the analysis and the Genetic Algorithm must be provided with the freedom to evaluate and optimize the capture area scramjet engine layout on the fuselage. To achieve this, a parameter driven design code was written in FORTRAN which was capable of performing all of the above tasks. The design process moves in sections from the bow to the aft of the fuselage and from the wing-fuselage interface to the tip of the lifting surfaces.

A full arbitrary three dimensional FORTRAN based paneling scheme has been created for this effort to allow complete and detailed modeling of different aerospace vehicles and structures. The grid generator is set up as a subroutine within the ISAA that reads an input file on the airframe and then creates the three dimensional exposed surface grid for use within the aerodynamics analysis. The input file is set up to provide sufficient data to the Airframe Meshing Algorithm (AMA) to be able to build in detail the external exposed surfaces and also the internal flow passage types on the design vehicle with complete freedom of design choice regarding geometry location and geometric parameters. The Auto-Grid Refinement System (AGRS) is designed to accept this input grid, which may have intersecting surfaces and odd geometries, and convert it into the final version of the grid that enters the ISAA Aerodynamics Package.

The AGRS proceeds by eliminating all internal intersecting surfaces to determine the final form of the set of externally exposed surfaces that would then form the ‘wetted’ surfaces of the design when evaluating the aerodynamic loads and wake predictions. The use of the AGRS as an auto-correction system for input grids serves multiple purposes. Firstly, it allows the use of simple geometric shapes that are easy to input into the system, for creating more complex shapes and structures without the user having to provide more complicated input data. Secondly, the AGRS provides a buffer between the extremely grid-sensitive aerodynamic codes used to evaluate the loads and the necessarily austere and versatile input grid data. This buffer ensures that the grid received by the load predictor codes is well designed and stable, while reducing the need for the user to do the same, thereby easing the user workload. The following sections discuss both the AMA and the AGRS in detail.

Conceptually, every airframe design is broken down to two basic types of structures: Fuselage shapes and lifting surfaces like wings, fins etc. The AMA has been designed to approach the initial grid design process in the same concept. Input files contain the locations and geometry of fuselage shapes and lifting surfaces separately. It should be noted here that the fuselage shapes can be built up from several simpler shapes as mentioned previously and the AGRS can be activated to compensate for the errors and provide a final grid structure. It should also be noted that in some cases the AGRS may not chosen for employment when for example the intersecting surfaces are in fact real and not an error induced by surface overlaps. As such, the AMA has an optional employment for the AGRS that is set by the user beforehand. In most cases, the AGRS setting is kept active considering the rarity of using extremely odd shaped intersecting surface products in an airframe model.

The simpler initial AMA fuselage structures are created using a modular section add-up scheme. In this system, a given three dimensional fuselage shape is divided into numerous cross-sections where the geometric parameters change along the length of the body. It should be noted that all fuselage shapes that are entered within the AMA are aligned so that their individual body axis must be parallel or coincide with the primary aircraft body axis, designated throughout the course of this paper as the ‘X’ axis. As a result, each cross section of each fuselage structure is in the ‘YZ’ plane. It should be noted that even though the AMA fuselage structures have local body axes that are parallel to the three axes Body-Inertial-Frame (BIF), their exact offset from the BIF axes remains a set of independent AMA inputs that can be changed to place each of these fuselage structures in three dimensional space independent of the other structures. Once the placement of all such structures is complete, the AGRS can be activated to investigate potential surface grid errors and also to trim and construct the final AGRS grid.
Each cross-sectional plane of a given fuselage structure is a plot of the exposed outer perimeter of two intersecting ellipses with their major axes at ninety degrees to each other. One of these ellipses has its major axis parallel to the vertical axis of the BIF system, referred from this point onwards as the ‘Z’ axis. This ellipse is also located so that its major axis is always located on the primary XZ plane of that fuselage shape which is also parallel to the BIF-XZ plane. This ellipse’s major axis remains fixed within this plane and the ellipse is only able to expand or contract along its major and minor diameters, both of which act as independent AMA inputs. For this reason this ellipse is designated as the Primary-Sectional-Ellipse (PSE).

The PSE for each section of a given fuselage shape therefore have coincident major axis vectors when viewed down the local Body-X axis. The only other independent parameter for the PSEs is the ability of the user to locate their centers at different elevations above or below the local Body-X axis. Therefore, the PSE of each Body-YZ plane is able to move up or down relative to the PSE before it but not move sideways. It should be noted that the major and minor axis for each PSE of every cross-sectional plane of a given fuselage section can have different values and are independent AMA sectional input parameters.

The Secondary-Sectional-Ellipse (SSE) is defined in a manner similar to the PSEs but with more independence in location. The SSE is the second of the two elliptical shapes that are used to define a given cross-sectional shape of a given fuselage structure. The SSE has its major axis directed along the local Body-Y axis and therefore the minor axis is located along the local Body-Z and hence coincident with the major-axis of the PSE. The SSE, however, does not have a location restriction placed on its center, and hence is able to move in the vertical direction above or below the local Body-X axis within a given cross-section. It is not able to move in the horizontal direction along the local Body-Y, however, and hence the exposed perimeter captured in each cross section of the fuselage structure is symmetric. Asymmetric fuselage shapes may be obtained by locating various fuselage structures around the fuselage in three dimensional spaces as mentioned previously but each such structure itself will remain symmetric within the AMA.

