

Aerodynamic Analysis and Database Development of a Deployable Heatshield Reentry Vehicle

Zohar R. Hoter

Outpost Technologies Corporation, Playa Vista, CA, USA

Gabriel C. Nastac and Kevin E. Jacobson

NASA Langley Research Center, Hampton, VA, USA

Grant E. Palmer

AMA, Inc. at NASA Ames Research Center, Moffett Field, CA, USA

The Carryall Block 1 reentry vehicle being developed by Outpost Space utilizes a strut supported semi-rigid deployable heatshield. The advantages of a deployable heatshield include reduced head loading and earlier deceleration in the trajectory. Deployable heat shield geometries also come with difficulties in analysis including complex wake features, near rarified flow conditions at primary points of interest in the trajectory, and strong shocks. The Carryall Block 1 is analyzed using NASA's FUN3D and DPLR CFD solvers. The aerodynamic results are found to be within a percent for both solvers and both structured and unstructured mesh types. The aerodynamic database is then developed for the reentry trajectory. Fluid Structure Interaction (FSI) analysis is currently a work in progress and will be expanded on in the future.

I. Introduction

As the rate of annual launches and mass transferred to space increases, technologies to return mass to Earth, especially high value cargo and in space manufactured products, must be urgently developed. Deployable heat shields, a form of aerodynamic decelerator, work by increasing the frontal cross sectional area and thereby the drag per payload mass. The ratio of vehicle mass to drag is represented by the ballistic coefficient

$$BC = \frac{m}{C_d A}$$

where A is the frontal cross sectional area. This lowers the heat pulse of reentry significantly as the deceleration and kinetic energy is bled to lower density air higher in the atmosphere. Several low ballistic deployable heat shield concepts have been developed and some even tested.

NASA's Adaptable Deployable Entry and Placement Technology (ADEPT) program developed a strut supported fabric aeroshell. The program culminated in the SR-1 sounding rocket test of a 0.7 m vehicle which collected aerodynamic and GNC data on the configuration [Dutta, 2022]. The Low-Earth Orbit Flight Test of an Inflatable Decelerator (LOFTID) mission demonstrated a six meter inflatable heat shield from orbital reentry velocities. The vehicle was successfully recovered along with its reconstructed trajectory and aerodynamic performance [Hollis, 2025].

Outpost Space is developing a strut supported flexible thermal protection system as part of the Carryall Block 1 vehicle shown in **Figure 1**. The Carryall Block 1 will return 200 kgs of payload sized to ISS double middeck locker

standards to support scientific and in space economy missions. The deployable heat shield is a three meter flexible carbon-based fabric stretched over radial spars. A sphere cone angle of 70 degrees is considered in this paper. An octagonal satellite bus sits behind the shield with a center of gravity 0.16D of the shield diameter behind the tip of the nose. The analysis geometry has a cylindrical satellite bus.

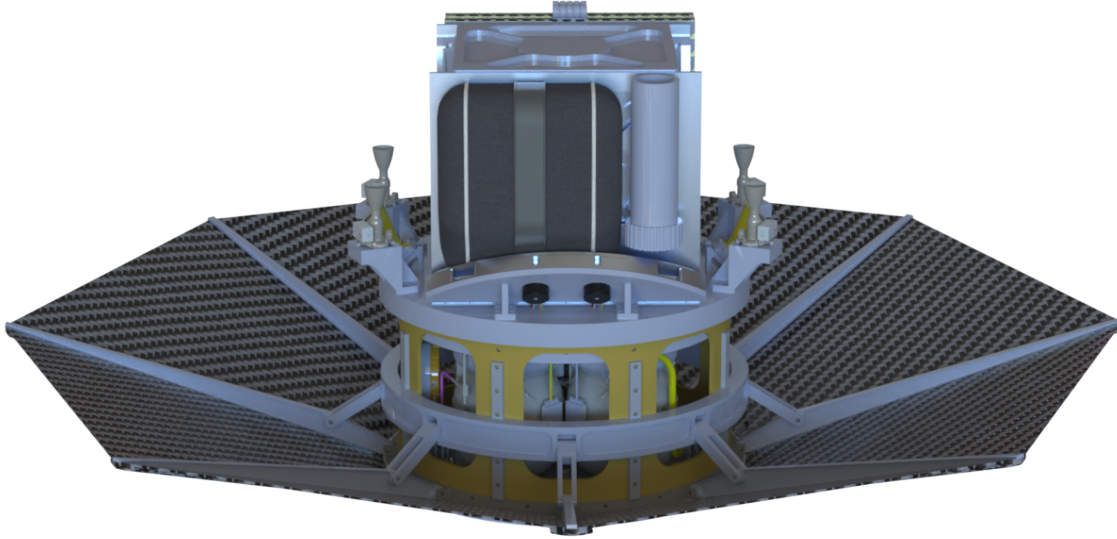


Fig. 1 The Outpost Space Carryall Block 1 Vehicle.

