A Review of Hypersonic Vehicle Engine Optimization

Nicholas J. Pisani[1] and Peter A. Waszkowski[2]

Florida Institute of Technology, Melbourne, Florida, 32901, United States of America

Supersonic combustion within hypersonic aircraft is an area of research with great potential in the modern world. With improvements in hypersonics, aircraft will be able to travel at relatively high speeds compared to contemporary aircraft and have the potential to benefit both military and civilian organizations greatly. Hypersonic vehicles must utilize supersonic combustion in engines with maximum efficiency at speeds above Mach 3 to provide sufficient thrust. Based on previous research, some possible difficulties facing hypersonic vehicle engines are inlet efficiency, nozzle performance, and supersonic combustion stabilization. Inlet efficiency must be optimized to decrease the effects of shock losses and spillage drag. Since nozzle performance influences interactions between exhaust flow and airflow around the vehicle, poor performance can induce drag losses that occur at the trailing edge of the engine. Supersonic combustion encounters issues with flame stabilization and chemical reaction timing, both essential components of fuel efficiency and thrust control within the engine. This paper outlines progress towards solutions to these difficulties by following the flow through the whole engine, intending to inform future studies on whole engine optimization.

I. Introduction

Ready access to hypersonic vehicles can provide military organizations with decreased response time to threats and emergencies while improving stealth capabilities and survivability [1]. Commercial use of hypersonic vehicles can serve to provide a means of high-speed travel and thrill-seeking opportunities. Furthermore, hypersonics has the potential to shape future space transportation by providing reusable launch vehicles for missions. Improvements in the aspects of hypersonic vehicle engines are essential to achieving these possibilities. Inlet efficiency must be optimized to streamline shock interactions through variable geometry and support integration in combined cycle engines. Nozzle performance losses are improved through Rao's method and proper design analysis, increasing efficiency and limiting trailing edge flow interactions [2]. Stabilization of flame distribution and chemical reactions can be improved using flameholding, mixing, and ignition techniques [3]. The ability of hypersonic vehicles to operate within subsonic conditions can and must be improved by using combined cycle engines that provide optimal efficiency throughout the flight envelope [4]. Combining techniques that improve these aspects equips hypersonic vehicles with the necessary tools to ensure whole-engine optimization for a given application.

II. Inlet Efficiency

Inlets serve as the first component of the complete engine system and, therefore, control and affect the efficiency of components down the flow path. Inlet efficiency is one of the key factors restricting the performance of the entire engine, affecting not only the efficiency and stability of the propulsion system but also the aerodynamic shape and characteristics of the aircraft [1]. Considering there are no moving parts such as compressors or turbines in scramjet engines, flight vehicles that use these engines to fly at hypersonic speeds must integrate an intricate inlet geometry to slow down, compress, and pressurize the flow solely using boundary layer shock interactions. Oblique

---

[1] Student Member, Undergraduate Student, Department of Aerospace, Physics, and Space Sciences, 1810905
[2] Student Member, Undergraduate Student, Department of Aerospace, Physics, and Space Sciences, 1787260

shocks must occur at the leading edge of the flight vehicle leading into the scramjet engine to maintain a flow velocity above Mach 1 for adequate thrust to be produced down the flow path. Furthermore, since these engines are only efficient at high Mach numbers, usually above Mach 5, additional engines with lower Mach number efficiency must be integrated for the flight vehicle to operate independently in any capacity. Inlets utilized in hypersonic flight vehicles must then account for efficiency within separate engines at various Mach numbers unless designers choose to integrate separate inlets that each serve a singular purpose with each engine.

Veeran et al. conducted a study to assess the performance of a ramjet compression system in the application of a hypersonic vehicle. The vehicle designed and tested in this study was to accomplish an independent complete flight profile, similar to a commercial airliner, using a combination of engines while sustaining flight speeds of up to Mach 8. While a separate turbojet engine was integrated for Phases 1 and 2 of the flight envelope, Mach numbers 0-2.5, Phase 3 involved dual operation of the turbojet and a ramjet engine for Mach numbers 2.5-4, followed by a dually-integrated ramjet and scramjet system being used for the final Phases 4 and 5 of the flight envelope, Mach numbers 4-8. The inlet design preferred in this study for the dually integrated ramjet and scramjet engines was a straight cowl lip inlet. It is a common misconception that alternative designs, such as slightly curved cowl lips, can help manipulate flow direction and increase efficiency. The truth is, as the Mach number increases into hypersonic flight regimes, the curved cowl lip will serve to reduce the efficiency of the inlet. Three-dimensional CFD analysis of this inlet concept showed that while shocks originated at the forward walls of the scramjet and fell straight into the engine, the shocks were falling onto the ramjet inlet at an angle due to shock interference, affecting overall efficiency (Fig. 1) [5].
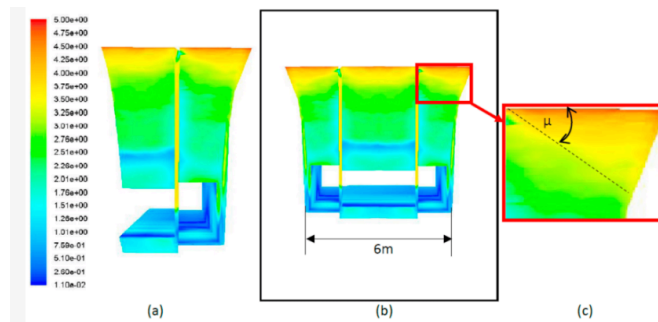
Fig. 1   Mach Number Contour for Original Inlet Configuration [5]

