

OVERFLOW Analysis of Supersonic Retropropulsion Testing on a Blunt Mars Entry Vehicle Concept

Logan D. Halstrom*

NASA Ames Research Center, Moffett Field, CA, 94035

Thomas H. Pulliam[†], Robert E. Childs[‡], and Paul M. Stremel[§]

Science & Technology Corporation, Moffett Field, CA, 94035

Supersonic Retropropulsion (SRP) flow over a Hypersonic Inflatable Aerodynamic Decelerator (HIAD) blunt-body vehicle was simulated using the OVERFLOW Computational Fluid Dynamics (CFD) solver. Two nozzle configurations were tested (1E and 1F) as a subset of the seven total configurations in the Descent Systems Study (DSS) testing campaign. A generalized, Adaptive Mesh Refinement (AMR) shock capturing and plume refinement technique was developed and calibrated for producing automatic and optimized grid systems unique to any given testing condition. Simulations were subjected to successive levels of grid refinement in order to determine solution independence from grid resolution. OVERFLOW Reynolds Averaged Navier-Stokes (RANS) solutions produced realistic SRP flow phenomena, including Mach disk normal shocks contained in high-thrust plumes and bow shock triple-points. Solutions for the 1E nozzle configuration were steady. Conversely, a subset of the 1F test conditions were unsteady and exhibited periodic, non-sinusoidal expansions and contractions of the streamwise-oriented plume, resulting in unsteady aerodynamic loads on the vehicle.

I. Nomenclature

Symbols

C_T	=	thrust coefficient
C_v	=	thermal energy coefficient
M	=	Mach number
C_D	=	drag coefficient
CO_2	=	carbon dioxide
α	=	angle of attack
C_P	=	pressure coefficient

Acronyms

SRP	=	supersonic retropropulsion
HIAD	=	Hypersonic Inflatable Aerodynamic Decelerator
CFD	=	computational fluid dynamics
AMR	=	adaptive mesh refinement
RANS	=	Reynolds averaged Navier-Stokes
SST	=	shear stress transport
EDL	=	entry, descent, and landing

*Aerospace Engineer, Systems Analysis Office (ARC-AA), AIAA Member, logan.halstrom@nasa.gov

[†]Senior Research Scientist, Systems Analysis Office, AIAA Associate Fellow.

[‡]Senior Research Scientist, Systems Analysis Office.

[§]Senior Research Scientist, Systems Analysis Office, AIAA Member.

II. Introduction

DEVELOPMENT of the science and technology required to enable human exploration of Mars has been ongoing for decades and is moving forward at an increasing pace, at NASA and other organizations. One of the remaining key technical challenges in active development is the ability to safely and reliably land significantly large payloads on the planet's surface. Mars's atmosphere is tenuousness in nature, which diminishes the deceleration that can be achieved through traditional aerodynamic drag, but is still dense enough to create complex vehicle aerodynamics when retropropulsion is used. Recent NASA Rover missions have successfully employed four stages of deceleration and landing technology: a conventional supersonic aeroshell, a super/subsonic parachute, retropropulsion, and finally a sky-crane. It has proven to be a reliable system, but it is complex and has distinct disadvantages for crewed missions. NASA's critical future mission requirement is for an entry vehicle to be able to land large payloads on the order of 20 metric tons, which is 20 times more than current successfully landed payloads on Mars. These heavy mass Mars missions include both planned precursor robotic and follow-on human landings [1, 2].

Two such vehicle concepts are the mid-L/D CobraMRV vehicle [2–4] and the HIAD (Hypersonic Inflatable Aerodynamic Decelerator) [5]. Fig. 1 gives examples of the current vision for these vehicles. Both concepts are designed for traditional aerodynamic deceleration and guidance in supersonic flight, down to $M \sim 2$, at which point retropropulsion (RP) is used for final deceleration and landing. The engines are integrated into the vehicle and shielded for initial hypersonic entry, and the nozzle exit planes are exposed for retropropulsion.



Fig. 1 Concept Mars EDL vehicles for human exploration missions [1]

The aerodynamics of RP vehicles are complex, which increases the demand on CFD modeling if to provide good accuracy. This paper is focused on the challenges of tailoring the CFD process in order to accurately represent the essential features of such flows. Principal aspects of the aerodynamic complexity are:

- The vehicle flies through the wake of its own engines' exhaust.
- The bow shock, when present, has back-pressure support from turbulent jets, and thus has the potential to be highly variable, in space and time.
- Significant shock losses through a variable shock can induce a highly variable post-shock flow that impacts the vehicle; mixed regions of strong shocks and oblique weak shock can dramatically alter the dynamic pressure in the post-shock region.
- Large scale dynamics are relatively common.
- Feedback mechanisms between plume and shock are observed.
- Issues like the modeling of turbulence and reacting flows can significantly affect CFD fidelity.
- The Mars atmosphere is poorly modeled by ideal gas laws; specifically, the thermal energy coefficient C_v varies significantly with temperature through a typical re-entry shock.
- The range of physically relevant length scales varies over the range of roughly 10^5 , based on 1cm scales for near-vehicle plume shear layers and 1km for the plume's furthest extent in low speed flight. Commensurate time scale will exist.

Attention to key details of vehicle aerodynamics is important for obtaining the best possible results, especially for non-conventional vehicle and aerodynamics. The RP rocket plumes exert a strong influence over the vehicle's aerodynamics, to the point of reducing the vehicle aerodynamic drag (excluding rocket thrust) to negative C_D , for some vehicle/nozzle combinations. These types of aerodynamic data can only be generated with CFD methods. Vehicle flight conditions in the thin, mostly CO_2 , Mars atmosphere are practically impossible to replicate in ground test facilities and truly impossible to match in the Earth's atmosphere. The accuracy of CFD methods is, therefore, impossible to validate with experimental data, at true flight conditions. Fortunately, supersonic wind-tunnel testing in air is a very

good surrogate for actual flight conditions, as the differences between Earth and Mars atmospheric gas properties are expected to be less challenging than many other issues.

A significant wind-tunnel test campaign was initiated under NASA Aerosciences Evaluation and Test Capabilities (AETC) portfolio office around 2017 to assess the relative merits of supersonic wind-tunnel testing and CFD methods for fulfilling future aerodynamic data-generation needs. A primary criterion is the ability of CFD to provided vehicle aerodynamics data that is “of comparable quality” to that produced by wind tunnel experiments. Ross et al. give a description of the objectives and scope of this effort [6]. One component of this testing campaign was to partner with NASA’s Space Technology Mission Directorate (STMD) Descent System Study (DSS) on supersonic retropropulsion (SRP) on two concept vehicles, HIAD and CobraMRV. This paper describes the work done to predict the aerodynamics of the HIAD concept using the OVERFLOW CFD solver. Companion papers [7, 8] describe the experiment, work done with several CFD solvers, and work on CobraMRV with OVERFLOW [9]. OVERFLOW is a Computational Fluid Dynamics (CFD) flow solver under development by NASA that uses structured overset grids to simulate fluid flow [10].