The advantages of deployable heat shields come with their own challenges in analysis, specifically the strong shocks in near rarefied flow conditions and the highly vortical flow behind the heat shield and in the wake. Furthermore, aerothermodynamic analysis is of interest to ensure that the heat pulse is being mitigated sufficiently which requires full thermochemical non-equilibrium physics modeling in the analysis.

II. Analysis Methods

CFD analysis of the Carryall Block 1 is run using two solvers, NASA Fully Unstructured Navier-Stokes 3D (FUN3D) and NASA Data Parallel Line Relaxation algorithm (DPLR) to compare and cross validate. The full trajectory is generated using FUN3D while solver-to-solver comparisons as well as mesh studies included DPLR.

A. Trajectory Development

The Optimal Trajectory Implicit Simulation (OTIS) tool from NASA Glenn Research Center was first used along with notional aerodynamic coefficients from ADEPT [Dutta, 2002] and Carryall Block 1 mission parameters to generate a reentry trajectory. From this trajectory, discrete CFD simulation points were chosen. Extra points were chosen around the peak heating point in the trajectory, peak dynamic pressure, and low supersonic regime. The regime between peak heating and peak dynamic pressure was more sparsely simulated as due to Mach number independence principles, the aerodynamic coefficients varied less. After an initial aerodynamic database was completed, the OTIS trajectory was refined using the Carryall specific aerodynamic coefficients and new discrete analysis conditions were chosen.

B. NASA FUN3D

NASA FUN3D is an unstructured finite-volume CFD flow solver developed at NASA Langley Research Center [Anderson, 2025]. The solver is heavily used at NASA and across the US aerospace community for flows ranging from subsonic incompressible speeds to hypersonic speeds involving thermochemical nonequilibrium. The governing equations in FUN3D for thermochemical nonequilibrium flows are described in [Gnoffo, 1989].

A median-dual finite-volume approach is used for spatial discretization on general unstructured meshes containing tetrahedra, pyramids, prisms, and hexahedra. A dual-mesh is generated by bisecting every edge and connecting them to cell centroids to generate polyhedral control volumes. Inviscid fluxes are computed at each dual-face using an approximate Riemann solver, HLLE++ [Nastac, 2022] in this work, with second-order accuracy obtained using an unstructured Monotonic Upstream-centered Scheme for Conservation Laws (UMUSCL) reconstruction with unweighted least-squares gradients. Viscous fluxes are computed using a Galerkin-based approach. The equations are integrated in time implicitly.

The FUN3D simulations in this work are carried out on AMD and NVIDIA GPUs using the GPU-optimized approach described in [Nastac, 2021] and [Nastac, 2023]. The software is written in a thin abstraction over CUDA C++ which enables primarily single-source software capable of efficiently running on NVIDIA, AMD, and Intel GPUs as well as multicore CPUs. Performance is commensurate with memory bandwidth, which enables modern GPU nodes consisting of a few GPUs to be equivalent to thousands of CPU cores of performance, substantially reducing power and space requirements versus CPU-based systems.

The software is routinely being used to run hypersonic simulations of complex geometries using the hybrid CAD-2-Solution (C2S) approach described in [Nastac, 2025]. With this approach, the user supplies only a CAD file and CFD inputs, and a hybrid mixed-element adapted mesh is automatically generated based on a specified target mesh point count. This paper primarily utilizes pre-generated fixed surface and boundary-layer meshes to reduce computational cost, with off-body tetrahedral adaptation as noted in the reference.

C. NASA DPLR

The NASA DPLR CFD flow solver [Wright, 2009] is a parallel, structured, multi-block finite volume software that solves the reacting Navier-Stokes flow equations including finite-rate chemical reactions and, if needed, thermal nonequilibrium. The DPLR solver has been one of the workhorse CFD solvers used by NASA, academic institutions, and industrial partners over the past 20 years and is well suited to generating flow solutions over spacecraft entering a planetary atmosphere. The Euler fluxes are evaluated using a low-dissipation form of Steger-Warming flux vector splitting with third-order spatial accuracy achieved with monotonic upstream schemes for conservation law (MUSCL) extrapolation [Wright, 2006]. The solver offers a wide variety of gas chemistry and turbulence models that can be applied to a given planetary entry simulation. DPLR is a structured solver, so the meshes consist of rectangular or hexahedral elements. In the present paper, the DPLR solver will be used to provide solver-to-solver comparisons of surface heating rates, pressures, and aerodynamic coefficients against FUN3D values.

III. Solver and Mesh Comparisons

A comparison of structured and unstructured meshes and solvers is carried out. A hypersonic (Mach 24~) condition is considered. The wall is set to a fixed cold temperature with fully-catalytic conditions to maximize predicted heat transfer for conservative predictions. A 5-species air Park gas model is employed. The flow is modeled as laminar due to low enough Reynolds number.

The analysis geometry in this section assumes a rigid heatshield with curved trailing edges. The satellite bus was simplified to a cylindrical geometry of representative width and length. Structural members such as struts, and other features like thrusters and antennas were removed. The geometry also has filleted edges at the aft face of the bus.