The AMA is capable of evaluating the combined external perimeter of the PSE and SSE shapes. This external perimeter can therefore be given any shape as required by the user and then broken up into the grid structure to capture that exact shape. Such two dimensional shapes from every cross-section are then connected to the consecutive shape of the next cross-section beyond it within the AMA until the entire fuselage structure shape in all three dimensions is captured within the grid. It should be noted that this grid is not the grid that enters the aerodynamics package for usage, and therefore the user is not placed under pressure to provide a refined set of cross-sectional planes. This initial AMA grid is then passed within the AGRS where the latter system determines the best refinement of the grid between each pair of consecutive cross-sections of every fuselage shape.

Figure-2.1 The PSE and SSE concept of Modular Aircraft Grid Generation

The Secondary-Sectional-Ellipse (SSE) is defined in a manner similar to the PSEs but with more independence in location. The SSE is the second of the two elliptical shapes that are used to define a given cross-sectional shape of a given fuselage structure. The SSE has its major axis directed along the local Body-Y axis and therefore the minor axis is located along the local Body-Z and hence coincident with the major-axis of the PSE. The SSE, however, does not have a location restriction placed on its center, and hence is able to move in the vertical direction above or below the local Body-X axis within a given cross-section. It is not able to move in the horizontal direction along the local Body-Y, however, and hence the exposed perimeter captured in each cross section of the fuselage structure is symmetric. Asymmetric fuselage shapes may be obtained by locating various fuselage structures around the fuselage in three dimensional spaces as mentioned previously but each such structure itself will remain symmetric within the AMA.

The AMA is capable of evaluating the combined external perimeter of the PSE and SSE shapes. This external perimeter can therefore be given any shape as required by the user and then broken up into the grid structure to capture that exact shape. Such two dimensional shapes from every cross-section are then connected to the consecutive shape of the next cross-section beyond it within the AMA until the entire fuselage structure shape in all three dimensions is captured within the grid. It should be noted that this grid is not the grid that enters the aerodynamics package for usage, and therefore the user is not placed under pressure to provide a refined set of cross-sectional planes. This initial AMA grid is then passed within the AGRS where the latter system determines the best refinement of the grid between each pair of consecutive cross-sections of every fuselage shape.
Every single fuselage structure that is created within the AMA is therefore the combination of a set of fuselage cross-sections from the tip of the structure to its aft end. The only other independent parameter within this system is the location of the cross-sections themselves and therefore the distances between them. Overall, every cross-sectional plane for each fuselage structure has a total of six independent parameters. The number of planes chosen is also an additional independent parameter as is the choice and number of fuselage structures chosen to build up the airframe. It is noted that the number of parameters increase drastically once complex designs with external payloads etc are chosen. To ease up this problem, the AMA has an additional option of saving and storing the AGRS grid data. Once saved, the AMA also has an option of being able to read past created data files from the storage database so that complex geometries may not be required to be created repeatedly within the input file. The AMA would simply be pointed towards the file parameters and the design would be uploaded from the database and appended to the new AMA grid before being passed back into the AGRS for upgrades.

The lifting surfaces that are loaded within the AMA structure are also designed to be versatile. As mentioned previously, the possibility of use of predefined grid data files allows for the use of any non-traditional
airfoil cross-sectional shape required. This is particularly important with preliminary design problems involving optimization, whereby the necessary airfoil shape may be produced using non-traditional polynomials rather than the standard NACA airfoil designs. By default, however, the AMA is loaded with the NACA sectional geometry equations for numerous digit-airfoils. The surface is modeled so that only the mean camber surface of the geometry is created within the grid as opposed to creating both the upper and lower surfaces of the wing/fin. The need for this is based on the design of the aerodynamics package that is discussed later. However, it should be suffice to note here that sufficient correction factors are evaluated from the chosen geometry to account for the wetted surface area calculation that is important for the skin friction drag computations.

The number of independent parameters for each lifting surface grid includes the root chords, tip-chords, spans lengths, leading-edge angles, dihedral angles, pitch angles, the airfoil types, locations in three dimensional spaces of the wing root locations and the number of sections that are used to build up a given lifting surface. It should be noted that since a single lifting surface can be made up of numerous sections having different angles and geometries, most modern wing and fin designs can be created from the AMA created for this effort including designs such as winglets etc. The initial grid design obtained from the AMA for the lifting surfaces is similar in concept to the fuselage shapes grids in that both are created with the view of capturing the required geometry using simple concepts. The net combined grid obtained from the combination of this process is then passed collectively to the AGRS for enhancement and deployment towards the aerodynamics package.

Figure 2.4: The Scramjet Engine Array on the Sample Hypersonic Aircraft

Typical design features of this hypersonic airframe design system in the code include a ventral fore-body that allows for air pre-compression using a three-dimensional shock. The choice of using this type of pre-compression ventral fore-body design as opposed to a planar shock pre-compression design is because of the need for versatility with regard to the engine layout and capture area requirement that was mentioned previously. A semi-conical design allows for the placement of the scramjets along an arc as shown in Figure 2.4. This gives the code an option of varying the number of engines used for the same fore-body design and thus evolves the engine intake capture area as is needed for higher speed flights. Figure 2.5(a) and Figure-2.5(b) shed some light on this discussion.

