



e-ISSN: 2319-8753 | p-ISSN: 2347-6710

IJRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY

A futuristic digital interface with a glowing Earth globe in the center. The background is dark blue with various data visualizations including bar charts, line graphs, and network diagrams. Text elements like 'BUSINESS', 'NETWORK SEARCH', and 'LADING 100%' are visible. A person's hand is seen at the bottom, interacting with the interface.

INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 12, Issue 9, September 2023

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 8.423

 9940 572 462

 6381 907 438

 ijrset@gmail.com

 www.ijrset.com

Study on Atmospheric Re-entry Vehicles at various Mach Regimes

Brenda Supriya A ^{1*}, Sonali Gupta ²

M. Tech Research Scholars, Department of Aeronautical Engineering, Nitte Meenakshi Institute of Technology, Bengaluru, Karnataka, India¹

Assistant Professor, Department of Aeronautical Engineering, Nitte Meenakshi Institute of Technology, Bengaluru, Karnataka, India²

ABSTRACT: These studies focus on the comprehensive analysis of re-entry vehicles encompasses a multifaceted approach, combining aerodynamics, Computational Fluid Dynamics (CFD) analysis, and design scrutiny. Aerodynamics plays a pivotal role in understanding the complex flow dynamics that govern the vehicle's trajectory during re-entry into Earth's atmosphere. CFD analysis, a powerful computational tool, allows for a detailed investigation of the aerodynamic and thermal aspects, aiding in the optimization of re-entry trajectories and the development of effective thermal protection systems. Concurrently, the design analysis of re-entry vehicles involves a meticulous examination of structural integrity, materials, and geometries to ensure the vehicle's safe and efficient passage through the harsh re-entry environment. This synergistic approach, merging aerodynamics, CFD analysis, and design assessment, is fundamental in advancing the field of re-entry vehicle technology, facilitating successful space missions and ensuring the secure return of spacecraft to Earth.

I. INTRODUCTION

In the inception phase of re-entry vehicles, the manipulation of design parameters assumes a pivotal role in comprehending their overarching performance implications. The configuration and geometric intricacies of an entry vehicle wield substantial influence over its aerothermodynamic attributes. Prolonged and iterative modelling of the vehicular configuration permits an exhaustive scrutiny of the ramifications of shape variations on the performance during re-entry. Within the nascent stages of design, the employment of multi-objective optimization serves as a conduit for traversing the labyrinthine domain of shape parameters and discerning zones of paramount efficiency. Previous works have delved into the conundrum of optimizing the contour of re-entry vehicles, with instances such as the HL-20 vehicle and the crew exploration vehicle having undergone this optimization process. D. Dirckx et al [1] focuses on the establishment of a methodology for optimizing the form of vehicles during the conceptual design juncture, underpinned by expeditious evaluation protocols and optimization algorithms.[1]

In the realm of aero thermo-dynamics and flight dynamics, Sharp Edge Flight Experiment (SHEFEX)-I and its successor, SHEFEX II, orchestrated by DLR in 2005 and 2012, were orchestrated with the primary objective of showcasing the viability of aerospace vessels furnished with faceted Thermal Protection Systems (TPS) while concurrently enhancing their aerodynamic attributes. These experimental undertakings, followed by comprehensive computational simulations, have paved the way for profound exploration in the realm of sharp-edge re-entry vehicles. This multifaceted inquiry has encompassed diverse domains, ranging from aerodynamics, flight control, and structure design, to flight dynamics and subsystem formulation. The distinctiveness of sharp-edged vehicles, stemming from their intricate facets, endows them with commendable aerodynamic efficiency and a conspicuous stealth profile. While originally designed to facilitate fundamental scientific inquiry and component-level analysis, the revelations stemming from the SHEFEX initiatives have far-reaching implications. These revelations extend their influence to arenas as diverse as micro gravity experiment conveyances, high velocity global passenger conveyance, spaceborne passenger transport, & applications within the military domain. Guofu Liu et al [2] concentrates intently on the conceptualization,

schematic modeling, and comprehensive scrutiny of a sharp-edged, covert re-entry conveyance boasting a noteworthy lift/drag ratio. Additionally, it conducts a rigorous evaluation of pivotal parameters such as the lift-to-drag ratio, the center of gravity, and the covert capabilities of the vessel. This meticulous analysis leverages the Modified Newtonian Theory for a nuanced exploration of aerodynamic functionality. It also employs a procedural approach rooted in physical optics also the method of equivalent currents for the meticulous computation of radar cross-section.[2]