DPLR is first run with mesh adaptation to shock-fit the structured mesh. The surface forebody mesh for the DPLR CFD simulations is shown in **Figure 2** on the left. Since DPLR is a structured Navier-Stokes solver, the volume mesh elements are hexahedrons, and the surface mesh elements are rectangular. Symmetry planes are employed accordingly for the DPLR simulations; one symmetry plane for angle of attack cases and two orthogonal symmetry planes for ballistic cases. The outer boundaries of the initial meshes are far enough away from the body to enclose the

bow shock and wake structure. As the DPLR CFD solution progresses, the outer boundary is tailored to match the shape of the bow shock. The near wall cell spacing is also adjusted to reach a near wall cell Reynolds number of 2. Mesh tailoring is a key element to obtaining accurate surface heating predictions.

Both a one and two-temperature model are run at this condition with both solvers to examine differences – peak convective heat transfer differences between the models are less than 0.3% for DPLR which is also similarly observed in FUN3D; the remaining comparisons utilize a one-temperature model. The forebody blocks of the final mesh are extracted and run with FUN3D. The full body structured surface mesh is extracted, triangulated, and a thin prismatic boundary layer of 35 layers (1.2 growth rate with first point off the wall spacing identical) is extruded for the unstructured mesh approach. Future work will consider the full hybrid C2S approach, where the surface mesh is automatically generated directly from the CAD. **Figure 2** plots the adapted structured forebody mesh and an adapted unstructured mesh using the same surface but triangulated. The full (360-degree domain) structured mesh is approximately 14 million points. The extracted forebody structured mesh is approximately 7 million points. The final unstructured mesh is approximately 13 million points.

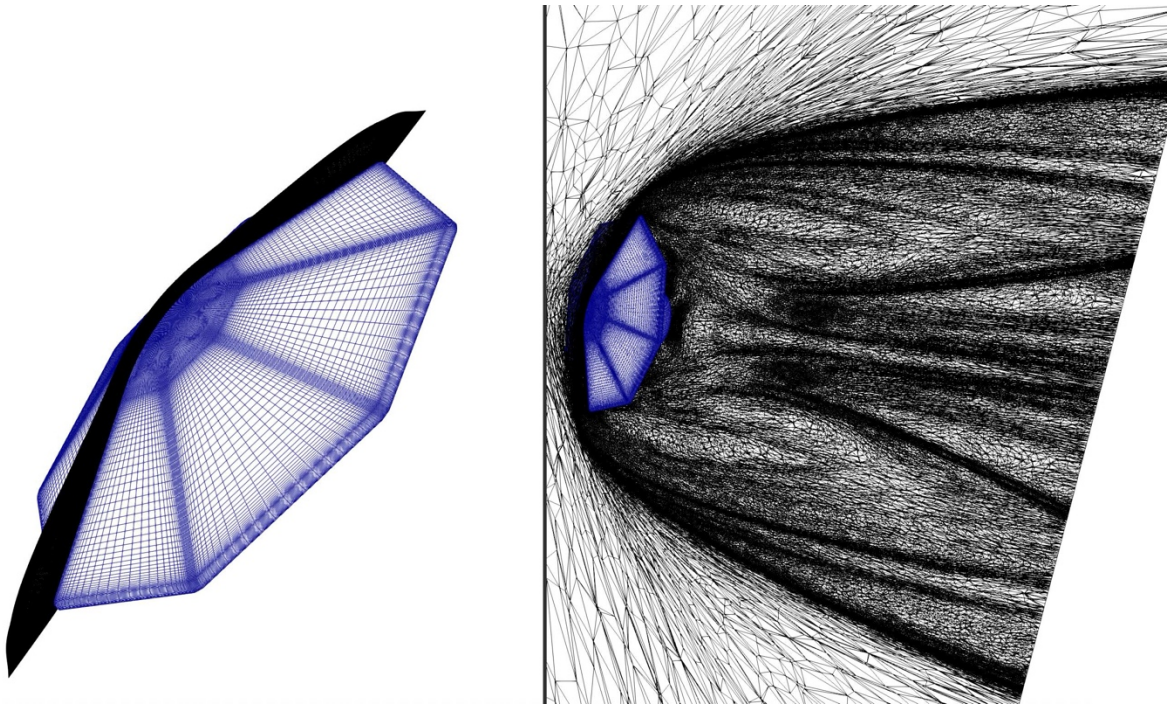


Fig. 2 Left: Adapted Structured Forebody Mesh. Right: Adapted Unstructured Mesh.

FUN3D is run with adaptive CFL control and automated solver termination when integrated convective heat transfer changes by less than 0.01% in 100 nominal steps. Twenty CFD/refinement cycles are carried out. The unstructured adapted simulations are run on AMD MI210, NVIDIA A100, or NVIDIA GH200 GPUs with total run times of about a few hours. The structured mesh FUN3D simulation take less than an hour as no adaptation needs to be performed.