The current AETC SRP wind tunnel testing effort will run tests for both the mid-L/D CobraMRV and the low-L/D HIAD vehicle concepts over a variety of supersonic test conditions with varying levels of engine thrust. The final test condition matrix for the campaign includes 7 geometries $\{1A, 1B, 1C, 1D, 1E, 1F, 2A\}$, 3 freestream Mach/Reynolds number conditions $(M, Re/ft) = \{(2.386, 1e6), (3.477, 1e6), (4.568, 1.5e6)\}$, 3 angles of attack $\alpha = \{0^\circ, 5^\circ, 10^\circ\}$, and 4 thrust coefficients $C_T = \{0.0, 0.5, 1.0, 2.5\}$. A range of nozzle configurations will be tested on the HIAD in order to investigate the performance effectiveness of nozzle design parameters such as exit area, area ratio, cant angle, and location placement. The resulting 207-case test matrix is summarized in Table 1.

Table 1 Text condition matrix

Geometry	1A, 1B, 1C, 1D, 1E, 1F , 2A
$(M, Re/ft)$	(2.386, 1e6), (3.477, 1e6), (4.568, 1.5e6)
α	$0^\circ, 5^\circ, 10^\circ$
C_T	0.0, 0.5, 1.0, 2.5

For the analysis conducted in this paper, the OVERFLOW CFD solver was utilized to simulate 63 unique test matrix conditions prior to wind tunnel testing, covering a subset of the HIAD nozzle configurations (boxed in Table 1) including the 01 (power-off), 1E, and 1F nozzle configurations. These configurations are compared in Fig. 2 and nozzles 1E and 1F have similar throat, nozzle, and exit geometries, but differ in both cant angle and location.

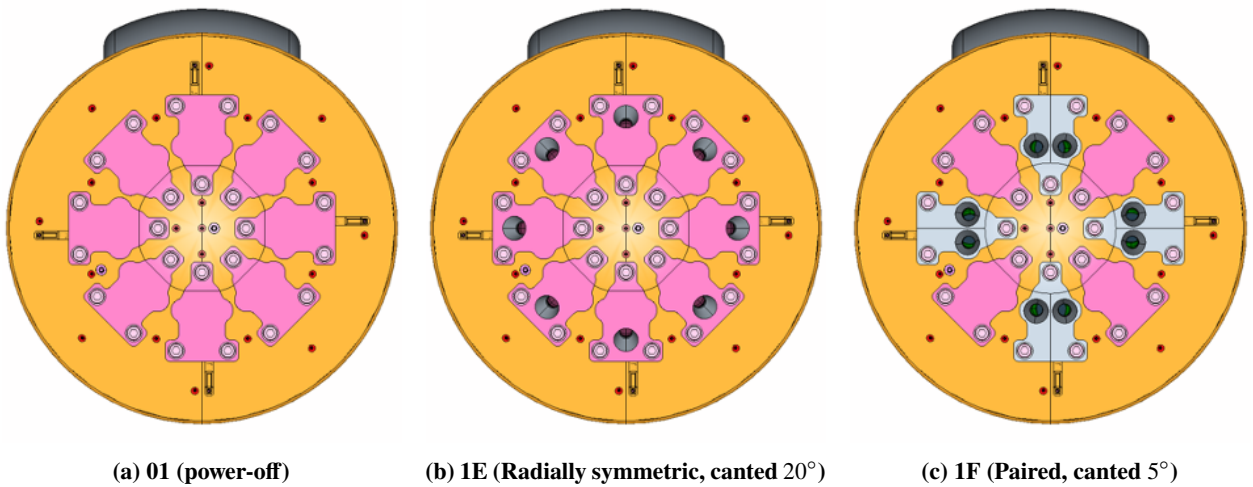


Fig. 2 Nozzle configurations for various DSS HIAD models (Source: Edquist et al. (2020) [11])

The DSS vehicle CFD simulations were performed in the test-section of the wind tunnel, using a prescribed inflow

boundary condition at the upstream end of the computational domain. The test-section flow cannot be idealized by neglecting the inflow boundary layers and subtle nonuniformity of the true test section flow. The upwash angle of the flow due to the upstream asymmetric wind tunnel nozzle-block expansion can be up to one degree in magnitude, and there are weak, wake-like features from hardware in the settling chamber that survive passage through the wind tunnel nozzle-block throat and influence the test-section flow. CFD simulations of the “full” empty tunnel were performed by several groups in the AETC effort, including calculations done with Overflow [12]. That which is referred to as the “full tunnel” takes its inflow at the approximate location of the start of the settling chamber, which coincides with the last set of turning vanes. This is approximately 30 feet upstream of the start of the wind tunnel nozzle-block contraction. The settling chamber CFD geometry contains all features larger than about one inch in size and specifically includes two recessed access port-holes, associated steps, and a “T”-rail which helps to retain the sliding nozzle-block used to vary the test section Mach number. These full-empty-tunnel CFD simulations provide the inflow to the test-section for use in the vehicle CFD work, such in those being simulated in this paper. For the OVERFLOW simulations, the grid system was constructed to have a single high-resolution “BC-grid” spanning the entire test section at a fixed streamwise location. The full-tunnel CFD results were recorded on this grid and then used as inflow boundary conditions on the BC-grid in the test-section-only grid system including the test vehicle. The full-tunnel CFD solutions were unsteady at all flow conditions, due to flow separation at the access ports and the steep leading edge of the sliding nozzle block. Instantaneous “frames” of the CFD solution were recorded on the BC-grid. These were then averaged to produce a data-set of the steady inflow boundary-condition for the test section. Thus, the test section inflow is a steady average of the CFD approximation of the true flow in the test section.

III. Computational Methodology

A. Numerical Methods

CFD simulations were performed using OVERFLOW, an implicit Navier-Stokes solver that employs finite-differencing methods on structured, overset grids [13–15]. Overset grid systems were developed using Chimera Grid Tools (CGT) [16–18] and its associated Graphic User Interface (GUI) Overgrid [17]. Overset region communication, interpolation stencils, and cell blanking were computed using Domain Connectivity Function (DCF) [19, 20].

All solutions were computed using a time-accurate solver with five Newton sub-iterations at each time step to improve convergence. Inner-iteration convergence was assessed for each solution to ensure accurate numerics. Time was advanced using improved SSOR and HLLC upwind scheme throughout the tunnel test section and near-body region. HLLE++ upwind scheme was discretely employed in the bow shock grid block region for more accurate shock capturing. Time step size was determined by successively decreasing the step size until the behavior of the results became independent of this change.