In addition to this shock interference effect, the physical height of the inlets was undesirable as it affected the aerodynamic profile of the overall vehicle and contributed to insufficient thrust values. To limit these effects, the ramjet inlets were rotated 90 degrees to be mounted sideways and attached to the scramjet engine's external obliques (Fig. 2).
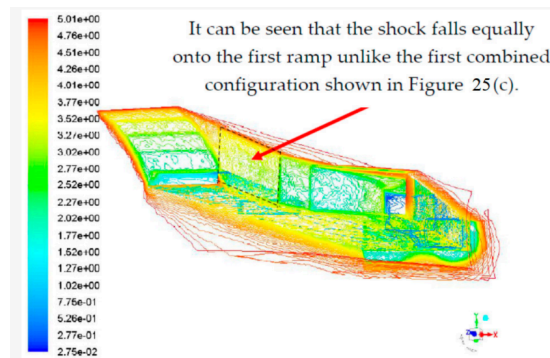
Fig. 2   Mach Number Contour for Alternate Inlet Configuration [5]

While this serves to nearly negate the shock angle by reducing the interference of the scramjet on shock formation leading into the ramjet inlet, the lack of compression assistance from the combined forebody at the leading edge of the engine could pose an issue. In this instance, all compression would have to occur via the ramp system along the outer edges of the scramjet inlet, leading into both ramjets. Additionally, a decrease in the wave-riding effect could be present due to the change in momentum of incoming flow particles, as they would be deflected outwards instead of downward [5]. Both designs are accompanied by pros and cons that affect engine efficiency; however, the shock interactions at the forefront of both inlets are the most influential factor and, therefore, can be controlled and limited by the latter design. The alternative combined cycle engine design also allows for better directional stability due to its smoother integration and more aerodynamic profile on the belly of the flight vehicle [5].

Pressure Recovery Factor is a dimensionless quantity used in fluid dynamics to characterize the efficiency of a system's ability to recover pressure levels in a fluid flow system, especially after boundary layer shock interactions occur. The conclusion section of this study used this quantity to describe the results of their experiment, stating that the overall performance of the ramjet compression system in the combined cycle engine was seen to become progressively poorer as the Mach number increased, lowering from roughly 80% pressure recovery at Mach 2 to 35% at Mach 5 [5]. While these results do not indicate an efficient inlet compression system, they do indicate that a variable geometry inlet used in combination with this system could serve to produce successful results, as the ramjet compression system did perform reliably and served to benefit the ramjet engines, but failed to do the same for the scramjet engine at hypersonic flight regimes. Although this could make vehicle design more difficult by increasing the intricacy of mechanical systems in the body of the aircraft, it would expand the capabilities of modern hypersonic aircraft. A variable geometry inlet would improve flight vehicle performance by broadening the range of operational Mach numbers.

Considering this difficulty of hypersonic inlets struggling to accommodate a large range of Mach numbers, Dalle et al. evaluated the effect of a variable-geometry cowl in inlet design. Their study considered three cowl motions used in a scramjet engine that could benefit inlet efficiency. These cowl motions, namely up and down motion, forward and backward motion, and rotation of the cowl lip, are evaluated using a low-order model designed for control-oriented applications that simulate wave interactions[6]. Since the most prevalent concern with hypersonic inlets is shock interaction with the possibility of an unstart, the formation of a shock-wave inside the inlet duct, this study focuses on evaluating the variable geometry in a way that proves it can decrease the possibility of these phenomena. To preface, actually building a variable geometry hypersonic inlet poses a multitude of technical difficulties that the paper does not actually address; rather, it aims to measure the potential benefit of allowing simple changes in inlet geometry[6].

The low-order fundamental model used to evaluate each variable-geometry hypersonic inlet design considered oblique shock waves, expansion fans, shock reflections, and wave interactions to determine which, if any, of these three cowl motions had a positive effect on inlet performance. An inlet design methodology was presented that allowed fixed-geometry, single-degree, or two-degree of freedom designs to all be obtained using the same method. This algorithm also included an operating map for values that variable-geometry inputs should take as a function of the flight condition. Results indicated that integrating variable geometry can strongly improve inlet performance over a wide range of flight conditions. Comparing two inlets designed for the same range of flight conditions, performance metrics showed that no negative results were found within the design range [6]. In the case of this study, vertical displacement (up and down motion) of the cowl was found to be far more advantageous than the other two cowl motions. While forward and backward motion was shown to assist inlet performance significantly, consideration was only made for the case that one variable geometry mechanism was available. Rotation of the inlet cowl was found to have no positive effect on performance, so it was not considered.