Precise anticipation of aerodynamic forces acting upon capsules during their atmospheric entry poses a formidable challenge for CFD. This challenge is particularly pronounced in the subsonic realm, where inaccuracies in turbulence modeling could exert a substantial influence on the projection of aerodynamic behavior. Scott M. Murman et al [3] delve into a multitude of facets concerning numerical and physical modeling in this context. Their exploration encompasses matters such as the implementation of inviscid numerical techniques, the adaptability of mesh structures, the intricacies of modeling rough surfaces, as well as the application of corrections for rotational and curvilinear aspects within eddy-viscosity models. Additionally, they employ Detached-Eddy Simulations to scrutinize the dynamic and unsteady wake patterns. The crux of this study underscores the utmost importance of adopting physics-based methodologies and conducting rigorous validation against experimental data pertinent to flight scenarios. Furthermore, it sheds light on the specific hurdles encountered when predicting aerodynamic loads for the Orion capsule within the subsonic regime. Notably, it draws a parallel by comparing these predictions to recent empirical data.[3]

In the scholarly work of SHAFEEQUE A P et al [4], the primary focus centers on the intricate realm of Computational Fluid Dynamics (CFD) Analysis pertaining to the Atmospheric Re-Entry Module. The concept of atmospheric re-entry encompasses the mesmerizing journey of human-engineered artifacts as they traverse the threshold from the cosmic void into a planetary atmosphere. Within this cosmic ballet, re-entry modules, possessing robust, blunt-body configurations, are meticulously designed to endure the formidable crucible of high thermal flux encountered during their atmospheric descent. These re-entry capsules exhibit a distinctive structural composition, commencing with a blunt forward body gracefully tapering into a conical rear body, whose base adopts either a pristine straight or gracefully rounded configuration. Notably, these capsules are outfitted with Ablative Thermal Protection Systems (TPS), a fortification mechanism essential for safeguarding against potential peril and insulating the valuable cargo ensconced within the vehicle's confines. In the pursuit of comprehending the intricate intricacies of hypersonic flows associated with re-entry conditions, the utilization of Computational Fluid Dynamics (CFD) emerges as a financially prudent alternative to the traditional wind and shock tunnel experiments. Within the purview of CFD analysis, intricate computations account for the dynamic fluctuations in temperature, density, pressure, the subtle influence of viscous forces, and the resounding shock waves that reverberate through the encompassing exosphere. The thermal analysis emerges as an indispensable facet in the meticulous orchestration of atmospheric re-entry vehicles, as they confront the formidable crucible of severe aerodynamic heating. The purview of CFD analysis extends to encompass an exhaustive comprehension of the temperamental fluctuations in temperature, density, pressure, the capricious influence of viscous forces, and the resonating shock waves that ripple through the ambient atmosphere [4]

S. Rathnavel et al [5] focuses in the meticulous examination and intricate design of re-entry capsules employed in human spaceflight endeavors, ensuring the secure reintegration of astronauts with Earth's embrace. Their simulations encompassed a wide array of free stream conditions, encompassing subsonic, supersonic, hypersonic speeds. To derive their findings, they harnessed the compressible Navier-Stokes equation, augmented by a k-omega turbulent model. This comprehensive investigation delved into the repercussions of diverse geometric parameters pertinent to the re-entry vehicle, such as the spherical nose radius and cone angle. These parameters wield a significant influence on crucial aerodynamic phenomena, including shock wave patterns, flow separation tendencies, and wake formation dynamics. Their meticulous analysis yielded compelling results, shedding light on factors like Mach contour, pressure distribution and temperature variations across the module. It is imperative to acknowledge that the choice of the Atmospheric re-entry capsule's shape, particularly the distinctive spherical nose, was predicated upon its inherent capacity to maintain optimal aerodynamic stability. Furthermore, the parameters concerning the flare angle and shoulder radius were discovered to wield substantial sway over both the forward body configuration and the point of separation of flow. [5]

Amidst the realm of manned interstellar expeditions, the process of atmospheric re-entry emerges as a formidable crucible replete with challenges, for it exerts a direct influence upon the gravitational forces exerted and the ambient temperatures endured by the intrepid crew. At the forefront of this cosmic odyssey, the Orion Crew Exploration Vehicle, an heir to the storied legacy of the Apollo program, has ascended to the esteemed status of the quintessential blueprint for the nascent Multi-Purpose Crew Vehicle (MPCV), slated to spearhead forthcoming forays into the cosmos. In an intellectual feat of paramount innovation, C. ANBU SERENE RAJ et al [6] have proffered a groundbreaking augmentation to the incumbent Orion-based re-entry conveyance. This ingenious enhancement involves the judicious integration of a duct that artfully circumnavigates the capsule, gracefully traversing the trajectory from shoulder to base, all in a concerted endeavor to augment the craft's prowess in the perilous domain of aerocapture. The magnanimous expanse of this scholarly exploration delineates a taxonomy of design iterations predicated upon the Annular Area Ratio (AAR), a non-dimensional parameter of eminent significance. It undertakes a meticulous scrutiny of the ramifications occasioned by this augmentation upon the drag coefficient also the ballistic coefficient, these hallmarks of DRV's, and effectuates a judicious juxtaposition with the canonical archetype. The resultant findings, unveiled through a laborious crucible of scientific inquiry, lay bare the irrefutable verity that the envisaged augmentation begets a pronounced amplification of drag, while concomitantly precipitating a discernible diminution in the ballistic coefficient. The overarching ambition animating this scholarly expedition resides in the realm of the optimization of the aerodynamic comportment intrinsic to re-entry conveyances, inclusive of the venerated Orion CEV. To this august end, a panorama of prospective modifications unfurls, embracing the conceptual gamut of supple heat shield mechanisms, articulated flaps, conspicuous strakes, and deployable heat shields, all converging with the unified purpose of elevating the performance metrics and the paradigmatic design attributes that collectively constitute the crucible of re-entry vehicle. Also signifies the lodestar of this scientific voyage, offering profound insights into the domain of interstellar exploration, and heralding the prospect of uncharted vistas within the cosmos.[6]