The two-equation Shear Stress Transport (SST) Reynolds-Averaged Navier-Stokes (RANS) turbulence model was employed [21] in the SRP calculations, as it has been shown to be better than other models available in OVERFLOW for flows with rocket plumes and other significant non-wall-bounded shear layers. Much of this type of work was done on the Orion Launch Abort Vehicle and re-entry crew module, for which extensive experimental data was available for model validation [22, 23]. Additionally, specific model options were found to be very important in obtaining the best possible CFD accuracy. For the under-expanded rocket plumes in SRP work, it was found that enabling the rotation and curvature (RC) option and disabling the compressibility (CC) option was a superior choice. This is counter-intuitive, as high Mach number flow generally benefits from the CC option. We explicitly depend on some error cancellation. The RC model often predicts insufficient shear layer growth in the barrel-shaped shear layers of under-expanded plumes, so disabling the CC model helps compensate for that deficiency, in our aggregate experience. This choice is not suitable for all high speed jets, but works well for the under-expanded ones like in these flows. It was also found that the QCR model [24, 25] is very beneficial in corner regions of the wind tunnel, but it has adverse effects in plumes. Overflow permits the use of different model options in different grids, so QCR is used in the grids that carry the bulk of the wall flow, including its corners, but QCR is disabled in all grids near the vehicle.

IV. Overset and Adaptive Mesh Methodology

A. Overset Grid System

This section describes the main overset grid system for the HIAD, engine BCs, and wind tunnel sting (see Fig. 5). It introduces the strategy of using Adaptive Mesh Refinement (AMR) to capture and refine the SRP flow.

B. Grid Refinement Study

When running preliminary simulations for this vehicle, it was determined that aerodynamic loads were dependent on the accuracy of the plume expansion prediction of the CFD solution, which was, in turn, dependent on the mesh resolution. Successive refinement of the AMR grids was used to refine the solution of the bow shock [26] and to determine grid-independence of the plume solution. Convergence was assessed by observing changes in the time-accurate aerodynamic loads, as depicted in Fig. 3.

Four levels of refinement were tested. Loads convergence prior to 1500 iterations represents the full multigrid and base, coarse grid solutions. The second level of refinement was initiated at 1500 iterations, the third at 2000, and the fourth and final at 4000. The magnitude of the change in heatshield axial force decreased with each level of refinement, until it became small enough to consider the solution grid-converged.

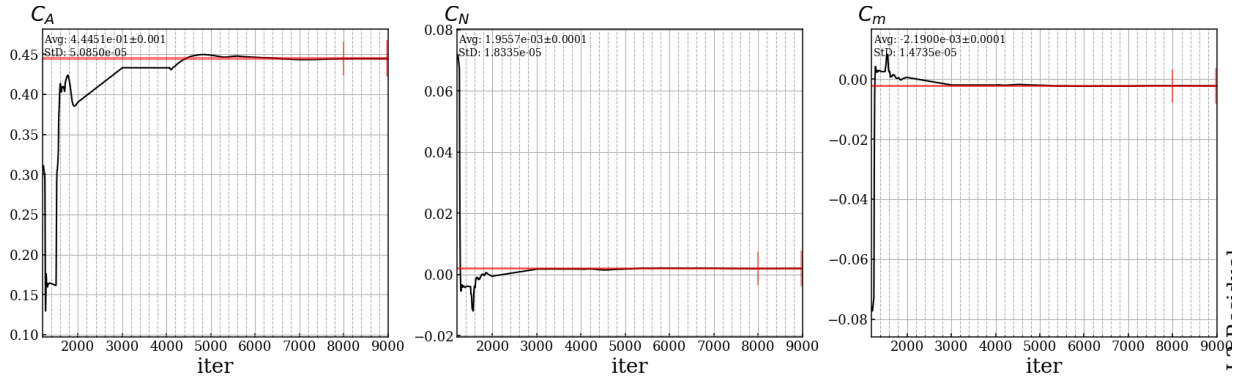


Fig. 3 Force and moment convergence history with successive doubling of AMR fineness at 1500, 2000, and 4000 iterations.

The significant variations in the integrated aerodynamic loads due to grid refinement observed in Fig. 3 can be attributed to the resolution of fundamental flow features like the Mach disk in the high-thrust plumes. Fig. 4 shows two OVERFLOW solutions on a coarse and a fine grid for the same test condition using identical solver settings. The grid cells in the plume adaption box in Fig. 4b have been refined to one half the size of their counterparts in Fig. 4a. This level of increased grid resolution allows the solver to identify and resolve the inflection point where the plume oblique shock transitions into a normal shock or Mach disk.

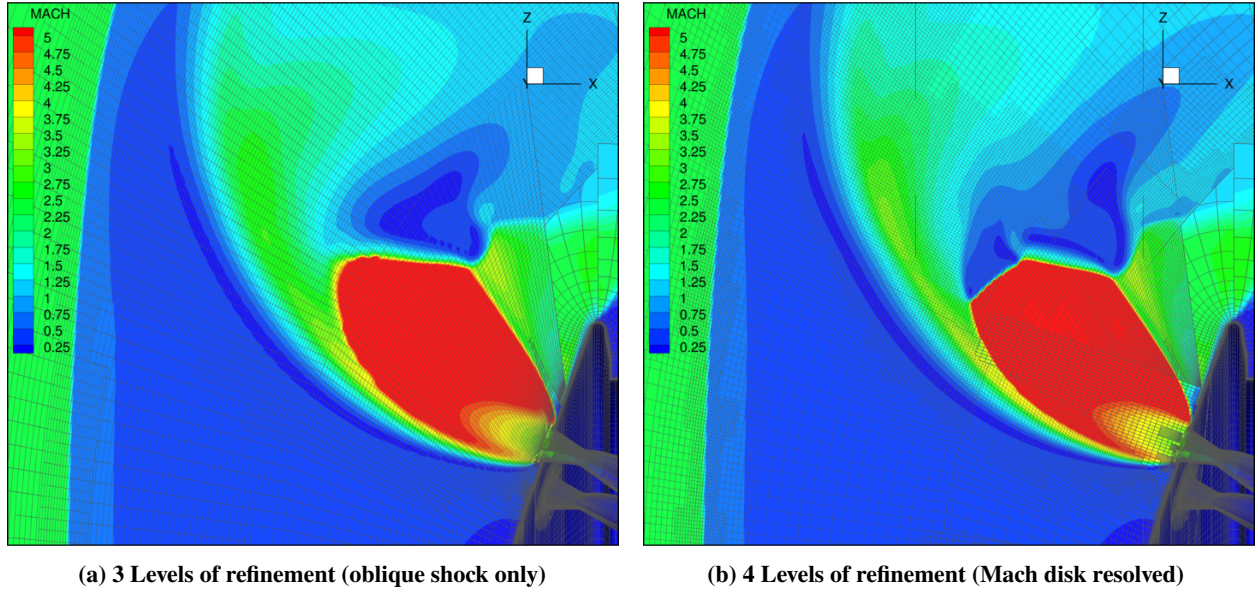


Fig. 4 Enhanced refinement of the plume-capturing grids results in Mach disk (SRP plume expansion solution for HIAD 1E at $M = 2.4, C_T = 2.5, \alpha = 0^\circ$)

C. Adaptive Mesh Plume Refinement Strategy

Even more so than typical supersonic bluff-body flows, supersonic retropropulsion flows exhibit an extreme variance in the shape and location of the bow shock due to the complicated interaction of the shock and the plumes. Initially, the OVERFLOW HIAD SRP grids were designed with bespoke, or custom-made plume and shock refinement boxes for each flight condition to capture these variations. However, the process for generating these grids was tedious and iterative, and the final grid system was often a compromise between optimal designs for different conditions.