Overall, vertical displacement cowl motion was found to be the most advantageous considering its positive effect on pressure recovery factor, compression ratio, flow capture area, and temperature ratio, all over a wide range of Mach numbers, from 6-9.5. Considering success in these four categories as indicators of improved inlet performance, up-and-down cowl motion is presented as a beneficial alternative to fixed-geometry inlet design. Implementation of a variable geometry inlet design such as this, accompanied by the combined-cycle engine inlet alterations presented in the previous study, would eliminate the pressure recovery factor inefficiencies faced at higher Mach numbers, leading to an inlet design with improved performance. A combination design would serve to

operate well in a flight vehicle capable of hypersonic transport missions and independent operation at all Mach numbers 0- 8+.

III.   Supersonic Combustion Stabilization

Supersonic combustion stabilization requires that the combustion reaction between the fuel mixture and intake air occurs at speeds above Mach 1. This reaction must be sustained to maintain a predictable thrust output required for travel at supersonic speeds. The flame stabilization thus involves three time-dependent events that follow: flow mixing time, flow residence time, and combustion chemical reaction time [3]. Decreasing the flow mixing time ensures that the flow and the fuel are at adequate ratios, allowing consistent combustion. Elongating the flow residence time enables the combustion to stabilize successfully within the combustion chamber and not further down the flow path. Reducing the combustion chemical reaction time will improve the reaction's ability to stay within the combustion chamber, preventing combustion from occurring within other engine areas and resulting in less efficiency.

A well-discussed aspect that impacts the combustion chemical reaction time is flameholding within the combustion chamber to provide the fuel/air mixture with the necessary activation energy to ignite. A popular method of flameholding in supersonic conditions is to have a cavity within the chamber with a fuel jet providing an area of rich, subsonic recirculation flow. This area of relatively slow flow allows for a flame to be sustained and can act as an ignition source while providing a hot radical supply. These combined effects provide for sustained combustion within the chamber [7].

A study conducted by Wang et al. [8] investigated the combustion characteristics resulting from a cavity flameholder under supersonic conditions. The study conducted experimental and numerical analyses of fuel jet pressures of 0.6, 1.2, and 1.8 MPa with equivalence ratios of 0.038, 0.076, and 0.11, respectively. Under a flow at Mach 2.52 and mass flow of 1 kg/s within the combustor, it was found that with a high-pressure injector (1.8 MPa), the combustion of the fuel jet occurred the furthest upstream. Additionally, the location of the flame base oscillates considerably, which suggests heavy separation in the upstream boundary layer. However, under a low-pressure injector (0.6 MPa), the fuel jet was not fully ignited until downstream of the cavity. The fuel jet was placed upstream of the cavity. They observed that the combustion is stabilized around the cavity shear layer where the interaction between slow-moving and fast-moving flow induces the required recirculation of hot radicals necessary for a flame base to be sustained [8]. Thus, it can be concluded that the cavity flameholder has sufficient potential to sustain combustion in supersonic flows but is heavily dependent on the fuel jet pressure. Moreover, it highlights the fact that the location and shape of the shear layer's interaction with the main flow will influence the shape of the flame produced. This then impacts the flame distribution and propagation into the main flow, where the bulk of the thrust due to combustion reactions takes place.

Additionally, the performance of the flameholder can depend heavily upon the jet pressure. Within the lowest jet pressure combustion, the spreading angle of the flame can be interpreted as less than that of the others, with the highest jet pressure giving the largest spreading angle [8]. The large spreading angle can positively affect the combustion process, improving mixing, flame propagation, and chemical reaction timing. However, with the increase in the pressure of the fuel jet, the reaction becomes more aggressive, encroaching into the cavity space and disrupting the boundary layer of the ceiling of the camber due to the shock waves created. This turbulent stationary bubble caused by bow shock interactions from combustion and the upper wall boundary layer could have a choking effect on the flow through the chamber, decreasing the efficiency of the reaction.

A second proposed method of flameholding is through an electrical discharge parallel to the flow, allowing for plasma to assist in flameholding. In an experimental study conducted by Savelkin et al. [9] with a combustion chamber flow of Mach 2 and an initial stagnation pressure of 1.3-2.0 Bar, the influence of electrical discharge-induced plasma on flameholding was investigated. The study used a fuel jet injector integrated with the electrical discharge device, making a singular plasma injection module (PIM), where the fuel is injected in the same place as the plasma was induced. Their findings show that when discharge is turned on with the flame front stabilized, the combustion of the fuel/air mixture progresses axially upstream, increasing the oblique shock angle and the combustion zone average angle with it. Thus causing the separation zone to progress upstream and inducing thermal choking. Additionally, there is large-scale combustion instability resulting from oscillations of combustor

pressure, flow structure, and electrical discharge power. According to the study, the process starts with the discharge turning on, creating ignition and an increase in chemical power. As a result, the flow's pressure increases, creating flow separation and, in turn, decreasing the discharge's length, voltage, and power. These conditions cause the combustion to blow out and decrease chemical power, resulting in the flow pressure decreasing and flow reattachment. These conditions increase the discharge length, voltage, and power, allowing reignition. This vicious cycle results in violent vibrations within the test equipment and the potential to cause damage in integrated aircraft [9].