Figure 2.5(a) and (b): The Scramjet Engine Array with a few engines over a smaller arc (left) and the Scramjet Engine Array with half the maximum possible engine array arc used up (right)
The code contains a robust model for the wing shape and design on the airframe. Each wing is capable of having up to ten separate sections and each section has the independence of design in terms of shape, angles and location. Further, if needed, the Genetic Algorithm can deploy further sets of fins or canards along the fuselage and can even deploy angled sets of vertical stabilizer surfaces with the same level of independence. Each wing surface is modeled as a diamond shaped airfoil with thickness and geometrical parameters available for optimization to the Genetic Algorithm. Figure-2.7(a) and 2.7(b) show this lifting surface design versatility in greater detail.

The dorsal fuselage of the airframe is designed to collaborate automatically with the ventral design in order to fit the required tankage and payload bays inside the fuselage. This section of the airframe is also free for variation within its parameter space so that proper lifting effects from the panels can be used to provide additional lift. There are two primary fuel tanks inside the fuselage that are shaped in the form of cylinders and arranged in a nonintegrated tankage\(^6\) (fuel tank is separate from the body) fashion. The structures that support the fuel tanks in place and the tanks themselves are assumed to have been insulated from the outside temperatures of the airframe by an insulation and active cooling system. The fuel tankage is therefore of the cool cylindrical type\(^6\). The payload and electronics bay of the fuselage is enclosed inside the inner surface of the ventral pre-compression cone. In addition, the shape of the bay is conformal\(^6\) to ensure that maximum usage of all available space is achieved.

Figure 2.6: A Typical SSTO All-Body Configuration

Figure 2.7(a) and 2.7(b): The lifting surfaces design versatility shown for a wing-body design (top) and an all-body hypersonic airframe design (bottom)
The AGRS design is a necessity driven by the grid-sensitive nature of the grid based numerical aerodynamics package employed for the drag prediction process. This grid system has shown significant sensitivity to the refinement process in order to yield good results. The presence of an improperly refined grid or untrimmed surface intersections can severely reduce the fidelity of the ISAA and hence the AGRS plays a crucial role between the AMA and the Aero-Predictive-Group of Codes-Hypersonic (APGOC-H). The aspect ratio of each panel is defined in the following manner:

\[ AR_{\text{panel},ij} = \frac{L_i}{L_j} \]  

(2.1)

It has generally been noted during the current effort that with the current version of the APGOC, the closer the aspect ratio of any given panel tends to move towards unity, the more accurate are the results. Applied for a surface, the AGRS is designed to reduce the mean panel aspect ratio for an exposed zonal surface by varying the number of grid panels in both the ‘i’ and ‘j’ numerical directions. The mean panel aspect ratio for a given zone number ‘k’ is then defined as:

\[ AR_{\text{mean},k} = \frac{1}{(ipts - 1) * (jpts - 1)} \sum_{i=1}^{ipts-1} \sum_{j=1}^{jpts-1} AR_{\text{panel},ij} \]  

(2.2)

The AGRS continues to deploy points along the lifting surfaces using varying successive ratios so that the minimum possible zonal aspect ratio is achieved for the minimum number of panels possible. For example, there would a lesser concentration of panels near the root than at the tip of a wing having a small tip chord but large root chord. This variation would be achieved using a successive ratio scheme for the panels moving from the root to the tip along the spanwise direction of the wing. For fuselage surfaces, the AGRS deploys points at equal intervals along the perimeter of each cross section depending on the complexity of the surface and then evaluates the number of intermediate cross-sections that would have to be deployed between the given cross-section and the next consecutive cross-sectional plane so that the mean zonal aspect ratio is close to unity. Each intermediate cross-sectional plane is therefore the result of interpolation between the PSE and SSE parameters between the two primary cross-sections. The minimum number of intermediate planes is zero, thereby indicating that the two primary cross-sectional planes are far too close to allow additional refinement between them, but the maximum number of intermediate planes is a user control setting that can be changed to allow varying computational times for varying grid refinements depending upon the applications.

The AGRS is also able to trim intersecting surfaces once the AMA grid has been loaded into the system by creating an external reference ‘cloth box’ in three dimensional spaces that encompasses the given airframe inside it. Thereafter, the exposed surface grid is created in concept by collapsing the cloth-surfaces of the cloth-box inwards until they ‘sit’ directly over the airframe surfaces. In this manner, the final parameters of the cloth box surfaces represent the net exposed surfaces of the airframe and these parameters are then forwarded into the grid refinement scheme.