Through testing and comparison, it was found that a more optimal and efficient approach was to remove the bespoke refinement boxes (Fig. 5b) entirely and, instead, capture the shock and plume system using Adaptive Mesh Refinement (AMR) on a large surrounding box grid (Fig. 5a). This technique has the advantage of both eliminating the guesswork of predicting the location of the bow shock and size of the plume expansion as well as providing an adapted grid that is optimally refined for the specific flight condition. Comparison to bespoke plume refinement grid solution to the generalized adaption solutions demonstrates similar resolution of SRP flow features, establishing confidence that the automated AMR approach produces consistent flow solutions.

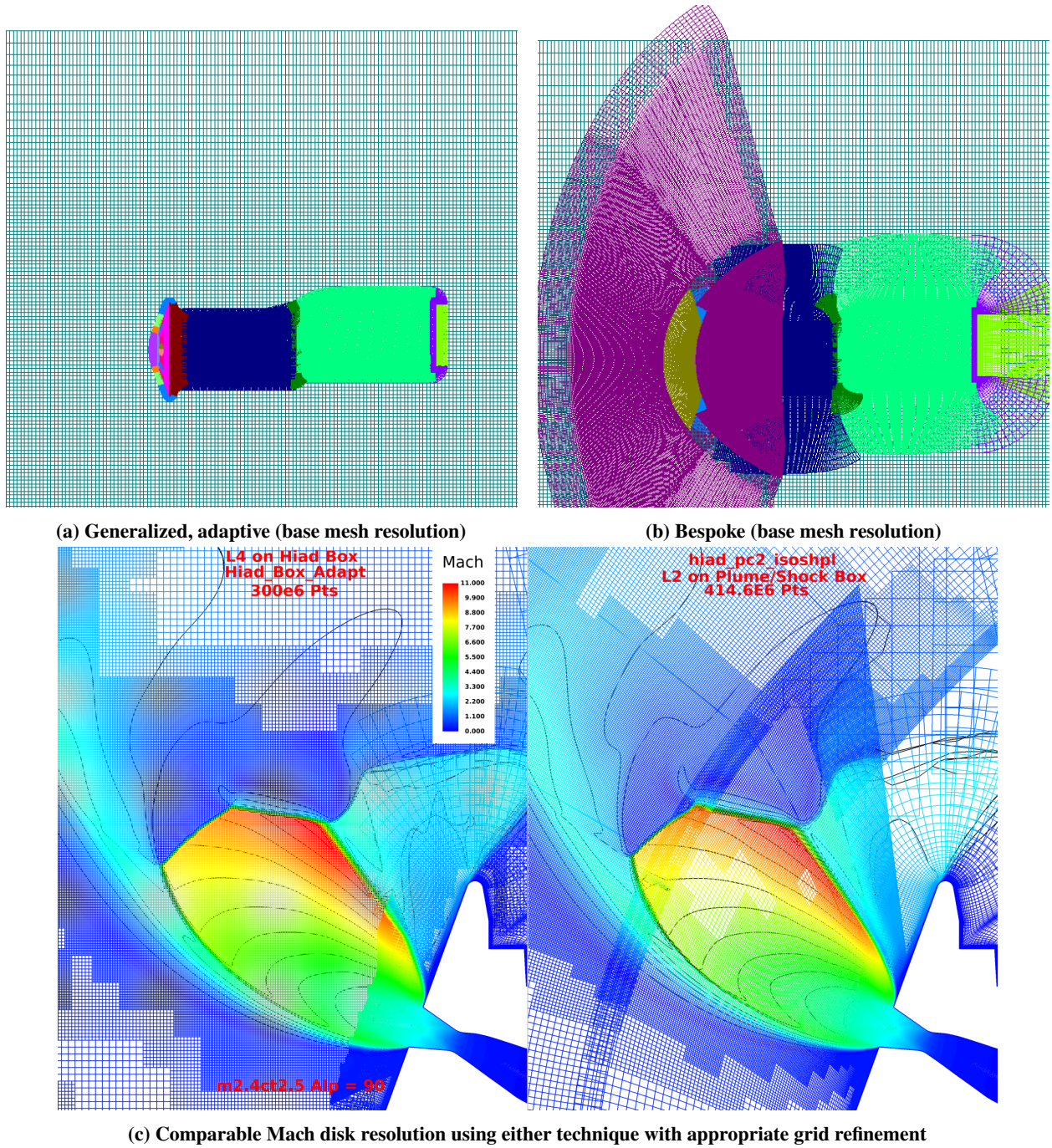


Fig. 5 Generalized, adaptive and bespoke shock/plume capturing grid systems and SRP flow solutions

The effectiveness of generalized AMR shock capturing and plume refinement can be observed in Fig. 6, which demonstrates the significant variation in bow shock and plume shape and location between thrust conditions. The figure shows that AMR is effective in detecting these variations and providing appropriate grid refinement in the regions of interest. An added benefit of the generalized AMR technique is a reduction in the total number of grid points in a solution, as AMR only refines the adaption box in regions with significant gradients, improving the overall efficiency of the solution.

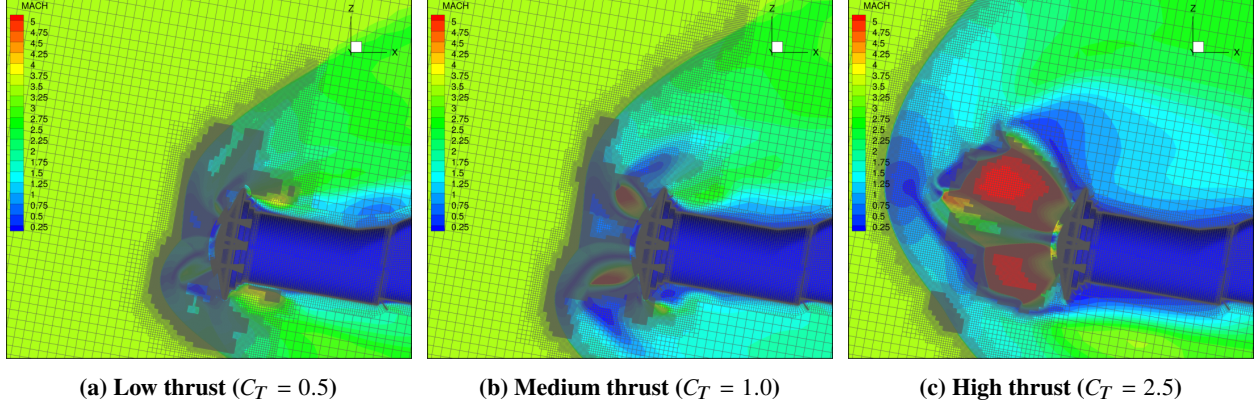


Fig. 6 Performance of generalized plume adaption box at multiple thrust conditions for HIAD 1F ($M = 3.5, \alpha = 80^\circ$)

V. Pre-Test CFD Results

This section reports the CFD results for the HIAD 1E and 1F configurations.