Stagnation line pressure and temperature profiles are plotted in **Figure 3**. Stagnation pressures differ by less than 0.1% between the structured results; the unstructured result is within 0.5%. Peak temperature differences are less than 0.5% between structured mesh results; the unstructured mesh result is about 2.5% lower. For this scenario, the unstructured mesh resolution is comparable at the shock (tetrahedra will be more dissipative than aligned hexahedra, so ideally there are more tetrahedra); further adaptation will tend toward the structured mesh result which is well resolved and adapted.

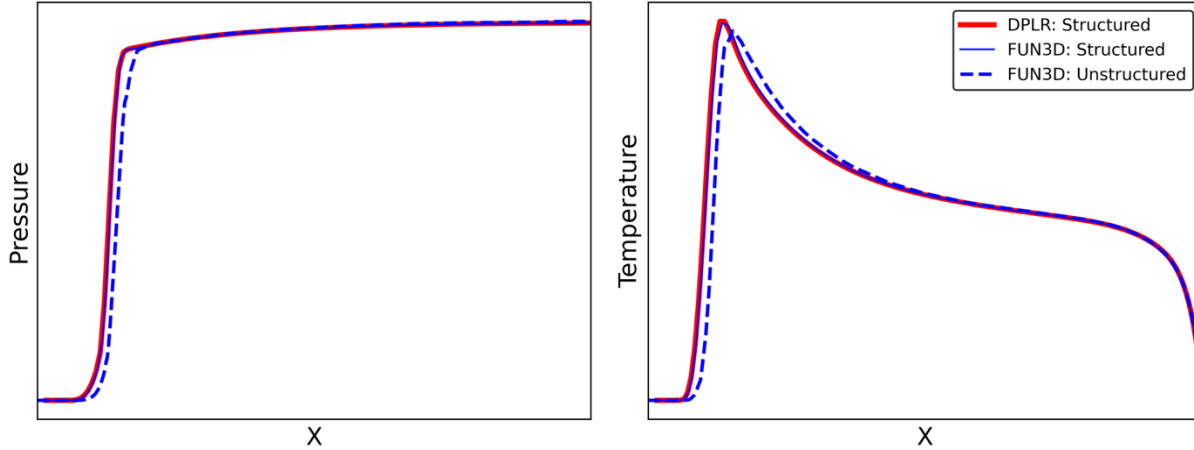


Fig. 3 Stagnation line pressure and temperature. Vehicle stagnation point is on the right.

Surface pressures, heat transfer, and shear stress are plotted in **Figures 4-6**. **Figure 4** plots contours of the surface variables on the heat shield. **Figure 5** plots quantities through the y-axis whereas **Figure 6** plots through a 22.5-degree rotation about the x-axis. DPLR and FUN3D structured mesh results plot only the forebody (although DPLR is also run with a wake mesh), whereas the FUN3D unstructured mesh result plots the aft as well. Surface pressures in all scenarios are in excellent agreement. Heat transfer predictions are also in good agreement overall; only the stagnation region is not entirely smooth for the unstructured mesh result, but the expected magnitudes are similar. Further mesh adaptation should improve these results further; it is generally recommended to not include the wake if unnecessary to reduce mesh point and focus adaptation on the forebody shock. Also, as mentioned previously, the shock resolution for the unstructured mesh is comparable to the structured mesh when it should be slightly finer as unaligned tetrahedra are more dissipative than aligned hexahedra. The shock must be well-resolved for smooth heat transfer and shear prediction. The unstructured mesh results predict slightly larger heat transfer and shear at the nose transition point for the y-axis results. Overall, all three results are comparable from an engineering accuracy point of view.

FUN3D centerline y-slice contours of the mesh, temperature, and Mach number are plotted in **Figure 7**. Results between the structured and unstructured meshes are nearly identical. Contours of the wake mesh, temperature, and Mach number are plotted in **Figure 8**. The wake features are well-captured and smoothly predicted with the unstructured mesh adaptation.

In conclusion, both the unstructured and structured mesh approaches adequately predict aerodynamic and aerothermodynamic quantities of interest. The FUN3D and DPLR results on the same adapted structured mesh agree within a % for surface pressure, heat transfer, shear stress as well as for quantities in the flow field including pressure and temperature. The adapted unstructured mesh simulation predicts surface pressure within a % of the structured mesh simulations but has slightly more variation for heat transfer and shear stress. The shock at the stagnation point is not as well captured in the unstructured mesh versus the shock fit structured mesh – further adaptation will tend towards the structured mesh results and smooth the stagnation heat transfer profiles as shown in previously published results. Overall, the adapted unstructured mesh approach with the current mesh point count is suitable for force and moment prediction as well as overall convective heat transfer and shear stress prediction (e.g., in terms of peak values). Further unstructured mesh adaptation should be performed to achieve mesh-converged heat transfer and shear stress to be more in line with the adapted structured meshes.