In the AGRS, this is conducted by moving in a series of positions along the ‘Y’ direction of the cloth-box for a given ‘X’ axis value and a given X-Y plane of the cloth box, i.e. for a fixed ‘Z’ axis value. When the moving ‘Y’ axis location hits the first exposed surface of the loaded AMA grid, further movement is presented and the location of the ‘Y’ value is stored. In this way, since the ‘X’ and ‘Z’ values of the grid point were predetermined within the cloth-box, the intercepted value of the ‘Y’ value on the surface of the AMA grid represents the exposed point of the airframe. At this point the ‘X’ value is changed leaving the ‘Z’ value as before and the process is repeated for a host of ‘X’ and ‘Z’ values respectively until the entire external exposed surface of the airframe has been probed and mapped by the AGRS. In this manner, internal intersecting fuselage shapes and surfaces are removed from the system and this allows the AMA and hence the user to build a complex geometry using the intersection of several simpler geometric models.
III. The Aero-Predictive Group of Codes- Hypersonic (APGOC-H)

The APGOC is a collection of subroutine systems within the ISAA that have been created for this effort with the aim of predicting the aerodynamic loads and distributions over the AGRS grid surfaces for a given airframe. The aerodynamics of the hypersonic airframe is handled by passing the gridded airframe as discussed previously through the solver for the Euler Equations as established in the flow field around the aircraft. The boundary conditions are imposed on the equations as applicable for far-field as well as fuselage wall conditions and the resulting set of equations are solved in three dimensional spaces to arrive at the pressure, velocity and density fields around the aircraft. The Steger-Warming flux vector splitting with limited upwind state extrapolation in the style of Van Leer’s MUSCL technique has been created for the Euler code. This model is explicit in time and the main numerical discretization scheme used throughout the code is based on the first-order forward difference scheme as applied to the case of the state and Inviscid flux vectors and is given as:

\[
\frac{\Delta y \Delta x}{\Delta t} \left( U_{i,j}^{n+1} - U_{i,j}^n \right) + \left[ \left( F_{i+1/2,j} - F_{i-1/2,j} \right) \Delta y + \left( G_{i,j+1/2} - G_{i,j-1/2} \right) \Delta x \right] = 0
\]

(3.1)

Where the Inviscid flux vectors are obtained as:

\[
U = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho e \end{pmatrix}, \quad F = \begin{pmatrix} \rho u^2 + (y-1) \left( \rho e - \rho \frac{(u^2 + v^2)}{2} \right) \\ \rho v \left( e + (y-1) \left( \frac{(u^2 + v^2)}{2} \right) \right) \end{pmatrix} \]

\[
G = \begin{pmatrix} \rho v \\ \rho vu \end{pmatrix} = \begin{pmatrix} \rho v^2 + (y-1) \left( \rho e - \rho \frac{(u^2 + v^2)}{2} \right) \\ \rho \left( e + (y-1) \left( \frac{(u^2 + v^2)}{2} \right) \right) v \end{pmatrix}
\]

(3.2)

The concept of Flux Vector Splitting (FVS) is used to provide a decomposition of the flux vectors such that:

\[
F = F^+ + F^-
\]

(3.3)

The above splitting functions can be summarized in the form as given below:

\[
F^\pm = \frac{\rho}{2y} \begin{pmatrix} 2(y-1) \lambda_{1,i}^\pm + \lambda_{3,i}^\pm + \lambda_{4,i}^\pm \\ 2(y-1) u \lambda_{1,i}^\pm + (u + c k_1) \lambda_{3,i}^\pm + (u - c k_1) \lambda_{4,i}^\pm \\ 2(y-1) v \lambda_{1,i}^\pm + (v + c k_2) \lambda_{3,i}^\pm + (v - c k_2) \lambda_{4,i}^\pm \\ (y-1) (u^2 + v^2) \lambda_{1,i}^\pm + \left( h_t + c (k_1 u + k_2 v) \right) \lambda_{3,i}^\pm + \left( h_t - c (k_1 u + k_2 v) \right) \lambda_{4,i}^\pm \end{pmatrix}
\]

(3.4)

Where,

\[
k_{1,2} = \frac{k_1 k_2}{\sqrt{k_1^2 + k_2^2}}; \quad \lambda_{1,i}^\pm = \frac{\lambda_i \pm |\lambda_i|}{2}; \quad \lambda_{1,2} = k_1 u + k_2 v; \quad \lambda_{3,4} = \lambda_4 \pm c \sqrt{k_1^2 + k_2^2}
\]

The terms \( k_1 \) etc represent the “orientation” of the flux vector in the sense that the contributions from the horizontal and vertical components of the velocities would be taken together as shown in the equations below and would therefore contribute towards the net flux vector oriented in any direction in the X-Y plane. This concept is applied generally and is extremely useful even when considering the case of the specialized aligned rectilinear two-
dimensional problems. Regardless of their redundancy for such simple applications, they have been embedded into
the code from the outset with an eye towards more complicated analyses obtained from the optimizations runs.

The choice of first or second order schemes was necessary as a user input. As a result, three schemes were
set up that allowed the user to quickly interchange between the schemes and conduct analysis as it which one to use
for a given problem. To do this, the use of the Van-Leer Monotone Upwind Scheme for Conservation Laws
(MUSCL) was used. This method can be described as:

\[ U_{i+\frac{1}{2}}^L = \frac{1}{2}(U_{i,j} - U_{i-1,j})\varphi_{i,j} \]

\[ U_{i+1,j}^R = \frac{1}{2}(U_{i+2,j} - U_{i+1,j})\varphi_{i+1,j} \]

\[ U_{i-\frac{1}{2}}^L = \frac{1}{2}(U_{i-1,j} - U_{i,j})\varphi_{i-1,j} \]

\[ U_{i,j}^R = \frac{1}{2}(U_{i+1,j} - U_{i,j})\varphi_{i,j} \]

(3.5)