A. Aerodynamic Characteristics

This section focuses on aerodynamic performance trends as a function of independent database parameters: α, C_T, M . Fig. 7 depicts the shock-plume interaction and resulting heatshield surface pressure distribution over both HIAD configurations for various thrust coefficients at a single angle of attack $\alpha = 10^\circ$ and tunnel condition $M = 3.5$. In the figure, engine thrust increases left to right, with the left-most column corresponding to the power-off or drag-only condition (which is identical for the 1E and 1F configurations as it is simply the Outer Mold Line of the HIAD vehicle with no engine nozzles). The power-off condition exhibits the typical convex bow shock expected to develop over a supersonic bluff-body. As thrust coefficient is increased, the bow shock is pushed forward and augmented into a concave shape by the nozzle plumes, and a shock triple-point also occurs for non-zero angle of attack. Plume expansion is relatively similar between nozzle configurations for all thrust conditions except the maximum $C_T = 2.5$, where the 1F plumes are significantly underexpanded, returning the bow shock to a convex shape.

Nozzle location has a significant effect on heatshield pressure distribution. In the lower left of Fig. 7, we see the baseline surface pressure distribution of a symmetric bluff-body, with diminishing pressure contours centered about the stagnation point. For the power-on cases, the pressure distribution is augmented near each nozzle by impinging high-speed flow from the plumes. The radially symmetric nozzle distribution of the 1E configuration results in a more distributed pattern of plume pressure shadows. Conversely, the 1F configuration produces concentrated regions of high-speed flow between nozzle clusters, resulting in an oscillating stagnation region that is not radially symmetric. At the highest thrust condition, the plumes have propagated the bow shock further upstream, causing the surface pressure distributions to appear more symmetric and, in the case of model 1F, reducing the distribution to nearly freestream pressure across the majority of the surface.

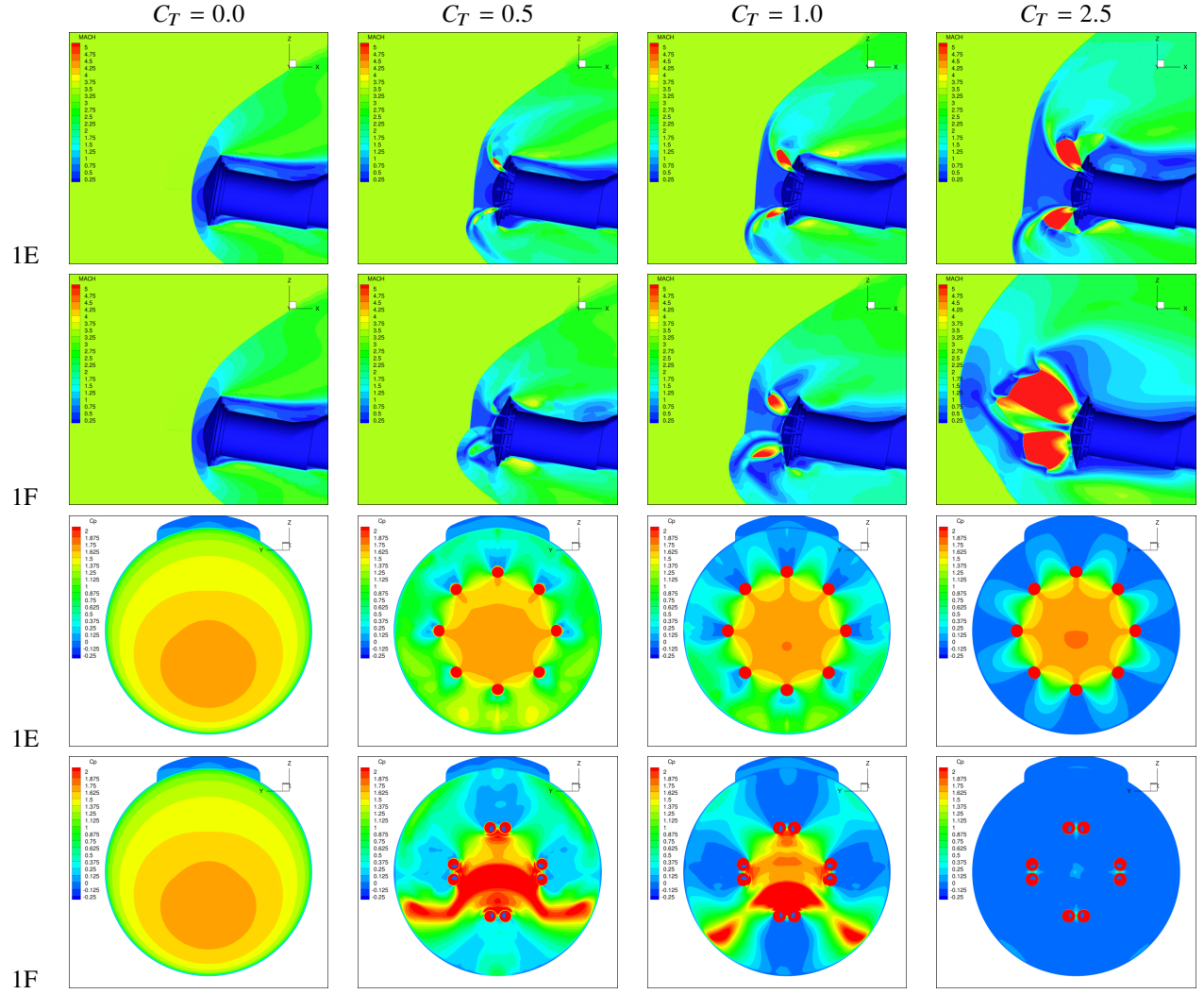


Fig. 7 Instantaneous $y = 0$ -plane Mach and heatshield surface C_p distribution for both models at $M = 3.5$, $\alpha = 10^\circ$ as a function of thrust coefficient

Fig. 8 demonstrates the variation in plume expansion behavior for a single nozzle configuration (1F) and tunnel condition ($M = 3.5$), where columns to the right represent increasing engine thrust and descending rows increasing angle of attack. The shock triple-point only occurs for low and medium thrust conditions at non-zero angle of attack.