Re-entry capsules have been designed with a blunt nose cone shape, which can impact their aerodynamic properties. This design process involves balancing factors like the heat flux, longitudinal static stability and drag coefficient. Previous research has explored shape optimization through methods like wind tunnel experiments and CFD simulations. However, making significant changes to the capsule's shape can be costly and may require additional validation of the updated design. Hirotaka Otsu et al [7] propose a novel approach by suggesting the use of hyperbolic contours to modify the front body shape of the atmospheric re-entry vehicles. Hyperbolic contours offer advantages such as smoother transitions between the nose and cone sections and easier adjustments to nose bluntness and cone angles. The primary objective of their study is to assess the aerodynamic characteristics of the atmospheric re-entry vehicles designed with hyperbolic contours through CFD analysis and compare them to capsules designed with traditional blunt nose cones.[7]

The current research is centered on the examination of the atmospheric re-entry vehicle's nose structures constructed from Functionally Graded Materials. FGMs exhibit varying compositions, transitioning from metal rich to ceramic rich materials, providing advantages such as reduction of thermal stress and enhanced heat resistance. Panneerselvam Balaraman et al [8] focused to investigate both the static and dynamic characteristics of Functionally Graded Materials shell structures designed for the nose section of atmospheric re-entry vehicles under different temperature conditions. The research involves quantifying critical buckling temperatures and natural frequencies of these FGM shells and exploring the influence of shell thickness on these properties. Ultimately, the aim is to determine the required thickness for large and thin FGM shells to withstand the thermal stresses encountered during atmospheric re-entry. The outcomes of this study are expected to advance our comprehension of the thermo-structural behavior exhibited by FGM nose structures within re-entry vehicles. Furthermore, these findings will provide valuable insights to guide the design of future re-entry capsules.[8]

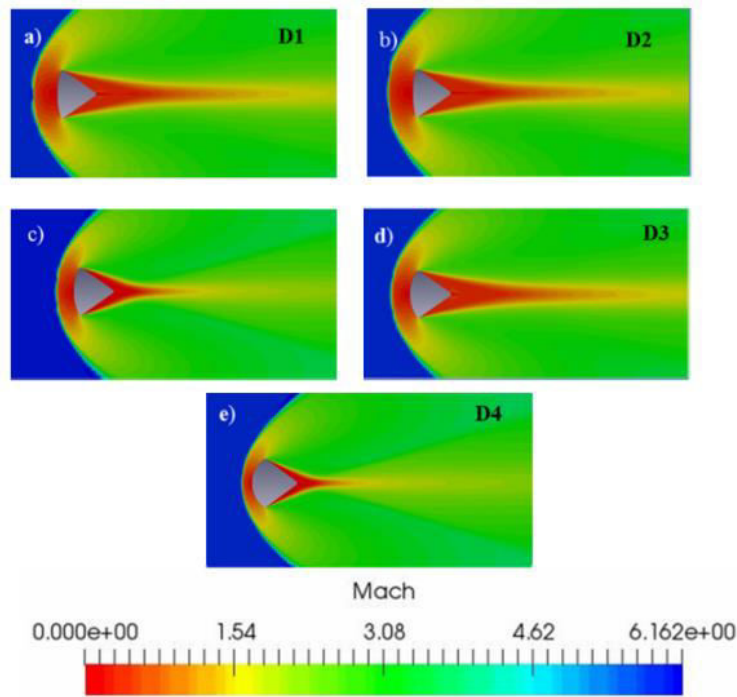
II. RESULTS & DISCUSSION

Panneerselvam Balaraman et al [8] primarily focuses on the thermal and structural analysis of nose structures for Atmospheric re-entry vehicles. These structures were constructed using functionally graded materials (FGM). The study considered 4 different FGM shell compositions: aluminum/silicon carbide, Ti-6Al-4V/silicon carbide, aluminum/aluminum oxide, and Ti-6Al-4V/aluminum oxide. The research revealed that the critical natural frequency and buckling temperature of these atmospheric re-entry vehicle nose structures decrease as the linear temperature rise and thermal environment increase. The thickness of the shell was found to have a significant impact on both the critical buckling temperature and natural frequency of FGM shells. Specifically, the FGM shell composed of Ti-6Al-4V/silicon carbide exhibited favorable buckling features in thermal applications. Additionally, the study identified that functionally graded materials shell with a thickness 0.05m and a parameter "k" value of 0.1 could be effectively utilized for atmospheric re-entry vehicle's nose structures due to its enhanced critical buckling temperature.[8]