However, Savelkin et al. demonstrated that, compared to other methods of plasma flameholding where the discharge occurs upstream from the fuel jet, a PIM creates higher normalized pressures at high fuel mass flow rates. This effect is potentially due to the radical generation resulting from free electrons striking fuel molecules and enhanced mixing between the fuel and air due to plasma convection [9]. High fuel mass flow rates result in a rich combustion mixture, pointing to the use of PIM in engines with lean combustion mixtures having less efficiency. Lean combustion mixture efficiency is currently sought after in commercial aircraft due to the environmental benefits of a fully combusted mixture. However, the high-pressure ratio found at high fuel mass flow rates demonstrates the practicality of PIM within engines where overall thrust output is the main concern.

Another process of initiating the combustion of fuel within a hypersonic flow is through the use of oblique shock waves that increase the stagnation temperature to the necessary level for the autoignition of the fuel used. Research was done by Veraar et al. [10] to investigate the use of oblique shock-induced combustion. Within their study, a test object with a half-cone angle of 5 degrees at the front and a half-cone angle of 39 degrees towards the back was used to facilitate the combustion of hydrogen injected at 1.28 g/s in a flow with a Mach 3.25 free jet nozzle. Their results confirmed the proof of concept in using oblique shocks to initiate combustion [10].

Within the experiment, two modes of combustion were observed: combustion within the boundary layer separation zone and combustion within the inviscid flow field. The two modes were due to a resulting 2-degree angle of attack to the flow due to thermal expansion within the equipment. Combustion within the inviscid flow field resulted in the peak heat load occurring downstream of the separation zone created. This observation signifies the ability of a scramjet engine to successfully initiate the combustion of reactants without the need to design the initiation area to withstand extreme combustion temperatures. At a sufficiently high airflow stagnation temperature, combustion products are not recirculated in an area of the engine, allowing for lower local heat loads. However, combustion within the separation zone amplifies the boundary layer separation zone and strongly increases the local heat loads, leading to thermal failure areas [10].

The implementation of oblique shockwave ignition occurs only through the specific geometry of the aircraft engine, reducing the number of parts and systems needed for combustion. The combustion process does not require a specific geometry designed for flameholding but instead provides sufficiently strong oblique shocks that can both compress and auto-ignite the fuel/air mixture. However, this process requires a flow with a sufficiently high stagnation temperature, which eliminates the use of this combustion process at subsonic and low supersonic speeds. Additionally, the process requires that the flow is uniformly distributed to allow for the oblique shocks to be stationary and uniform in nature. In the actual implementation of this phenomenon, the flow will not be uniformly distributed, creating a risk of an engine unstart, where the shocks are too strong and flow out the front of the engine, halting combustion. Thus, it is necessary to research techniques on how to mitigate these issues.

IV. Nozzle Performance

While inlets indirectly affect thrust production by properly stabilizing, compressing, and pressurizing the incoming flow, nozzles, situated at the trailing edge of the engine, directly influence thrust production through internal and external geometry. Since nozzles depressurize and accelerate the flow in the final stage of flow manipulation for a scramjet engine, they do not have a succeeding stage that could serve to fix any issues that arise. Therefore, their performance can be viewed as the most important in complete engine optimization. Furthermore, nozzles also influence interactions between bypass airflow and flow through the engine at the trailing edge of the vehicle, leading to more possible drag/backpressure if not designed efficiently. Considering this, scramjet nozzles must be designed to be as efficient as possible, with the goal of generating the greatest amount of thrust while also minimizing overall surface area[11].

A study conducted by Moizes et al. tested the effectiveness of three hypersonic nozzle designs that currently exist. The three tested were the de Laval nozzle, the bell nozzle, and the dual-bell nozzle. All three designs vary based on overall surface area and nozzle geometry. Unlike the dual-bell nozzle design, which was left in its original state when tested, the de Laval nozzle and the bell nozzle were altered to 60% of their original length to decrease surface area while maintaining sufficient thrust. MATLAB, SOLIDWORKS, and Simcenter STAR-CCM+ CFD programs were used to contour, model, and run computational fluid dynamics simulations on each nozzle to determine how effective they could be in a hypersonic flight environment.