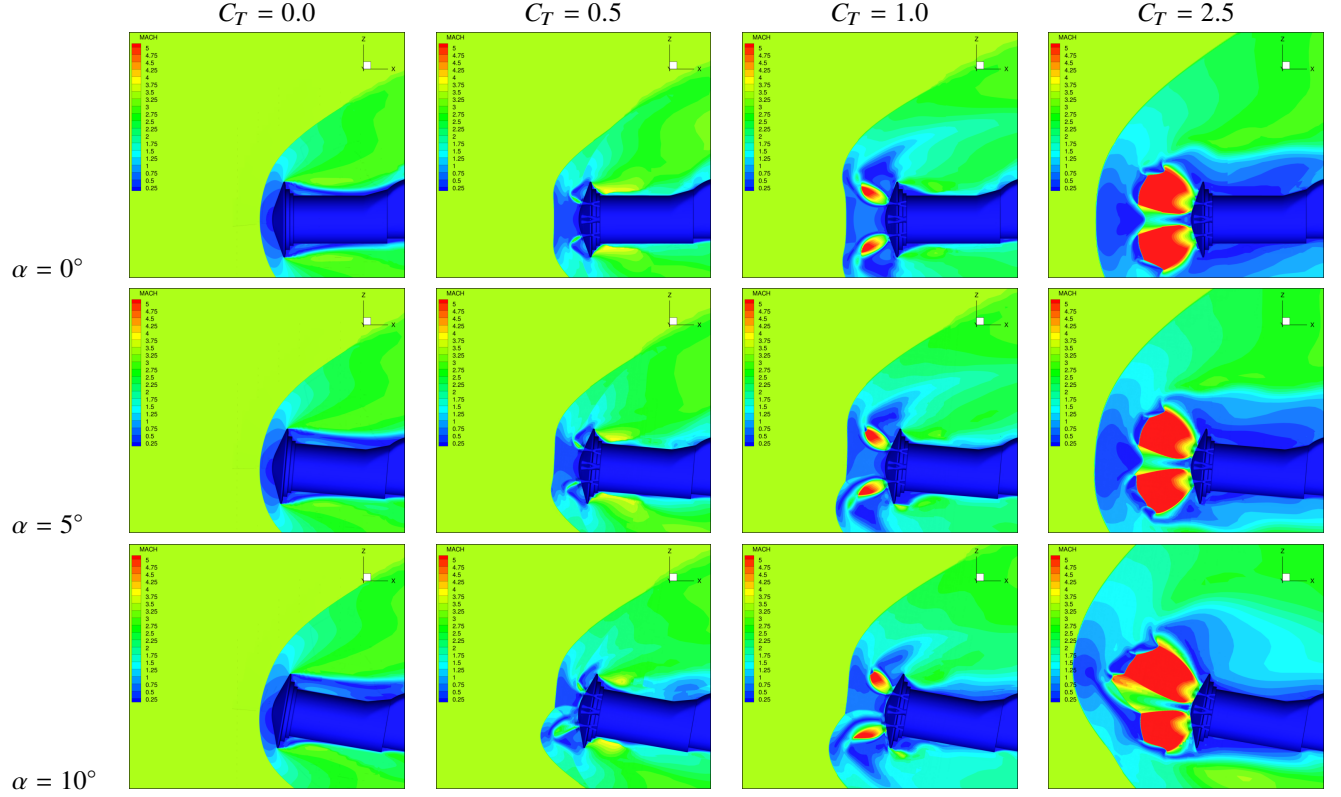


Fig. 8 Mach number contours for HIAD 1F at $M = 3.5$ as a function of angle of attack and thrust coefficient

B. Unsteady Flow Characteristics

The HIAD 1F nozzle configuration is particularly prone to unsteady oscillations at conditions with higher freestream Mach number, lower thrust, and non-zero angle of attack. The unsteady flow phenomena consists of a shifting of the shock triple-point location (depicted in Fig. 9) due to a periodic expansion/contraction of the plume from the most streamwise-aligned lower nozzle cluster. The effect on the aerodynamic body is a gross shift in the stagnation region's shape and location, leading to significant variation in the drag, lift, and pitching moment on the vehicle.

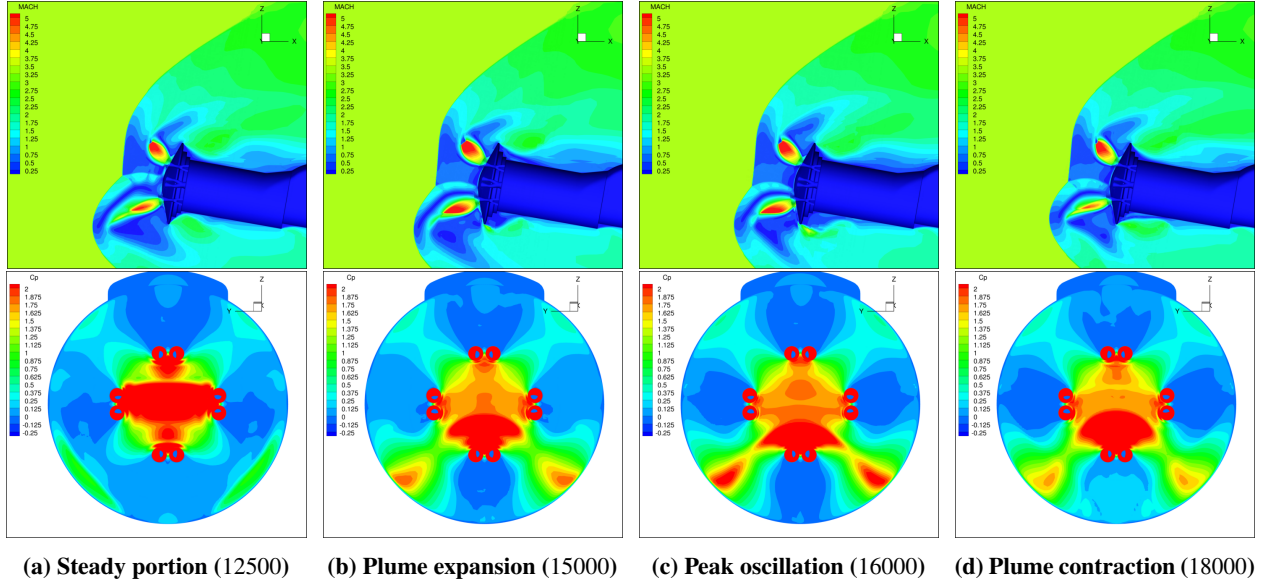
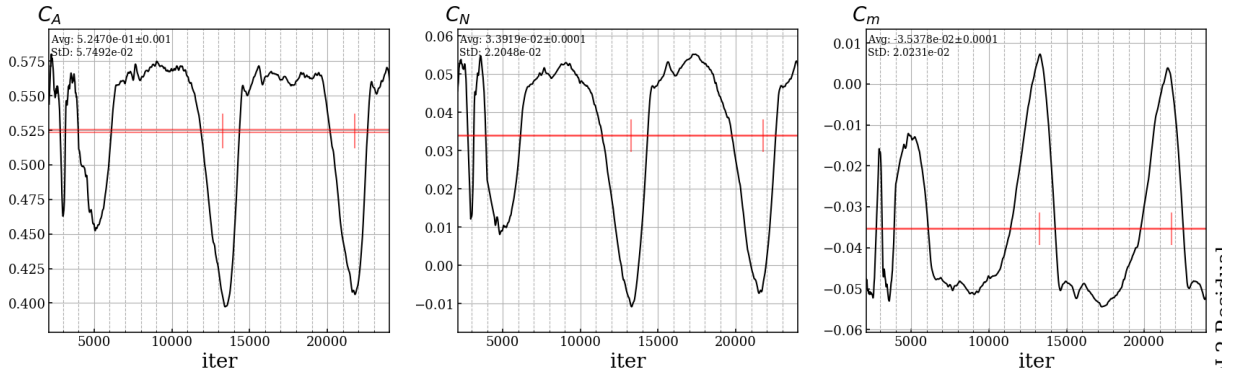