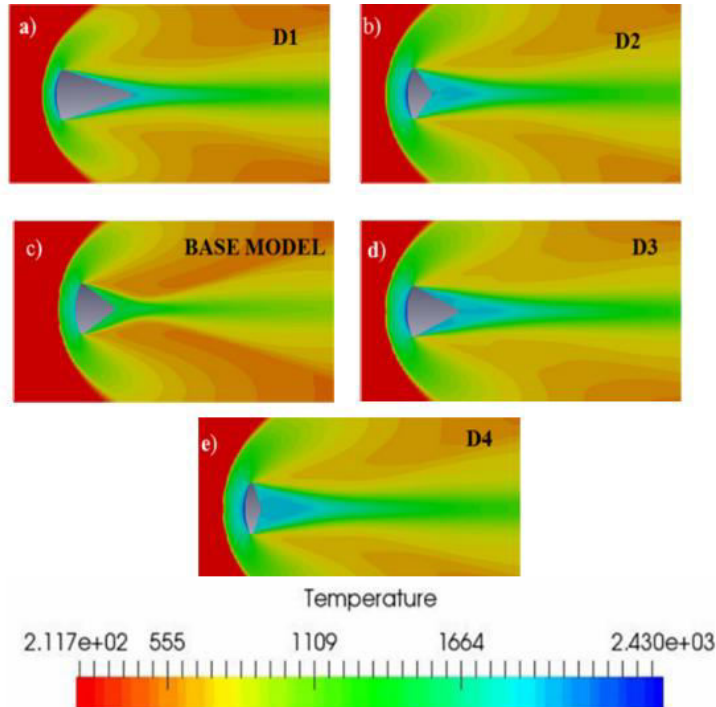
Hirota Otsu et al [7] focuses on exploring the hypersonic aerodynamic properties of atmospheric re-entry vehicles designed with hyperbolic contours. To conduct this analysis, the Computational Fluid Dynamics (CFD) code FaSTAR was utilized. The CFD results revealed that by employing hyperbolic contours, it was possible to enhance both the drag and lift coefficients of the re-entry capsules in comparison to the Hayabusa re-entry capsule. Importantly, these improvements could be achieved without significant alterations to the overall capsule shape. The study demonstrated that shape design based on the hyperbolic contours can effectively enhance the aerodynamic characteristics of atmospheric re-entry vehicles. Through CFD analyses, the research provided valuable insights into the improved drag and lift coefficients achievable with this design approach.[7]

C. ANBU SERENE RAJ et al [6] evaluated the drag and ballistic coefficient of DRV's at various Mach speeds and compared them with the baseline model. It was found that the proposed design of adding a circumventing duct significantly increased the drag for all Annular Area Ratios in the subsonic speed. In the supersonic speed, higher Annular Area Ratio configurations showed more promising results with increasing Mach number. Ducted Re-entry Vehicles also exhibited lower ballistic coefficients compared to the baseline counterparts. The addition of a circumventing duct in the DRVs reduced the ballistic coefficient significantly. The increase in drag seen in the Ducted Re-entry Vehicles was not due to the change in the effective f area but because of modified flow around and through the constant duct surrounding the baseline CEV. The pressure drag of the Ducted Re-entry Vehicles was due to two components: the pressure difference across the duct that produces a normal force on the duct and the pressure difference across the capsule that produces an axial force on the vehicle. Higher Annular Area Ratios configurations performed well under all flow regimes, while lower Annular Area Ratios configurations showed a negative increase in drag with increasing Mach number. The general trend was that with increasing Annular Area Ratios, drag contributed by the duct increased, and with increasing Mach regime, drag increased up to Mach 1.2, after which the pressure difference across the capsule is compromised, leading to a decrease in drag.[6]

S. Rathnavel et al [5] focuses on simulations that were conducted on atmospheric re-entry vehicle across different free stream conditions, including subsonic, supersonic, and hypersonic speeds. The research focused on understanding how design parameters, such as variations in the flare cone angle and front spherical nose radius, affect flow characteristics. The study analyzed and presented data on pressure distribution, Mach contours, and temperature profiles over the module. Notably, vortices were observed behind the capsule due to the wake flow field, and their presence decreased when the nose radius was reduced. Additionally, the pressure distribution at the nose exhibited a decrease with an increase in nose radius and vice versa.[5]