Regarding nozzles, most designs portray similar qualities as they aim to complete the same goal. Therefore, minor alterations in nozzle geometry are crucial to scramjet engine performance as they can heavily impact thrust and specific impulse values in hypersonic flight regimes. The three nozzles examined in this research paper have distinguishing characteristics that make them original designs. While the dual-bell nozzle is characterized as a type of de Laval nozzle, it has very different alterations that allow it to compensate well for varying flight altitudes. Its straightening of the nozzle contour, followed by a large step outward, causes the jet to separate at the step when flying at low altitudes, avoiding instability and massive overexpansion of the flow. At higher altitudes, the ambient pressure decreases, leading to a higher expansion ratio that fills the nozzle geometry and provides more pressure against the rest of the nozzle, subsequently increasing thrust by allowing exhaust gas to reach higher velocities. It is crucial within these nozzles that the exit pressure does not dip too low below the ambient pressure, as this could result in the exhaust not reaching hypersonic levels. De Laval nozzles showcase more of a rapid convergent-divergent design where the nozzle tube is almost pinched in the middle, followed by a gradual divergence leading to its trailing edge. Similar to other nozzles, this design is mainly based on the fluid dynamics principle of the Venturi Effect, aka Bernoullis Principle. The Bell Nozzle, one most commonly used on rocket engines for space-flight vehicles, is nearly a pure divergence nozzle, with a very minimal amount of its design volume converging. Exhaust flow is subjected to a high-angle expansion section directly behind the nozzle throat, with a gradual decrease in the expansion angle as the flow reaches the nozzle exit. The curve back inwards following the high expansion section gives the exhaust a near-straight flow out of the nozzle, attempting to direct all thrust perpendicular to the trailing edge nozzle face.

The results of the study are presented in the following table, highlighting values characteristic of hypersonic flight vehicles such as thrust, thrust/area, and specific impulse, all in SI units:

Table 1 Results of Nozzle Testing [11]

Properties	Rao	Dual-Bell	60% Laval	Units
Surface Area	1.31	1.25	2.42	m^2
Fuel to Air Ratio	0.01044	0.01044	0.01044	
Mass Flow Rate	4.55	4.55	4.55	kg/s
Thrust	2356.35	2287.86	2223.52	N
Thrust/Area	1798.74	1830.29	918.81	N/m^2
Specific Impulse	5110.34	4961.82	4822.21	N-s

Since the initial conditions of the three nozzles are identical, their fuel-to-air ratio and mass flow rate remain the same, allowing quantitative results to be easier to compare. The fuel chosen for analysis was Hydrogen, which represents the type of fuel a real scramjet nozzle would experience [11]. Even with a 40% decrease in length, the de Laval nozzle stood out from the others with a much larger surface area, diluting its thrust/area ratio against the other nozzles. Rao, in this table, represents the 60% bell nozzle and has similar results to the dual-bell nozzle. The results of Table 1 indicate that the shortening of the de Laval nozzle had an impact on its efficiency while also showing that both the 60% bell nozzle and dual-bell nozzle are competitive at hypersonic flight regimes due to their

similar geometry and overall design. Flow separation is a possible drawback to both of these nozzles at higher Mach numbers, a flaw exaggerated in the Dual-Bell nozzle due to its inconsistent geometry.

In the application of a bell nozzle such as the Rao in this study, the nozzle's unique divergence pattern and overall geometry serve to combat possible drag losses on the trailing edge of the vehicle, a common drawback of hypersonic nozzles. The conical design of the divergence section produces this unique divergence pattern, allowing exhaust to be more streamlined, exiting the nozzle perpendicular to the nozzle face and limiting spillage of flow or vorticity in the fluid. In doing so, interactions between exhaust and bypass flow decrease, allowing greater thrust production and limiting drag.

Developing these advanced hypersonic nozzles requires an extensive process that uses multiple forms of quantitative analysis and mathematical modeling of nozzle characteristics known to produce necessary effects. T Cain offers insight into the scramjet nozzle design process, going step by step in analyzing both theoretical and experimental design techniques [2]. They propose a lecture split into four parts, beginning with a reminder concerning control volumes and proper aerodynamic and propulsion force analysis. After expanding on choosing proper control volumes and defining propulsive force vectors, the paper focuses on Rao's method, a topic introduced but not explained thoroughly in the previous paper, for maximizing thrust while minimizing nozzle length. Concluding by introducing a new, simpler Method of Characteristics for non-equilibrium flows, the author covers each step in the nozzle design process, providing a comprehensive overview of the topic [2].

To fully understand and correctly display force effects on a hypersonic vehicle and its engine systems, the thermodynamic principle of control volume analysis must be done accurately. In propulsion/airframe integration (PAI), control volume analysis is a useful concept that is vital to properly account for force effects inside and outside of the system. Common applications of this concept include the analysis of CFD simulations on designated control surfaces in hypersonic flight regimes. Performance models of selected hypersonic vehicle control surfaces could provide conditions at the nozzle exit plane, contributing to quantitative data collection for total propulsion forces and moments. Only control volumes in which the pressure and momentum fluxes are known for every part of the three-dimensional surface that defines them are useful. A properly chosen control volume serves to not only simplify analysis but also guide the design process itself [2].