Where, the limiters \( \varphi_{i,j} \) etc are set up so that the system reduces to a first order system when these
limiters are equal to zero and then to a second order system when they are unity. The requirement of either of these
two choices is explained in far greater detail within the main body of the case–study analyses that follow in later
sections of this report. In addition to the above equations, the requirement for the MinMod scheme was proven
without doubt in the analysis of the shock-tube problem. The MinMod function as used in this report is the standard
derivation of the function and given as:

\[ \varphi_{i,j} = \text{minmod} \left( \frac{U_{i+1,j} - U_{i,j}}{U_{i,j} - U_{i-1,j}}, 1 \right) \]

\[ = \text{sign} \left( \frac{U_{i+1,j} - U_{i,j}}{U_{i,j} - U_{i-1,j}} \right) \max \left\{ 0, \min \left[ \frac{U_{i+1,j} - U_{i,j}}{U_{i,j} - U_{i-1,j}}, \frac{U_{i+1,j} - U_{i,j}}{U_{i,j} - U_{i-1,j}} \right] \right\} \]

\[ \varphi_{i,j} = \text{minmod} \left( \frac{U_{i+1,j} - U_{i,j}}{U_{i+2,j} - U_{i+1,j}}, 1 \right) \]

\[ = \text{sign} \left( \frac{U_{i+1,j} - U_{i,j}}{U_{i+2,j} - U_{i+1,j}} \right) \max \left\{ 0, \min \left[ \frac{U_{i+1,j} - U_{i,j}}{U_{i+2,j} - U_{i+1,j}}, \frac{U_{i+1,j} - U_{i,j}}{U_{i+2,j} - U_{i+1,j}} \right] \right\} \]

(3.6)

The boundary conditions used in the evaluation of the source strengths on the surfaces of the panels for this
effort are based on the Neumann model for surface boundaries. Models such as the Dirichlet model and the mixed
boundary condition problem were also evaluated for usage but were found to provide an insufficient increase in
accuracy of the results for the complexity brought into the solution schemes and hence were removed from the
boundary condition equations.
In addition to the above solutions, to simulate the flow physics accurately, the specific heats, viscosity, heat conductivity constants and the specific heat ratio are evaluated as functions of temperature alongside the viscosity and conductivity equations for atmospheric air. For this purpose the following equations are used:

\[ C_p = a_0 + a_1 T + a_2 T^2 + a_3 T^3 + a_4 T^4 + a_5 T^5 + a_6 T^6 + a_7 T^7 \]  
\[ (3.7) \]

Where, 
\[ a_0 = 0.25020051, \quad a_1 = -5.1536879 \times 10^{-5}, \quad a_2 = 6.551948 \times 10^{-8}, \quad a_3 = -6.7178376 \times 10^{-12}, \quad a_4 = -1.512825 \times 10^{-14}, \quad a_5 = 7.6215767 \times 10^{-18}, \]
\[ a_6 = -1.4526772 \times 10^{-21}, \quad a_7 = 1.011554 \times 10^{-25} \]

\[ \mu = \left( \frac{T^{1.5}}{T + 111} \right) (1.46 \times 10^{-6}) \]  
\[ (3.8) \]
\[ k = \left( \frac{T^{1.5}}{T + 112} \right) (1.99 \times 10^{-3}) \]

A one dimensional shock tube, and then a two-dimensional shock problem were investigated for the given effort to test the various sub-structures of the Euler code. A shock tube in the one dimensional sense is characterized by the movement of a shock structure downstream of the rupture of point of the diaphragm separating the two regions of dissimilar pressure and density systems into the region of the low pressure, low density fluid. The system is also characterized by the upstream movement of an expansion system for the high pressure fluid.

The given simulations were performed in a single dimensional system for the Euler code and this thus represented a highly restricted model of the generalized two-dimensional system utilized for later examples. The given initial conditions for the shock tube system in terms of normalized parameters are summarized as shown below:

\[ x < 0: \quad \rho = 1, \quad p = 1 \]
\[ x \geq 0: \quad \rho = 0.125, \quad p = 0.1 \]

The computational domain for this example consisted of \( x \in (-1,1) \) with 100 interior cells. \( \Delta x = 0.02 \). The solution was marched in time for 0.4 units. The maximum flux Jacobian eigenvalue for the given example was given to be 2.1916 and the CFL number of 0.3.

The given problem was investigated for the above mentioned conditions but also tested for varying CFL numbers and numerical solution techniques to visualize the nature of the solution. Of the models created for varying order accuracy of the solution, the data for the first and second order systems have been presented in this report. In addition, an investigation of the convergence of the solutions under varying numerical schemes and varying
operating conditions was conducted by defining the convergence criterion as discussed previously. The results for the shock-tube problem are shown in Figures-3.2-3.8.