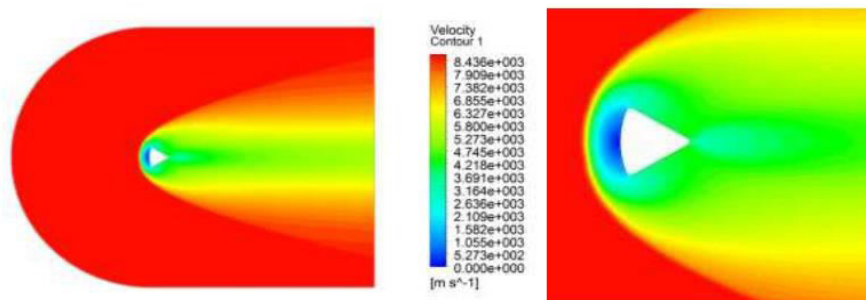
Mach distribution contour for different spherical nose radii at M=6.[5]



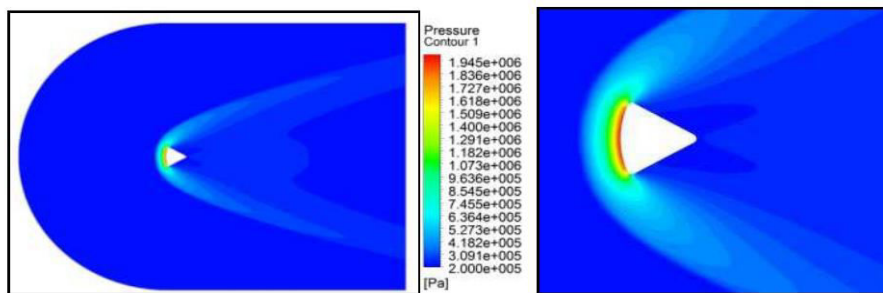
Temperature contours for different flare angles at M= 6.[5]

SHAFEEQUE A P et al [4] conducted the CFD analysis on the Apollo AS-202 re-entry vehicle, and the flow field around the vehicle was obtained using CFD. The simulation showed the formation of a bow shock also the boundary layer on the vehicle, followed by flow separation on the afterbody. A large recirculation bubble was formed on the leeward side, and a shear layer enclosing the separation bubble coalesced at the neck, forming a recompression shock.

The velocity distribution of the 2-dimensional model for Mach number 26.2 showed 0 velocity at the front body section of the re-entry vehicle and maximum velocity at the free stream. During re-entry into the earth’s atmosphere, the Apollo command module experienced convective and radiative heat fluxes. Radiative heating was approximately 1/3rd of the total heating rate, while convective heat transfer was significantly reduced by ablation products injected inside the boundary layer. The temperature contours over the capsule showed that the temperature was maximum at the heat shield, and the kinetic and potential energy decreased, dissipating as heat energy. The analysis provided information on surface wall temperatures, which are important for material selection for the survivability of the vehicle. The heat flux around the re-entry capsule body increased with increasing Mach number. The model AS-202 was tested for Mach 26.2 at zero degree and 18° angle of attack, and the wake was completely separated for the 0° model and partly separated for the 18° model. The flow separated at the shoulder and reattached downstream on the rear body frustum. The location of reattachment moved upstream with increasing angle of attack. The study demonstrated that this method of CFD analysis can provide aerodynamic information in a timely manner while retaining the cost and schedule of commercial programs.[4]

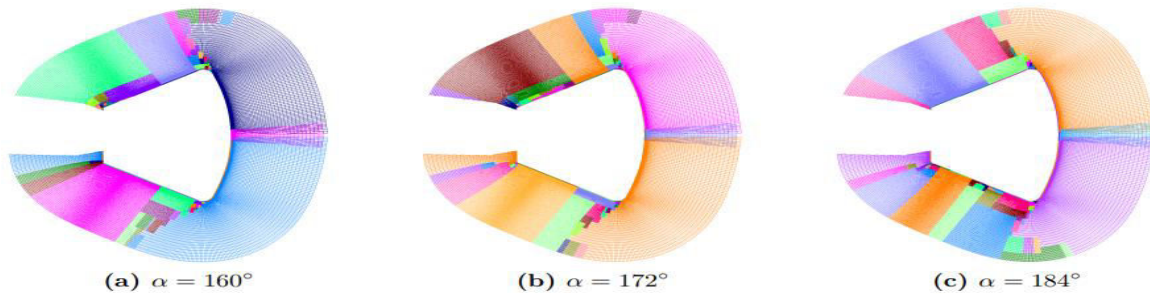


Velocity distribution contour of 2-dimensional model for Mach no =26.2.[4]