Furthermore, C.V. analysis will provide useful data to maximize cruise efficiency, and while that may seem too broad for the scope of this section, it is important to note that nozzle design and its effects can be viewed as a part of the overall design of the aircraft considering an increase in overall vehicle efficiency also means an increase in nozzle efficiency. For example, nozzle optimization for the Dual Fuel cruiser, a ventral engine configuration from which the experimental X-43A was based, involved a parametric study of the effect of length and expansion ratio on cruise efficiency, not nozzle efficiency directly. In this study, the engine and bulk of the airframe were fixed, and two parameters that defined the nozzle were allowed to vary. Results detailed how cruise efficiency, rather than axial thrust or other factors, significantly impacted the optimum nozzle configuration [2].

Rao's method simplifies the Guderley and Hantsch method for axisymmetric nozzles, which focuses on determining the best forms for supersonic and hypersonic nozzles by considering gas motions without rotation about the axis. The new method, developed by Gadicharla V. R. Rao, is a technique that involves introducing an additional degree of freedom. It does so by not specifying that the control volume surface, on which thrust and mass flow would be determined, is defined by the Mach line that intersects the nozzle lip. This allows the surface that intersects the lip to vary in angle relative to the axis as an unspecified function of radius, determining the surface as a result of optimization. This method designs hypersonic nozzles that can achieve maximum thrust with limited surface area and length. Analysis of this technique involves using the Method of Characteristics (MOC) to calculate nozzle contour and contribute to more efficient nozzle designs. Consequently, MOC enables nozzle contour to be developed for reacting, non-equilibrium expansions, allowing for greater numerical analysis and design capabilities.

Nozzle performance is a key characteristic of complete engine optimization, as nozzles serve as the gateway for exhaust and bypass flow interaction. Without an optimized and efficient nozzle design, a hypersonic vehicle could suffer drag losses and boundary layer flow interactions at the trailing edge of the vehicle that could be catastrophic not only for performance but also for the structural integrity of the aircraft and its materials. Three possible hypersonic nozzle designs were introduced and tested in this section, each varying based on geometric characteristics. The results of the CFD analysis on these nozzles were tabulated and displayed effectively, detailing

the performance of each in various quantitative categories. The most efficient of the three was altered using a method described in detail later in the section within a comprehensive research paper aiming to break down each step in the nozzle design process. Through this research, it is obvious that an academic background in thermodynamics and fluid mechanics, as well as proficiency in various computer modeling and simulation applications, is necessary to design and test hypersonic nozzle configurations accurately.

V.   Independent Sub-Sonic Operation

For hypersonic vehicles to be used in commercial and military aspects, the engines of these aircraft must be able to provide thrust through the entire flight, from takeoff to landing. This requires that the design of the engines incorporates aspects of turbine jets, ramjets, and potentially scramjets. The turbine jet portion of the engine must be able to operate from takeoff to supersonic regimes; from there, the transition from turbine compression to shock compression must be used to initiate ramjet operation. Following this, the use of specific inlet, isolator, and combustor geometry will allow for the transition from subsonic combustion to supersonic combustion between a ramjet and scramjet, respectively. The usage of rocket-based combined cycle (RBCC) engines has been proposed, which uses rocket propulsion to accelerate to a Mach number sufficient to transition to ramjet mode. However, RBCC engines have a specific impulse far less than that of turbine-based combined cycle (TBCC) engines at subsonic speeds due to the use of rockets instead of turbojets when turbojets are optimized for low Mach and subsonic operation.

A TBCC engine allows the use of a turbojet using compression fans to compress incoming air for combustion to take place efficiently at low speeds. At supersonic speeds, around Mach 2.5-4, the turbine-based thrust is expected to transition over to ramjet thrust, using the speed of the incoming air to create a normal shock, compressing and heating the intake air for combustion. Proceeding with this, at higher Mach regimes of 4 and beyond, the ramjet thrust mechanism would transition to scramjet mode, using an oblique shock train to increase the pressure and temperature of the air to combust with the fuel at supersonic speeds [4].

There are two general categories into which TBCC engines can be separated. The first is the parallel TBCC engine, where the turbojet section of the engine is separated from the ramjet and scramjet flow path. This allows for smoother flow characteristics within the engine, allowing for the complete shutoff of the turbojet from the airflow, giving a direct path for intake air into the ramjet and scramjet combustors. The second is called a tandem TBCC engine, which contains the turbojet integrated inline with the ramjet and scramjet sections and provides the flow bypass tunnels to easily move past the turbojet into the ramjet and scramjet areas. The transition between low-speed and high-speed modes (turbojet to dual-mode scramjet) is critical in the vehicle's operation.

Within a parallel TBCC engine, the low-speed to high-speed mode transition occurs at a sufficiently high Mach number, approximately Mach 3, for the high-speed section of the engine to create shocks strong enough to initiate combustion [4]. The low-speed flow inlet and exit are closed, allowing proper development of the shock train necessary for ramjet operation. It is important to note that an engine unstart within the low-speed section can cause an engine to unstart within the high-speed engine due to disruption within the oblique shock train. Ramjet operation brings the flow Mach number to regions where dual-mode operation is possible, about Mach 4, changing the final shock within the isolator and combustor from a normal shock to an oblique shock, allowing for the combustor to work in both subsonic and supersonic dual-mode. The isolator's purpose within the dual-mode engine is to prevent engine unstart from disruptions in the flow by implementing a constant area duct between the inlet and combustor, causing wave reflections and an oblique shock train. As the Mach number further increases towards Mach 6, the shock train moves opposite to the flow, out the front of the isolator, resulting in scramjet mode with hypersonic combustion [4].