Figure-3.2 shows the variation of the static pressure, density and single dimension velocity of the fluid at the final time of iteration within the computational space domain, which in this case corresponds to a single set of grid values in one dimension as shown. Figure-3.2 shows the nature of the system in terms of the variation in the pressure, density and velocity parameters. Note that at the given instant of time beyond the diaphragm rupture time, the shock structure has moved downstream into the low pressure fluid to the right of the rupture point. At the same time, the expansion system has moved to the left of the rupture point, shown by the reduced pressures and densities left of the rupture point. Notice that the solution as given in Figure-3.2 was evaluated at the desired CFL condition and with a First Order numerical discretization approach. As a result, the smeared nature of the normal shock moving into the low pressure fluid is visible with a highly diffused shock discontinuity slope. Such dissipative effects are of course associated with First-Order methods. It should also be noted from Figure-3.3 that the Second Order approach at the same CFL condition shows a much more proper discontinuity slope for the normal shock but at the cost of severe oscillations at most other locations.
Figure 3.4: U-Velocity comparison for the case of the 1D shock tube
(Left: CFL=0.3, t=0.4, First Order Accurate, Right: CFL=0.3, t=0.4, Second Order Accurate)

Figure 3.5: Density comparison for the case of the 1D shock tube
(Left: CFL=0.3, t=0.4, First Order Accurate, Right: CFL=0.3, t=0.4, Second Order Accurate)
Figure-3.6(a-b): Evolution of the Shock Angle as a function of pseudo time marching over the ramp for the case of the generalized oblique shock problem

Figure-3.7(c-d): Evolution of the Shock Angle as a function of pseudo time marching over the ramp for the case of the generalized oblique shock problem

Note that in this particular problem the solution is in fact a steady state solution. As a result during iterations, we march in pseudo time steps rather than real time steps as far as our ability track solution convergence history etc is concerned.
The above discussed Euler equations approach does not account for the skin friction drag. Data presented by Koppenwallner\textsuperscript{12} has shown that there is a large increase in the drag coefficient beyond that predicted by Newtonian approaches alone when the Knudsen number is greater than 0.01 for right circular cylinders. Based on the data presented by Koppenwallner\textsuperscript{12} the skin friction drag coefficient was evaluated using the following equation:

\[
C_{d,f} = \frac{5.3}{Re^2}\]

\(3.9\)

Figure-3.9 The Panel Interactions for the evaluation of the reference length

Where the Reynold’s Number used is that evaluated locally over the surface panel of the airframe using the distance from the exposed edge of the surface for the reference length and the local velocity and density of flow above the panel. When this friction coefficient is summed over the entire airframe, the net value of the skin friction
drag coefficient is obtained along with the net skin friction drag data for the vehicle. Figure-3.10(a) and -3.10(b) and Figure-3.11 present some of the results for the skin-friction coefficient distribution on the sample hypersonic airframe evaluated previously. The leading edge values for the skin friction are seen to noticeably higher than the rest of the fuselage and this has a profound effect on the aerodynamic heating characteristics of the vehicle (which will be characterized by the Stanton number distribution later in this paper) which in turn affect the structural design of the vehicle considerably. This model for the skin friction drag is seen to be very accurate in its results as will be discussed in the paper.

The surface pressures and velocities over the panels are evaluated using the Euler code data so that when combined with the panel areas the net pressure forces acting perpendicular to each exposed panel can be evaluated. This pressure force is then resolved into the vehicle primary BIF frame axis components and added up. Since most of the cases studied for this effort were symmetric designs about the BIF ‘X’ axis, the ‘Y’ direction pressure forces have almost always cancelled out. The ‘Z’ component forces are in fact contributions to lift from the body in cases where the design permits the body to behave as if it were at an angle of attack even when not directly so. The ‘X’ component of forces is crucial in that their direction specifies whether the body is assisting with the thrust or acting against it, i.e. pressure drag. This value of the pressure force acting opposite to the thrust is then referenced against some lifting surface geometric parameters such as wing area when wings are present or the mean body diameter in case wings are absent.

Figure 3.10(a) and 3.10(b): Skin Friction Coefficient distribution on the sample hypersonic vehicle (left). Notice the leading edge values of skin friction (right).
(Altitude: 120,000 ft, Mach 6)

Figure 3.11: Skin Friction Coefficient distribution on the sample hypersonic vehicle.
(Altitude: 120,000 ft, Mach 6)

At angles of attack, surface panels that are exposed to the flow need to be evaluated along with the exposure angles to fully apply the panel boundary conditions on the Euler equations. As mentioned previously, the airframe is fully gridded up for this purpose by the design code. For different angles of attack, different panels will be exposed on the airframe and those too at different exposure angles. This process has been completely automated in the code designed for this effort. Figures 3.12(a)-3.12(c) illustrate this process by showing the regions of the

American Institute of Aeronautics and Astronautics
The airframe surface on the sample hypersonic vehicle shown previously that are exposed to the incoming airflow at angles of attack varying from -10 degrees to +10 degrees.

Figure 3.12(a)-(c): Panel exposure distribution as a measure of the panel orientation concept on the sample hypersonic vehicle. From top to bottom: (a) +10 Degrees AOA, (b) 0 Degrees AOA, (c) -10 Degrees AOA (Altitude: 120,000 ft, Mach 6)

American Institute of Aeronautics and Astronautics
IV. Aerodynamic Heating Effects and Mass Properties

The use of the Euler Equations does not lend itself towards the evaluations of the Aerodynamic Heating effects by itself. An empirical model has been created and used in this effort since the heating affects the distribution of masses and structures within the airframe and the evaluation of the distribution of the masses is necessary to provide an accurate, airframe size-dependent value of the weight that is balanced by the lift from the combined lifting surfaces of a typical airframe during the optimization. The three dimensional grid developed for this analysis is used for the evaluation of the skin mass as it provides the essential interface between the external loads acting on the vehicle as evaluated from the aerodynamics module with the structural analysis module. Each panel is evaluated for its contribution to the overall skin mass of the vehicle by multiplying the panel area with the thickness of the panel and the material density.