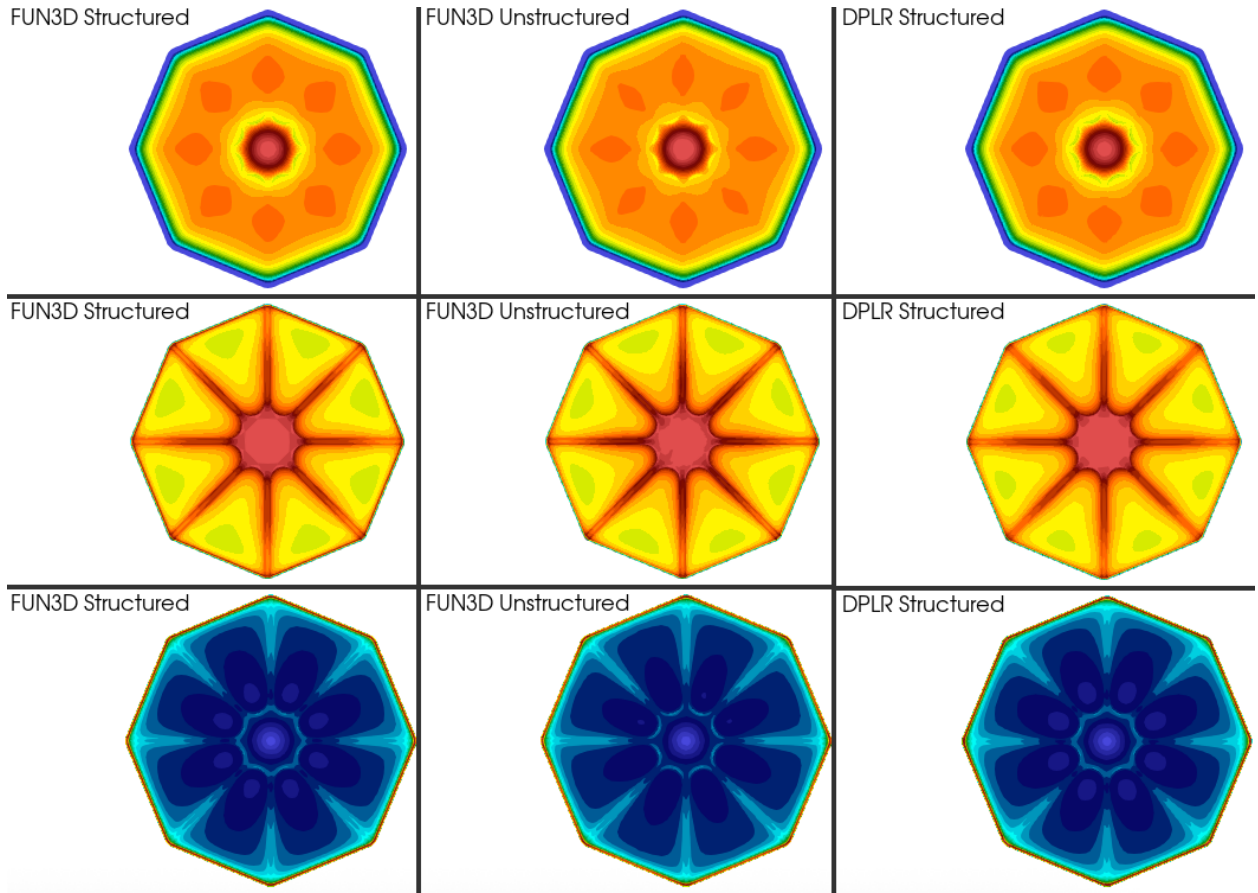


Fig. 4 Top: Surface Pressure. Middle: Surface Heating. Bottom: Surface Shear Stress.

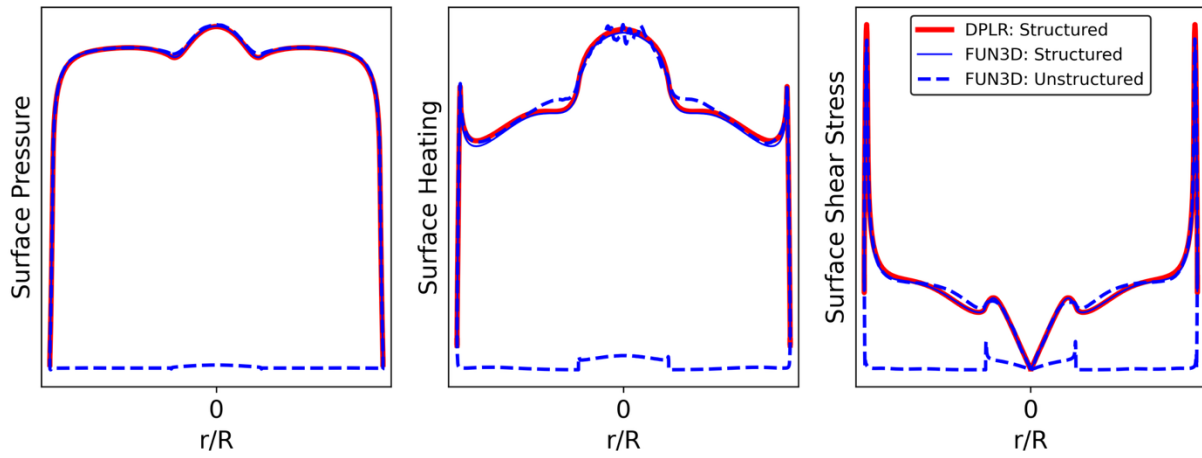


Fig. 5 Centerline surface pressure, heat transfer, and shear stress along the y-axis.