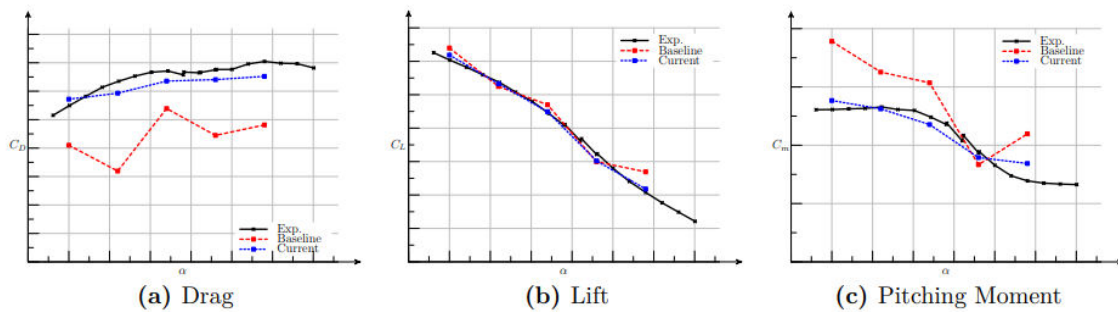


Pressure distribution contour of 2-dimensional model for Mach no =26.2.[4]

Scott M. Murman et al [3] conducted a study focusing on the accuracy of Computational Fluid Dynamics predictions regarding subsonic capsule aerodynamics. They based their analysis on recent NASA wind tunnel data obtained at high Reynolds number conditions. The study considered various modeling aspects, including numerical methods, mesh adaptation, rough wall modeling, and turbulence corrections. Their research revealed substantial enhancements in predictive capabilities achieved by integrating physics-based approaches and validating these against experimental data. Notably, the presence of surface roughness on the capsule was found to diminish suction effects near the heatshield shoulder, in line with experimental findings. By incorporating wall roughness modeling, specific turbulence models, and mesh adaptation, the study significantly improved the accuracy of simulations within the subsonic range. Furthermore, the research addressed challenges related to numerical methods and physical modeling in predicting capsule aerodynamics, offering proposed solutions that were subsequently validated through recent experimental data characterized by high Reynolds numbers.[3]

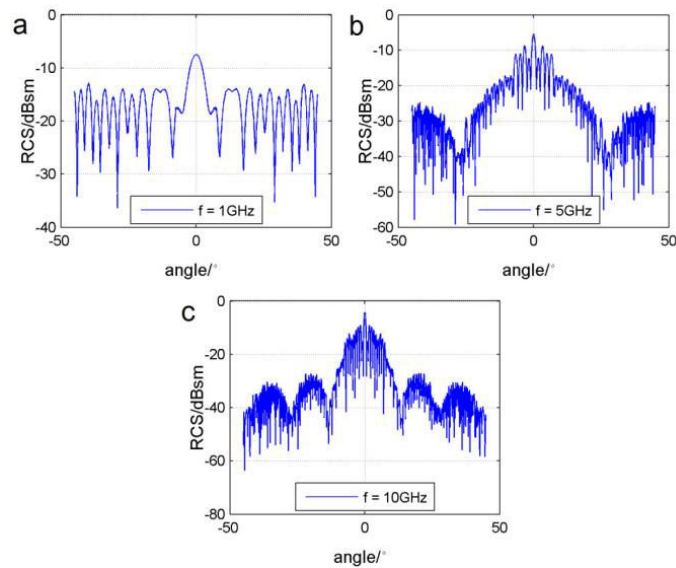


Modified overset grids conforming to the body's shape in the lateral plane of symmetry.[3]



Calculated and measured combined loads on an Orion capsule's structure using the techniques outlined in the present study. The conditions include Mach number (M_∞) of 0.5 and a Reynolds number (ReD) of 24M.[3]

Guofu Liu et al [2] focuses on the design, modeling, and analysis of a sharp edge stealthy atmospheric re-entry vehicle with a high lift-drag ratio. The authors analyze the lift-drag ratio and center of gravity to determine the configuration also the CG of the vehicle. They use the Modified Newtonian Theory for aerodynamic performance analysis, which is found to be more accurate for a sharp edge hypersonic atmospheric re-entry vehicle compared to the Shock Expansion Method and Piston Theory. The configuration of the re-entry vehicle is modeled, with dimensions of 1.68m long, 1.11m wide, and 0.525m high. The total mass, including payload, is 470 kg. The center of gravity is located at 64 percentage of the length of the vehicle, resulting in a lift/drag ratio above 2. The influence of flap angle on the center of gravity position is analyzed, the best position is found to be from 63% - 65% of the total length. The re-entry vehicle is found to have stability during the re-entry phase based on aerodynamic stability analysis. The stealth performance of the vehicle is investigated using a radar cross-section calculation method based on physical optics and the method of equivalent currents. The RCS is tested using the NASA radar cross-section Benchmark of cone-sphere. The atmospheric re-entry vehicle demonstrates good stealth performance with an radar cross-section of -10 dBm², comparable to the F117.[2]



Radar Cross Section (RCS) of the vehicle under varying incident wave frequencies: 1 GHz, 5 GHz, and 10 GHz.[2]