The evaluation of the thickness of each panel is a result of both the loads acting on that panel as also the thermal considerations for that panel. Panels located near the leading edges of wings are more exposed to high temperatures than the panels near the top surface trailing edge of the wings, for example. The panels exposed to high temperatures will require either additional thickness of the material, additional ablative coating on top of the existing thickness or an active cooling system underneath it. It becomes clear that the aerodynamic heating effects on the external surface of the vehicle need to be modeled alongside the structural analysis in order to get a realistic distribution of the center of gravity of the vehicle as well as the value of the overall mass of the vehicle.

The model used for the evaluation of the aerodynamic heating on the hypersonic airframe for this effort is based on the numerical integration modification of the Eckert Reference Enthalpy method of heat transfer for compressible flows. It uses the concept of a reference temperature for a constant property boundary layer that can replace the variable property boundary layer effects. This concept is applied for each panel by calculating the reference temperature over each panel using the following equation:

$$\frac{T_{\text{REF}}}{T_{\text{panel}}} = 1 + 0.032 M_{\text{panel}}^2 + 0.58 \left( \frac{T_{\text{wall}}}{T_{\text{panel}}} - 1 \right)$$

(4.1)

Once the reference temperature has been evaluated for a given panel, the Prandtl number, Reynolds number and Stanton Number are evaluated using the reference values of the flow properties. Note that the reference values of the flow properties are themselves evaluated using the reference temperature available from Equation-8. The resulting equations are then given as:

$$R_{e_x}^{\text{REF}} = \frac{\rho^{\text{REF}} U_{x} x}{\mu^{\text{REF}}}$$

(4.2)

$$Pr = \frac{\mu^{\text{REF}} C_{p}^{\text{REF}}}{k^{\text{REF}}}$$

(4.3)

$$St^{\text{REF}} = \frac{0.332}{Pr^{\frac{3}{2}} R_{e_x}^{\text{REF}} \frac{1}{2}}$$

(4.4)

The Stanton number is then used to evaluate the heat transfer rate over every panel using the following definition of this dimensionless parameter:

$$q_{w} = St^{\text{REF}} \rho^{\text{REF}} U_{e} (h_{\text{adiabatic},w} - h_{w})$$

(4.5)

Equation-4.5 is used to evaluate the heat transfer to the panel as a result of the aerodynamic heating effects. The overall heat distribution parameter for the sample hypersonic airframe is given in Figure-4.1. Notice the enhanced heating near the leading edges of the airframe. The heat transfer rate distribution is then used to evaluate...
the required thickness of the material in the presence of thermal stresses and external loads. This thickness evaluated from the above analysis for the vehicle skin is then used to find the masses of the panels at each section which is then summed up over the entire airframe to provide the skin mass of the vehicle. The paper discusses this model in greater detail.

![Figure 4.1: Aerodynamic heating based heat transfer rate distribution over the sample hypersonic airframe (Altitude: 120,000 ft, Mach 6)](image)

In addition to the skin masses, the payload and fuel carriage is also modeled. The fuel carried is modeled as having been placed in two primary tanks whose dimensions are automatically evaluated by the design code based on the external chosen design. These tanks are then modeled for their contribution to the vehicle mass properties using the thin wall pressure vessel theory. This analysis provides the tank thickness values which is then multiplied by the material density and the exposed surface of the tanks to obtain the mass of the tanks. The internal volume of the tanks is then used to evaluate the fuel masses and the corresponding center of gravity of both the tanks and the fuel. The payload is taken as a fixed mass that is added to the overall mass of the vehicle and whose volume is also added to the existing volumetric requirements inside the airframe. These are discussed in detail in the results and discussion section of this paper.

In addition to the calculation of the heat transfer rate distributions, the required coolant flow rate is also evaluated as it decides the maximum allowable temperature on the surface of the vehicle. In this effort, liquid hydrogen is used as the primary fuel inside the airframe. The propulsion module evaluates the required fuel flow rate for the scramjet combustor array. Since this flow rate also acts as the coolant for the exposed skin of the airframe, a numerical integration of the exposed surface areas and the aerodynamic heating acting above those areas is conducted using the design grid panels. A net heat transfer rate is thus achieved when Equation (4.5) is provided with the maximum allowable temperature on the surface of the airframe defined from a materials standpoint.

This heat transfer rate is then compared with the ability of the coolant flow to absorb this heat transfer. For this effort it was assumed that the dissociation of the hydrogen fuel moving towards the combustor section is undesirable and therefore the maximum temperature of the coolant was 1500K. Assuming cold storage values of the fuel within the tanks (whose additional masses were added to the mass properties tabulation), the maximum allowable adiabatic temperature of the external flow was evaluated using an enthalpy recovery factor of unity at the wall surface. Since the value of the heat transfer rate per unit time is a function of the design of the airframe, and the fuel flow rate is also variable based on the design, the balance of heat transfer versus heat removal was set up as an additional goal within the optimization run. Any design that could not meet the minimum required conditions of heat removal could therefore not be classified as a design that would be able to operate at those speeds unless additional coolant material was used instead. For this analysis, the primary fuel acts as the only coolant within the vehicle in order to maximize on the payload carrying capacity.