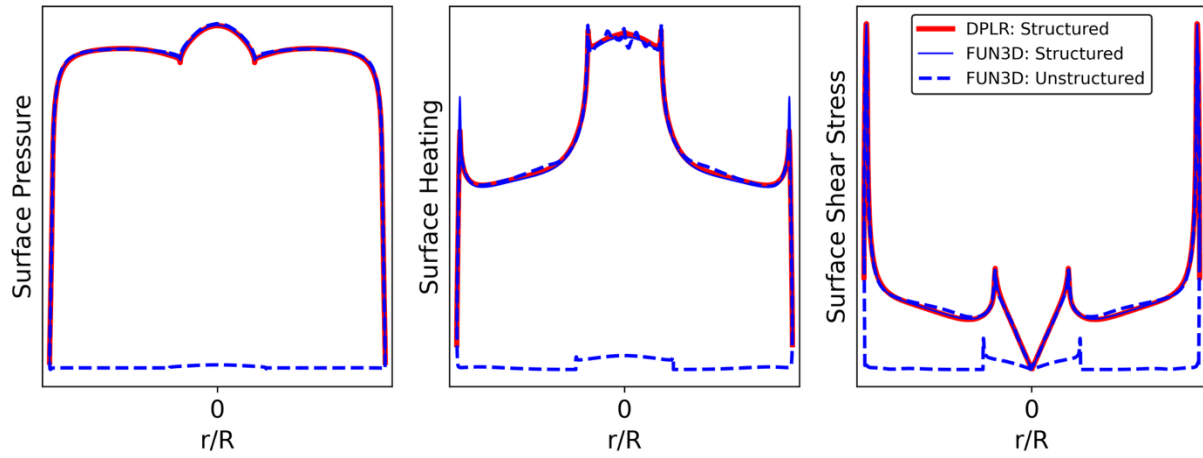


Fig. 6 Centerline surface pressure, heat transfer, and shear stress along a 22.5-degree rotation about the x-axis.

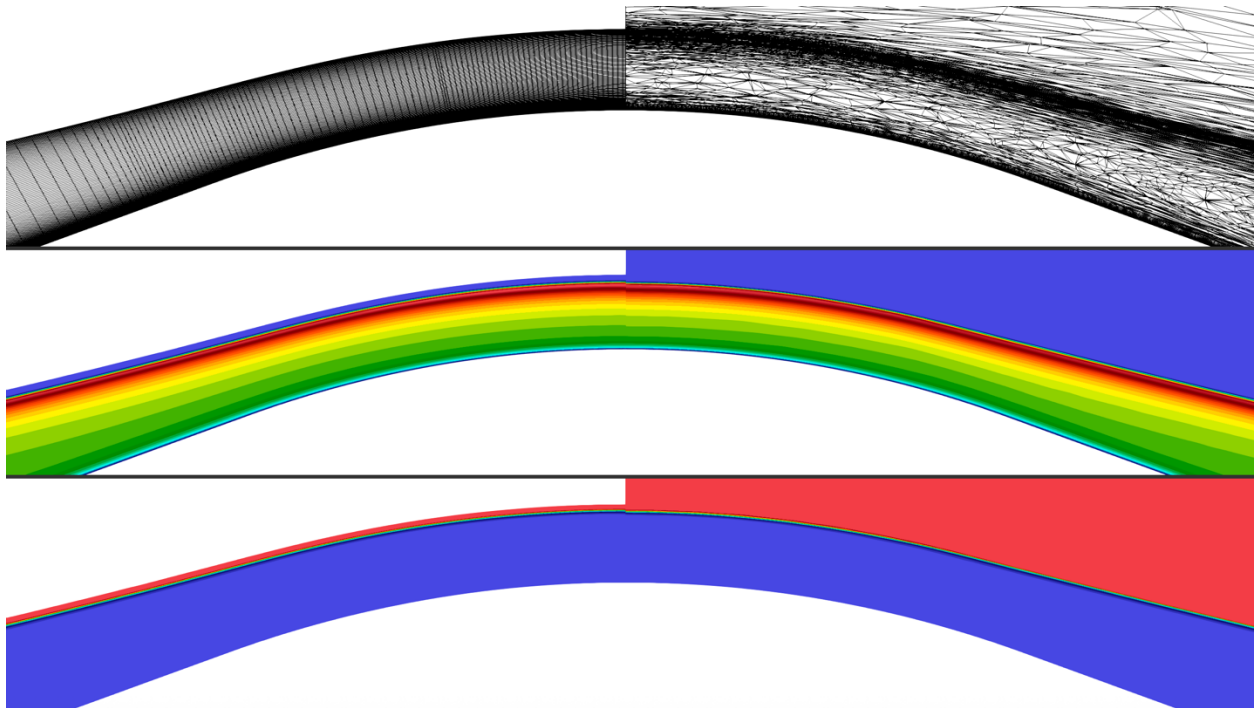


Fig. 7 Left: Adapted Structured Mesh. Right: Adapted Unstructured Mesh. Top: Mesh. Middle: Temperature. Bottom: Mach number.