D. Dirx et al [1] focuses on the optimization of the shapes of two vehicle types: capsules and winged vehicles. They conducted an analysis of the aerodynamic characteristics of these vehicles using local-inclination methods and calculated entry trajectories under trimmed conditions. To optimize the vehicle shapes, the study employed multi-objective particle swarm optimization, considering objectives related to mass, volume, and range. The results of the optimization revealed significant variations in vehicle performance across the explored parameter space. Notably, the study identified areas of strong non-linearity near the two-dimensional Pareto fronts, emphasizing the importance of robustly exploring how vehicle shapes influence overall system performance during engineering trade-offs. For winged vehicles, the optimization process tended to converge toward narrow-wing solutions. Additionally, when pitch-stability was imposed for winged vehicles at all angles of attack, the resulting vehicle shapes required upward control surface deflections for most of the entry phase. In the case of capsule optimization, there was competition observed between various objectives, such as flight range and heat load, as well as between volumetric efficiency and range. Vehicle mass was identified as a significant factor driving the competition between heat load and range. The study's optimization approach demonstrated the feasibility of conducting rapid design-space exploration and optimization of re-entry vehicle shapes using conceptual methods. Importantly, the optimization process provided insights into the trade-offs between performance parameters and constraint functions, highlighting the need for careful consideration during the conceptual design phase. It's worth noting that the capsule-shape optimization took approximately 1 hour, while the winged-vehicle optimization required around 12 hours on a typical laptop computer.[1]

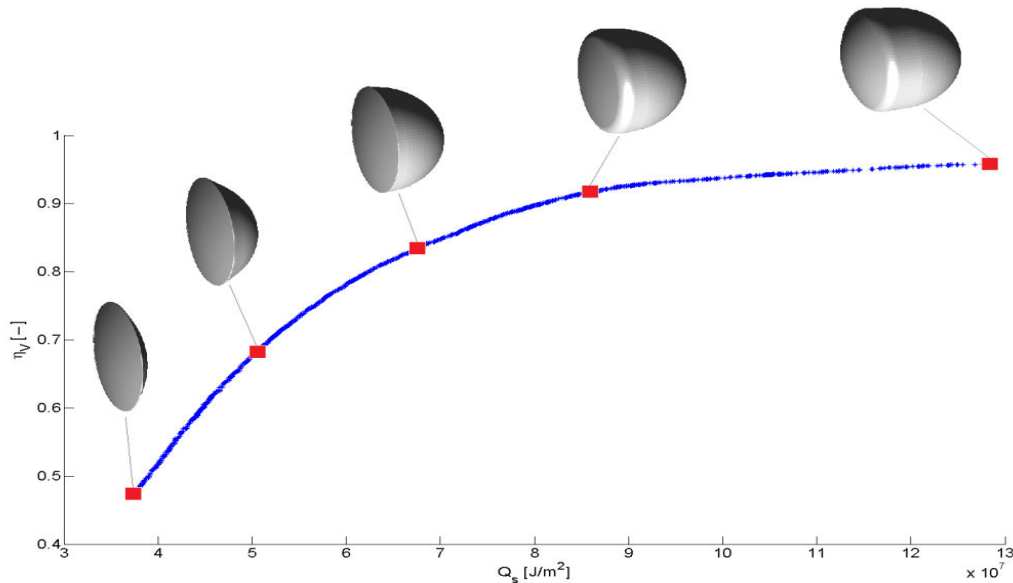


Figure of capsule shapes along the $Q_s - \eta V$ front.[1]

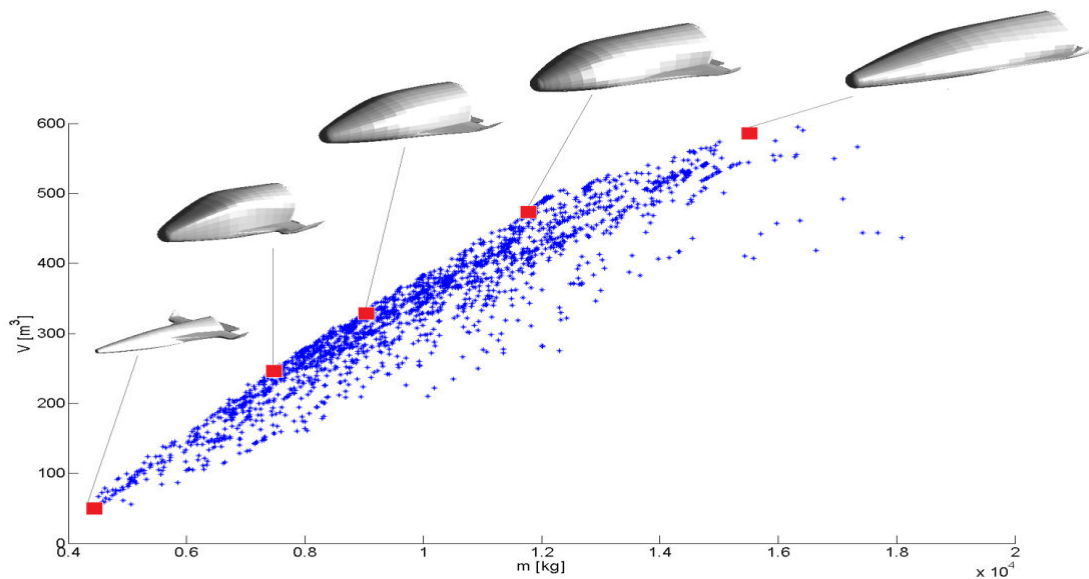


Figure of winged vehicle along the $m - V$ front, with all Pareto-optimal shapes projected onto this plane shown.[1]