Within a numerical simulation of a parallel or over/under TBCC engine transition from turbojet mode to ramjet mode, the exhaust thrust was found to gradually increase with time, the total thrust coefficient of the exhaust system was over 0.9, and the lift of the exhaust system was negative most of the time during the transition mode [12]. At higher Mach numbers, the thrust coefficient is expected to increase as ramjet operation is more efficient at high Mach numbers than turbojet operation. The thrust increase can be used to compensate for the increase in drag created by the ramjet mode inlet, which is inversely proportional to the drag caused by the turbojet inlet during transition. Additionally, a total thrust coefficient maintained above 0.9 demonstrates the engine's ability to

efficiently provide thrust during the transition mode, which is necessary for practical usage. However, the lift generated by the exhaust system was strongly negative during the transition mode, with a sharp decrease in values [12]. This predicted effect by the engine during the transition mode shows an extremely difficult scenario to overcome in terms of maintaining flight equilibrium and stability. The lift generated decreases from -93N to -1153N over a two-second interval. Strong aerodynamic moments are created by this force, which will need to be corrected by flaps and other inputs to maintain the stability of the aircraft during transition. These numerical simulations were conducted at an expected flight speed of Mach 2.5 and altitude of 18km, which are towards the lower end of the transition Mach regime, which is predicted to be from Mach 2.5-3.5.

A simulation of a small-scale tandem TBCC engine investigated the mode transition performance at similar conditions where the Mach number was 1.8 and an altitude of about 11 km due to the restrictions within the simulated turbine used. The determination of the transition Mach number was done by comparing the total pressure at the inlet of the high-pressure compressor to the static pressure at the exit of the high-pressure turbine at $Ma_0=1.55$. Through the transition from turbojet to ramjet operation mode, the net thrust varied only by 4%, while the mass flow rate through the tandem engine varied by less than 8% [13]. This agrees with the previous simulation, where there was little fluctuation within the total thrust coefficient, demonstrating both of the engine types' ability to be used in practical applications. However, a difficulty within the tandem TBCC engine was that during the transition when the turbojet was blown out, there was a sharp decrease in the Mach number of the inlet to the ramjet combustion area. This was found to be related to the decrease in the total temperature of the turbojet exit with a small percentage decrease in the total pressure, making the inlet conditions adjust sharply to maintain the constant mass flow parameter. If conditions similar to this occurred within the practical application, a sharp disturbance in the flow might negatively impact the combustion occurring within ramjet mode, specifically unstart.

Based on the simulation results, it can be said that the use of TBCC engines within hypersonic vehicles can be practically done within hypersonic vehicles where the ability to take off and land independently is necessary. Both types of TBCC engines, parallel or over/under and tandem, can provide a relatively constant thrust during the transition from turbojet to ramjet mode. The use of tandem TBCC engines will most likely be used in applications where space within the body of the aircraft is limited and at lower hypersonic speeds. At higher hypersonic speeds, the drag due to the turbojet section of the engine will be too great to justify the usage of tandem TBCC engines. Instead, using over/under TBCC engines will allow for no drag created by the turbojet section during ramjet or scramjet mode. However, this design decision takes up a larger portion of the aircraft's body vertically. That being said, earlier characteristics of a TBCC engine were employed within the SR-71's J58 engines. At max speed, Mach 3.2, the turbine component of the engine only provided 20-30% of the total thrust, while the rest came from afterburning of bypass air; very similar to tandem TBCC at high Machs, which uses ram effect instead of afterburning for combustion outside of the turbine. The nozzle of the J58 can be used within Mach 6 applications; however, the engine design itself did not allow for supersonic combustion and high Mach speeds due to the limited technology at the time of development in hypersonics [14]. Currently, Hermeus has developed and tested a tandem TBCC named Chimera. The engine has shown its capability to transition from turbojet to ramjet combustion within laboratory conditions in November 2022.

VI. Conclusion

This paper outlines techniques to optimize each section of hypersonic engines. Vertical displacement cowl motion was found to be most beneficial to inlet performance due to its positive effects on the pressure recovery factor, compression ratio, flow capture area, and temperature ratio. Implementing this variable geometry in combination with a ramjet compression system, such as the one depicted in Figure 2, will provide flight vehicles with the ability to perform effectively over a wide range of Mach numbers. The inefficiencies at hypersonic flight regimes presented by the compression system would be improved by introducing variable geometry. Combustion stabilization can be enhanced through oblique shock ignition, plasma discharge ignition, and cavity flameholding techniques. Each method can benefit different practical applications, but due to the intricacy of each method, the combustion process is limited to incorporating a select technique optimized for the utilization of the engine. Using control volume analysis, Rao's method, and the Method of Characteristics in the hypersonic nozzle design process is crucial to producing optimized and effective designs that sustain necessary thrust and specific impulse values over all flight regimes. Turbine-based combined cycle engines are proven capable of providing a vehicle with independent hypersonic and subsonic intentions with the means to do so. Their ability to combine efficient existing engine technologies into one unit demonstrates their future use within the field.