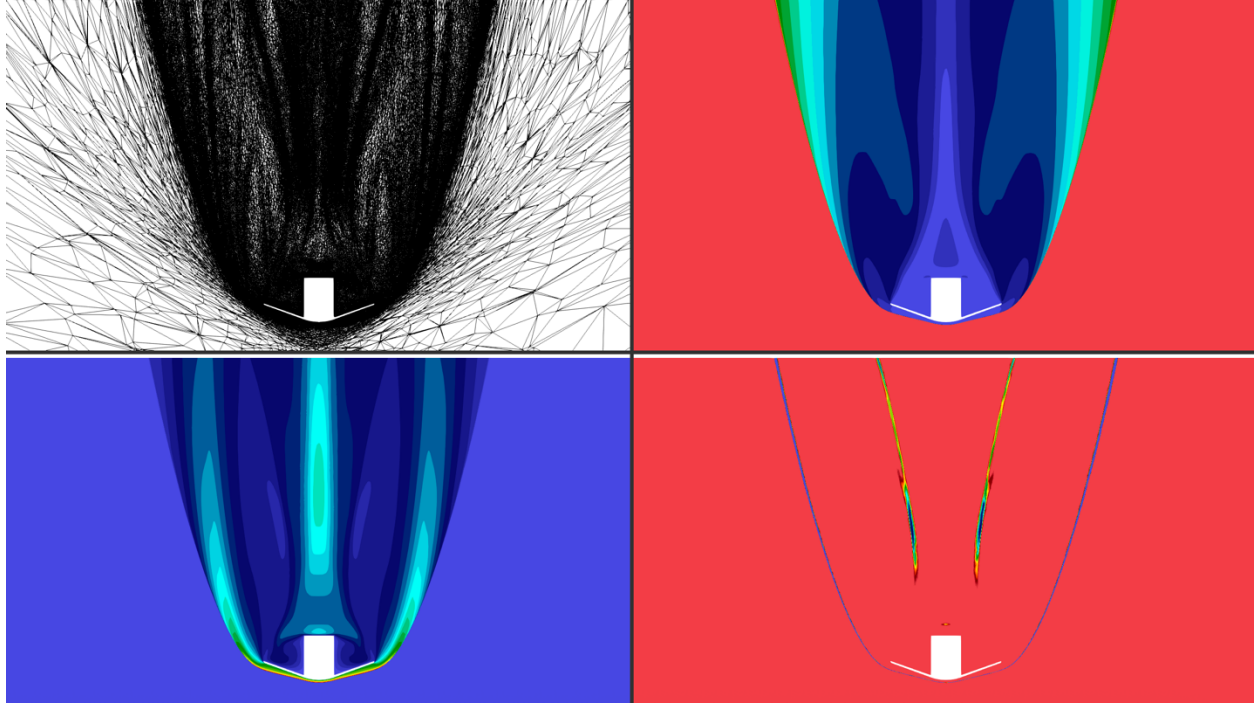


Fig. 8 Centerline y-slice. Top Left: Mesh. Top Right: Mach Number. Bottom Left: Temperature. Bottom Right: Shock switch (blue = shock).

IV. Aerodynamic Database

The Carryall Block 1 database consisted of nearly 140 cases sweeping five angles of attack and 23 Mach numbers. The nondimensionalized altitude, Mach number, wall temperature, and dynamic pressure through the trajectory are shown in **Figure 9**.