V. Propulsion

The propulsion model for this effort uses the Euler code grid to evaluate the fluid flow entering the Scramjet Combustor sections with an additional grid fixed inside the combustor for modeling the combustion
physics and flow using a modification of the equilibrium based numerical codes for high speed combustions. The effects of propulsion can be considered to begin from the moment the flow goes through the pre-compression shocks and till the time they are behind the airframe expansion surfaces. The flow parameters directly behind the pre-compression shock emanating from the fore-body semi-cone are evaluated using the three-dimensional shock solution for the fuselage based intake and the aft-body exhaust expansion section as discussed previously. Figure-5.1 shows the Mach number distribution over the external compression cone of the sample hypersonic airframe.

Figure 5.1: Mach number distribution over a Sample Hypersonic airframe

Once the flow has cleared the pre-compression shock, it heads for the engine inlet where it undergoes a series of shocks that reduce the Mach number down such that supersonic combustion can take place in the combustor section further downstream. The length of the combustor is not fixed but is decided based on the number of shocks needed to slow the flow down to the required Mach number. In addition, the length required to actually burn the fuel is added to this length to give the effective combustion chamber length. The burn length model is adapted for use from the work presented by Mattingly\textsuperscript{14} for use in each of the scramjet array sectors on the aircraft.

The flow exiting the combustor is then allowed to flow over the aft section of the airframe that is designed solely with the idea of expanding this flow without needing a ducted nozzle. The net thrust for the vehicle is the summation of the thrust resulting from the pressure forces acting on the aft side of the fuselage and the thrust developed simply by the scramjet array. It has been noticed that the former is the major component in the thrust when the airframe is being designed for very high mach numbers. This section of the fuselage also contributes to the lift for the vehicle as was discussed in previous sections.

VI. Optimization Goals and the Design Space

The Genetic Algorithm used in this effort is a FORTRAN based, binary encoded GA\textsuperscript{16}. The attachment of the Genetic Algorithm as discussed previously with the comprehensive design, aerodynamics and propulsion models completes the construction of the automated network that can now be used for design optimization.

The primary goal of the optimization runs include matching the net lift of the vehicle as obtained from the aerodynamics module with the weight of the airframe as obtained from the mass properties subroutine. Another primary goal is to match the drag obtained from the vehicle with the thrust produced by the scramjet array. Secondary goals include minimizing the pitching moment, rolling moments and yawing moments of the aircraft along with reducing the root bending stresses of the wing attachments. The coolant heat transfer is also matched with the external heating to provide a multidisciplinary attachment between the propulsion, aerodynamics and the heat transfer aspects of the design.

The vehicle parameters that are available for change include all of the dimensions of the aircraft, the angles and weights. In addition, Bernstein Polynomials have been applied along the stream-wise and chord-wise directions of the wings and along the longitudinal axis of the upper fuselage. The lower fuselage has additional polynomials to model the external compressive inlet and the aft expansion surfaces. Each of the five coefficients of these
polynomials is available for independent variation by the GA. Each of the above design variables, if independent, is assigned a variation space within which the Genetic Algorithm is able to vary its values at a particular specified resolution. Overall, the above setup led to the presence of twenty-five variables in the design optimization process.

VII. Summary and Additional Results

The combined use of the Euler code embedded within the APGOC-H and used as part of the ISAA has led to the development of a very computationally efficient paneling method that is suitable for use in the preliminary design phase of hypersonic flight aircraft design. The validation of the system with experimental and flight test data has shown that the system is accurate within bounds. The use of this model with missiles is also being considered. The APGOC has been structured so that it not only provides the drag estimates as presented within this effort, but it is also capable of providing detailed aerodynamic force and moment data for the given configurations. As a result, the current code could find use in performance estimation and flight simulation systems as well.

The software foundation for this effort’s ISAA was designed with the ability to allow it to merge with standard optimization codes with relative ease and very little change to the existing algorithm structure. The authors have already tested the current ISAA with Genetic Algorithms for optimizing both individual components within an otherwise fixed design as also to develop completely new designs that meet the desired flight performance parameters. It is highly feasible that other such optimization algorithms could also be used to conduct detailed
single-discipline, multi-goal optimizations with the current ISAA or multi-disciplinary, multi-goal optimizations by the addition of custom, user defined objective functions that work alongside the APGOC while analyzing the AGRS grid.

The conference presentation will discuss the results of the validation effort for the aerodynamic prediction codes versus the NASA X-43 performance data by loading the X-43 within the three dimensional generic mold grid generator codes and evaluating the predicted performance data using the Euler Equation solver code developed for this effort. In addition, the skin-friction and heat models also created for this effort will be validated against the X-43 data obtained from the previously mentioned references.

The presentation will also include the results of the optimization run for the Hypersonic Designs as evaluated for single and multiple range Mach number flight regimes using the overall optimization infrastructure and will discuss additional improvements that are being made to the combustion physics models as well as the Euler solver codes in an effort to improve the preliminary design effort for hypersonic flight.

References


American Institute of Aeronautics and Astronautics