While the information provided on studies conducted in this field of research is beneficial to advancing the capabilities of hypersonic vehicles, further research must be conducted. Studies of full engine assemblies that integrate multiple aspects presented in this paper must be conducted for a better understanding of how each component influences and is influenced by surrounding sections. Research on using multiple cowl motions in a single inlet configuration would prove useful in furthering scramjet inlet performance. Advancing knowledge of interactions within the combustion at different flow parameters in turbine-based combined cycle engines will be useful in determining proper transitions between each cycle. The design process previously mentioned for hypersonic nozzles would benefit from further computational analysis done on nozzles developed using all three techniques. Moreover, physical experimentation of processes simulated through computational analysis will prove to both confirm present findings and unveil possible unknown phenomena within this underdeveloped field of research.

References

[1] Ma, Y., Guo, M., Tian, Y., & Le, J. (2024). Recent advances and prospects in hypersonic inlet design and intelligent optimization. Aerospace Science and Technology, 146, 108953. https://doi.org/10.1016/j.ast.2024.108953

[2] Cain, T. (2010, September). Scramjet nozzles - DTIC. NATO Research and Technology Organization. https://apps.dtic.mil/sti/pdfs/ADA592961.pdf

[3] Qili Liu, Damiano Baccarella, Tonghun Lee, Review of combustion stabilization for hypersonic airbreathing propulsion, Progress in Aerospace Sciences, Volume 119, 2020, 100636, ISSN 0376-0421, https://doi.org/10.1016/j.paerosci.2020.100636. (https://www.sciencedirect.com/science/article/pii/S0376042120300488)

[4] McDaniel, J., Goyne, C., Edwards, J., Chelliah, H., Cutler, A., & Givi, P. (2009). US National Center for Hypersonic Combined Cycle Propulsion: An overview. 16th AIAA/DLR/DGLR International Space Planes and Hypersonic Systems and Technologies Conference. https://doi.org/10.2514/6.2009-7280

[5] Veeran, S., Pesyridis, A., & Ganippa, L. (2018). Ramjet compression system for a hypersonic air transportation vehicle combined cycle engine. Energies, 11(10), 2558. https://doi.org/10.3390/en11102558

[6] Dalle, D. J., Torrez, S. M., & Driscoll, J. F. (2011). Performance analysis of variable-geometry scramjet inlets using a low-order model. 47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit. https://doi.org/10.2514/6.2011-5756

[7] A. Ben-Yakar, M.G. Mungal, R.K. Hanson, Time evolution and mixing characteristics of hydrogen and ethylene transverse jets in supersonic crossflows, Phys. Fluids. 18 (2006) 026101. https://doi.org/10.1063/1.2139684

[8] Wang, H., Wang, Z., Sun, M., & Qin, N. (2013). Combustion characteristics in a supersonic combustor with hydrogen injection upstream of Cavity Flameholder. Proceedings of the Combustion Institute, 34(2), 2073–2082. https://doi.org/10.1016/j.proci.2012.06.049

[9] Savelkin, K. V., Yarantsev, D. A., Adamovich, I. V., & Leonov, S. B. (2015). Ignition and Flameholding in a supersonic combustor by an electrical discharge combined with a fuel injector. Combustion and Flame, 162(3), 825–835. https://doi.org/10.1016/j.combustflame.2014.08.012

[10] Veraar, R., Mayer, A., Verreault, J., Stowe, R., Farinaccio, R., & Harris, P. (2009). Proof-of-principle experiment of a shock-induced combustion ramjet. 16th AIAA/DLR/DGLR International Space Planes and Hypersonic Systems and Technologies Conference. https://doi.org/10.2514/6.2009-7432

[11] Vianna Moizes, D. A., Kotler, A. R., Thornton, M. R., & Ahmed, K. A. (2023). Comparison and analysis of hypersonic scramjet nozzles. AIAA SCITECH 2023 Forum. https://doi.org/10.2514/6.2023-0716

[12] Li, C., Xu, J., Mo, J., & Zhang, K. (2009). Numerical simulation of the unsteady mode transition process of an over-under TBCC exhaust system. 45th AIAA/ASME/SAE/ASEE Joint Propulsion Conference &Amp; Exhibit. https://doi.org/10.2514/6.2009-5301

[13] Zhang, M., Wang, Z., Liu, Z., & Zhang, X. (2016). Analysis of mode transition performance for a tandem TBCC engine. 52nd AIAA/SAE/ASEE Joint Propulsion Conference. https://doi.org/10.2514/6.2016-4573

[14] Russel, J. (2009). Turbine-based combined cycle propulsion. Turbine-Based Combined Cycle Propulsion. https://www.colorado.edu/faculty/kantha/sites/default/files/attached-files/russell.pdf