Fig. 9 Time-varying, “chugging” plume oscillation of HIAD 1F ($M = 3.5, C_T = 1.0, \alpha = 10^\circ$), ordered by time step corresponding to Fig. 10

The nature of this oscillation is a cyclic, non-sinusoidal “chugging”, with a period of $315Hz$, as shown in Fig. 10.



C. Vehicle Aerodynamic Performance

An initial comparison of nose-referenced pitching moment between configurations can be accomplished by observing the $\frac{dC_m}{d\alpha}$ trends in Fig. 11. The 1E and 1F nozzle configurations both have negative, restorative $\frac{dC_m}{d\alpha}$ at lower angles of attack, but this trend reverses for the model 1F at the highest tested angle of attack. Inferior performance at high angle of attack and inconsistent performance at low thrust due to unsteady shock-plume interaction make the 1E a potentially better choice for stable atmospheric entry.

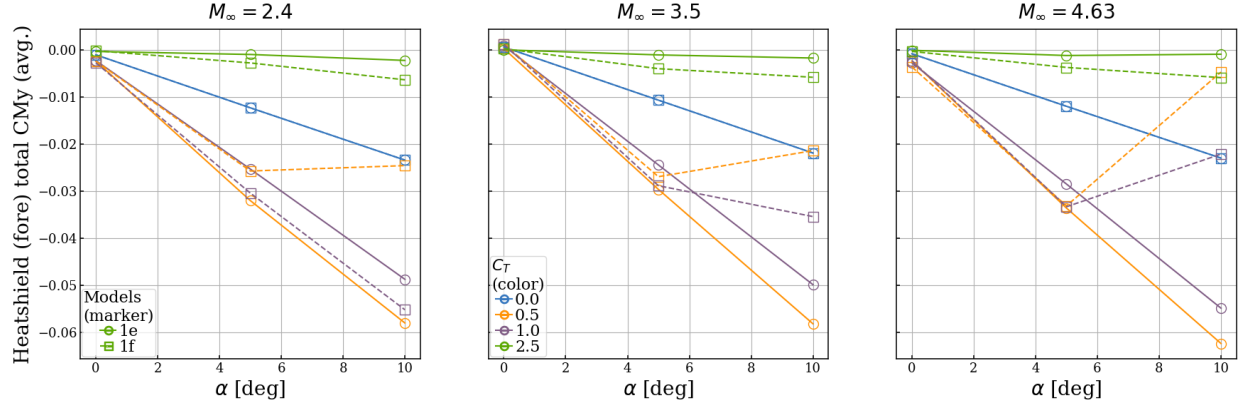


Fig. 11 Pitching moment of various HIAD configurations

It is also important to consider the effectiveness of the vehicle as an aerodynamic decelerator for the Mars EDL mission. For the HIAD geometry, a significant amount of deceleration is achieved from the reverse thrust of supersonic retropropulsion, but the aerodynamic drag coefficient of the heatshield, shown in Fig. 12, must also be accounted for. The 1E configuration tends to have notably more drag for a given amount of thrust, and, in addition, the 1F configuration has a drag reversal at the highest thrust coefficient. These characteristics indicate that the 1E should perform better at the Mars EDL task.

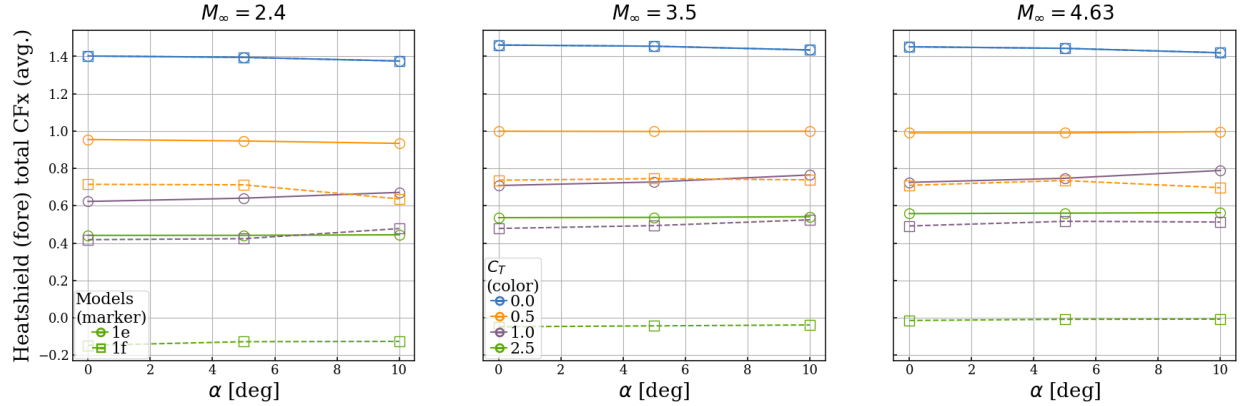


Fig. 12 Drag performance of various HIAD configurations

VI. Conclusion

This paper presented the preliminary analysis of supersonic retropropulsion over a blunt Mars Entry vehicle using OVERFLOW CFD solutions.

A generalized, adaptive shock capturing and plume refinement methodology was developed and calibrated in order to produce automatically optimized overset SRP grid systems for a variety of testing conditions.

Simulations of the HIAD 1E configuration were steady for all test conditions, while the 1F configuration tended towards unsteady, periodic oscillations of the plume and shock at low thrust, high angle of attack conditions. Both HIAD configurations showed similar pitching moment trends during SRP at low angle of attack, but the performance of the 1E was more consistent throughout the range of test conditions as the 1F pitching moment diverged at high angle of attack.

In future, post-test simulations, further attention is needed in resolving the time-accurate behavior in these flows. The small subset of cases that were unsteady and non-periodic might be better modeled with a reduced time step or increased sub-iterations. Improvement of turbulence modeling fidelity in the form of Detached Eddy Simulation (DES)

may also be pursued to more accurately represent unsteadiness in the bluff-body wake. Finally, better models of the wind tunnel test section inflow boundary conditions will be available and should help address uncertainty quantification in the current simulations.