III. CONCLUSION

After grasping the main ideas from the mentioned papers, the key points that can be summarized is as follows.

It explores the optimization of atmospheric re-entry vehicle's shapes, including capsules and winged fuselage configurations. It uses trajectory simulations up to Mach 3 and local-inclination methods for aerodynamic analysis. The study highlights non-linearities in shape optimization, especially for vehicle designs. A separate investigation focuses on a sharp edge stealthy atmospheric re-entry vehicle, emphasizing the influence of flap angles on aerodynamic stability. It also showcases favorable stealth performance compared to the F117. The study incorporates high Reynolds number wind tunnel data for capsule aerodynamics, enhancing predictive accuracy in the subsonic regime through improved CFD methods. Heat shield analysis reveals pressure gradient variations leading to shock wave formation,



with temperature and heat flux directly proportional to Mach number. Mach 26.2 wind tunnel testing demonstrates the method's ability to provide timely aerodynamic information, with material selection playing a crucial role in mitigating localized heating. Numerical simulations offer insights into flow fields and aerodynamics in supersonic and hypersonic regimes, impacting heat shield design. A ballistic aerocapture method using a circumventing duct is discussed, significantly reducing the ballistic coefficient and modifying drag characteristics. A study on hyperbolic contour re-entry capsules shows increased drag and lift coefficients compared to blunted nose cones. Lastly, the paper explores Functionally Graded Materials (FGM) for re-entry vehicle nose models, considering structural, thermal, and vibration characteristics. Overall, these studies provide valuable insights into re-entry vehicle optimization, aerodynamics, and structural considerations for future research and design efforts.

REFERENCES

- [1] D. Dirks, E. Mooij "Optimization of entry-vehicle shapes during conceptual design" Delft University of Technology, Faculty of Aerospace Engineering, Section of Astrodynamics and Space Missions, Kluyverweg 1, 2629 HS Delft, The Netherlands
- [2] Guofu Liu, Daochun Li, Jinwu Xiang*, Wenli Luo "Design, Modeling and Analysis of a Sharp-Edge Hypersonic Stealthy Re-entry Vehicle" School of Aeronautic Science and Engineering, Beijing University of Aeronautics and Astronautics, Xueyuan Road No.37, Haidian District, Beijing 100191, China
- [3] Scott M. Murman, Robert E. Childs*, and Joseph A. Garcia "Simulation of Atmospheric-Entry Capsules in the Subsonic Regime" NASA Ames Research Center, Moffett Field, CA, USA
- [4] SHAFEEQUE A P "CFD ANALYSIS ON AN ATMOSPHERIC RE-ENTRY MODULE" Assistant Professor, Department of Mechanical Engineering, Cochin College of Engineering and Technology, Valanchery, Malappuram, Kerala, India
- [5] S. Rathnavel, K. Balaji, Prem Kumar P. S., N. Sathesh Raja, K. Vijaya Sankaran "Investigation of Computational Flow Field over a Re-entry Capsule at High Speed"
- [6] C. ANBU SERENE RAJ, M. NARASIMHAVARADHAN, N. VAISHNAVI, S. ARUNVINATHAN, A. AL ARJANI, S. NADARAJA PILLAI "Aerodynamics of ducted re-entry vehicles" Turbulence & Flow Control Lab, School of Mechanical Engineering, SASTRA Deemed University, Tamil Nadu 613401, India, College of Engineering, Prince Sattam Bin Abdelaziz University (PSAU), Al-Kharj 11942, Saudi Arabia
- [7] Hirotaka Otsu "Aerodynamic Characteristics of Re-Entry Capsules with Hyperbolic Contours" Faculty of Advanced Science and Technology, Ryukoku University, Kyoto 612-8577, Japan
- [8] Panneerselvam Balaraman, Vijayaraj Stephen Joseph Raj, Velloorillom Madhavan Sreehari "Static and Dynamic Analysis of Re-Entry Vehicle Nose Structures Made of Different Functionally Graded Materials" School of Mechanical Engineering, SASTRA Deemed University, Thanjavur 613401, India



INNO SPACE
SJIF Scientific Journal Impact Factor
Impact Factor: 8.423



ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

9940 572 462 6381 907 438 ijirset@gmail.com



www.ijirset.com

Scan to save the contact details