Acknowledgments

This work is supported by the joint effort between NASA's Aerosciences Evaluation and Test Capability program office under the Aeronautics Research Mission Directorate and the Game Changing Program's Descent System Study (DSS) under the Science Technology Mission Directorate. CFD simulations were run on NASA Advanced Supercomputing Division resources and were funded by the Game Changing Development Program's Descent System Study. The authors thank the AETC and DSS teams for their advice and review of research.

References

- [1] Cianciolo, A., Korzun, A., Samareh, J., Sostaric, R., Calderon, D., and Garcia, J., "Human Mars Entry, Descent, and Landing Architecture Study: Phase 3 Summary," 2020.
- [2] Cerimele, C. J., Robertson, E. A., Sostaric, S. R., Campbell, C. H., Robinson, P., Matz, D. A., Stachowiak, S. J., Garcia, J. A., Bowles, J. V., Kinney, D. J., and Theisinger, J. E., "A rigid mid-lift-to-drag ratio approach to human Mars entry, descent, and landing," *AIAA Guidance, Navigation, and Control Conference*, 2017, p. 1898.
- [3] Polsgrove, T., Dwyer-Cianciolo, A. M., Robertson, E. A., Percy, T. K., Samareh, J., Garcia, J., Lugo, R., Sostaric, R., Cerimele, C., and Garcia, J. A., "Human Mars Entry, Descent, and Landing Architecture Study: Rigid Decelerators," *2018 AIAA SPACE and Astronautics Forum and Exposition*, 2018, p. 5192.
- [4] Calderon, D., Sostaric, R. R., Garcia, J. A., Bowles, J., Gaytan, C., Newton, H., Amar, J., and Wiens, Z., "Structural Mass Optimization with Manifest Packaging, and Outer Mold Line Updates of a Rigid Mid Lift-to-Drag Mars Entry Lander Vehicle," *AIAA Scitech 2020 Forum*, 2020, p. 1511.
- [5] Cianciolo, A. D., Dillman, R., Brune, A., Lugo, R., Polsgrove, T., Percy, T. K., Sutherlin, S., and Cassell, A., "Human Mars Entry, Descent and Landing Architecture Study: Deployable Decelerators," *AIAA SPACE*, 2018.
- [6] Ross, J. C., Rhode, M. N., Falman, B. E., Edquist, K. T., Schoenenberger, M., Witte, D. W., Brauckmann, G. J., Kleb, B. L., West, T. K., and Alter, S. J., "Evaluation of CFD as a Surrogate for Wind-Tunnel Testing for Mach 2.4 to 4.6 - Project Overview," *Submitted to AIAA Aviation 2021 Forum*, 2021.
- [7] Edquist, Falman, Watkins, Burns, Pham, and Morr, "Testing of Mars Retropropulsion Concepts in the Langley Unitary Plan Wind Tunnel," 2022.
- [8] Edquist, Childs, Matsuno, Halstrom, Canabal, Korzun, Wood, Glass, and Alter, "Status of Computational Modeling of Mars Retropropulsion Concepts Tested in the Langley Unitary Plan Wind Tunnel," 2022.
- [9] Matsuno, Childs, Stremel, Garcia, and Pulliam, "OVERFLOW Analysis of Supersonic Retropropulsion Testing on the CobraMRV Mars Entry Vehicle Concept," 2022.
- [10] Buning, P. G., "NASA OVERFLOW CFD Code," <https://overflow.larc.nasa.gov/>, 2021. Accessed: 2021-05-20.
- [11] Edquist, K. T., Korzun, A. M., Kleb, W. L., Hawke, V., Rizk, Y. M., Olsen, M. E., and Canabal, F., "Model Design and Pre-Test CFD Analysis for a Supersonic Retropropulsion Wind Tunnel Test," *AIAA Scitech 2020 Forum*, 2020, p. 2230.
- [12] Childs, R., Stremel, P., Hawke, V., Garcia, J., Kleb, B., Hunter, C., Rivers, M., Parikh, P., Patel, M., Alter, S. J., and Salari, K., "Flow Characterization of the NASA Langley Unitary Plan Wind Tunnel, Test Section 2: Computational Results," *Submitted to AIAA Aviation 2021 Forum*, 2021.
- [13] Buning, P. G., Jespersen, D. C., Pulliam, T. H., Chan, W., Slotnick, J. P., Krist, S., and Renze, K. J., "Overflow user's manual," *NASA Langley Research Center, Hampton, VA*, 2002.
- [14] Nichols, R., Tramel, R., and Buning, P., "Solver and turbulence model upgrades to OVERFLOW 2 for unsteady and high-speed applications," *24th AIAA Applied Aerodynamics Conference*, 2006, p. 2824.
- [15] Nichols, R. H., and Buning, P. G., "User's Manual for OVERFLOW 2.1," *University of Alabama and NASA Langley Research Center*, 2008.

- [16] Rogers, S., Roth, K., Nash, S., Baker, M., Slotnick, J., Whitlock, M., and Cao, H., "Advances in overset CFD processes applied to subsonic high-lift aircraft," *18th Applied Aerodynamics Conference*, 2000, p. 4216.
- [17] Chan, W., "The overgrid interface for computational simulations on overset grids," *32nd AIAA Fluid Dynamics Conference and Exhibit*, 2002, p. 3188.
- [18] Chan, W., "Developments in Strategies and Software Tools for Overset Structured Grid Generation and Connectivity," *20th AIAA Computational Fluid Dynamics Conference*, 2011, p. 3051.
- [19] Meakin, R., "Object X-rays for cutting holes in composite overset structured grids," *15th AIAA Computational Fluid Dynamics Conference*, 2001, p. 2537.
- [20] Chan, W. M., Kim, N., and Pandya, S. A., "Advances in domain connectivity for overset grids using the x-rays approach," 2012.
- [21] Menter, F. R., "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA journal*, Vol. 32, No. 8, 1994, pp. 1598–1605.
- [22] Childs, R., Garcia, J., Melton, J., Rogers, S., Shestopolov, A., and Vicker, D., "Overflow Simulation Guidelines for Orion Launch Abort Vehicle Aerodynamic Analyses," *29th AIAA Applied Aerodynamics Conference*, 2011, p. 3163.
- [23] Childs, R. E., Garcia, J. A., Rogers, S. E., and Vicker, D. J., "Overflow Aerodynamic Simulation of the Orion Launch Abort Vehicle," *Joint Army Navy NASA Air Force*, 2013.
- [24] Mani, M., Babcock, D., Winkler, C., and Spalart, P., "Predictions of a supersonic turbulent flow in a square duct," *51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, 2013, p. 860.
- [25] Spalart, P. R., "Strategies for turbulence modelling and simulations," *International journal of heat and fluid flow*, Vol. 21, No. 3, 2000, pp. 252–263.
- [26] Buning, P. G., and Pulliam, T. H., "Near-body grid adaption for overset grids," *46th AIAA Fluid Dynamics Conference*, 2016, p. 3326.