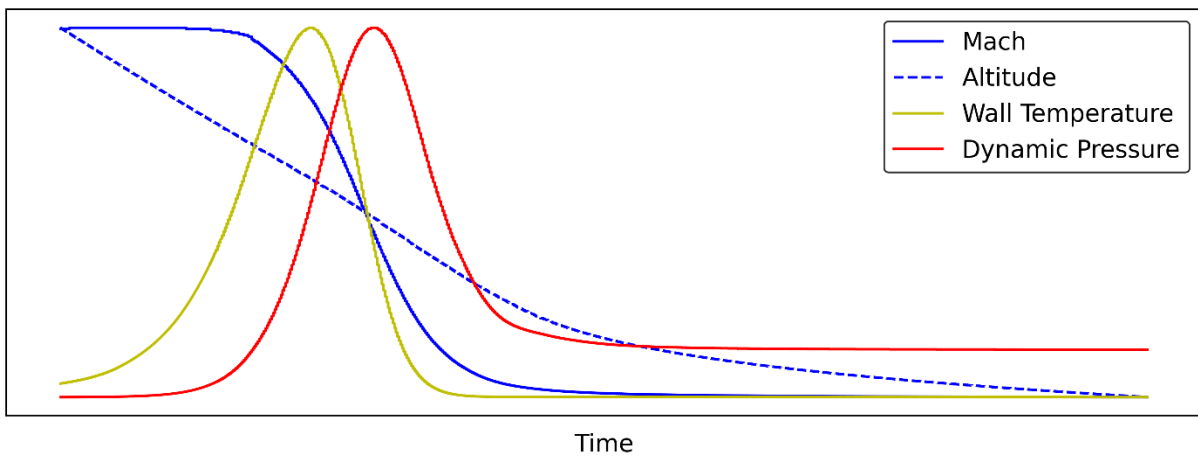


Fig. 9 Nondimensionalized Mach Number, Altitude, Dynamic Pressure, and Wall Temperature throughout the Carryall Block 1 Trajectory as Predicted by OTIS.

For the aerodynamic database, the initial unrefined mesh is roughly 4M points with surface spacing varying from 10mm to 20mm and volume mesh spacing increasing to 5M in the farfield. Twenty boundary prismatic layers are extruded using Pointwise T-Rex with a growth rate of 1.2 starting from 1.5E-3 mm initial wall spacing. The near wall spacing was determined by running a study of six cases with varying near wall spacing. Several of the smallest values resulted in the same peak heating and so the largest of these spacings was picked. This results in a first cell Reynolds number of about 2. DPLR cases confirmed both the spacing and heat flux value. Mesh refinement is run 5 times with the solution being run to iterative convergence between each mesh refinement. Final adapted mesh point counts are roughly 7-8M.

The primary goal of the aerodynamic database is to develop aerodynamic coefficients for GNC software. As aerodynamic coefficients are generally accurate with less mesh resolution and require fewer iterations to converge than heating values, a smaller mesh was run to reduce computational cost. With that in mind, heating values are within 5% of the more refined meshes discussed in Section III.

A. Results

The aerodynamic database showed similar shape and values to the aerodynamic database used in LOFTID. While the LOFTID vehicle was dissimilar in many ways, it is one of the only low beta decelerators that has reentered from low Earth orbit. Comparison to its reconstructed aerodynamic database, especially in the hypersonic regime, has been a valuable method of validation. A maximum drag coefficient is observed at the beginning of the trajectory and decreases gradually through the hypersonic speeds by about 30%. There is a slight bump near peak dynamic pressure which is potentially due to a pause in density in the atmospheric model used to develop the trajectory and conditions.

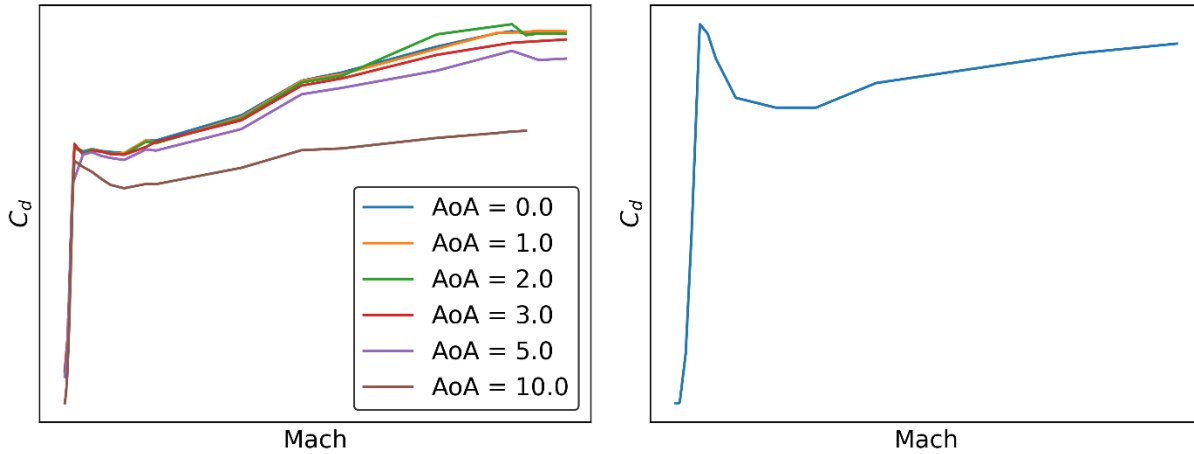


Fig. 10 Drag Coefficient vs Mach of Carryall Block 1 (left) and LOFTID (right, reproduced from [Hollis, 2025]) at 0 degrees angle of attack.

A rise in drag coefficient is observed in the low supersonic range peaking just above supersonic and then falling rapidly through high subsonic as shown in **Figure 10**. This is consistent with transonic behavior, especially in blunt bodies. The peak drag coefficient for the Carryall vehicle is significantly lower than that for LOFTID. Drag coefficients fall with increasing angle of attack and lift coefficients rise with increasing angle of attack as expected. Lift coefficients are shown in **Figure 11**.

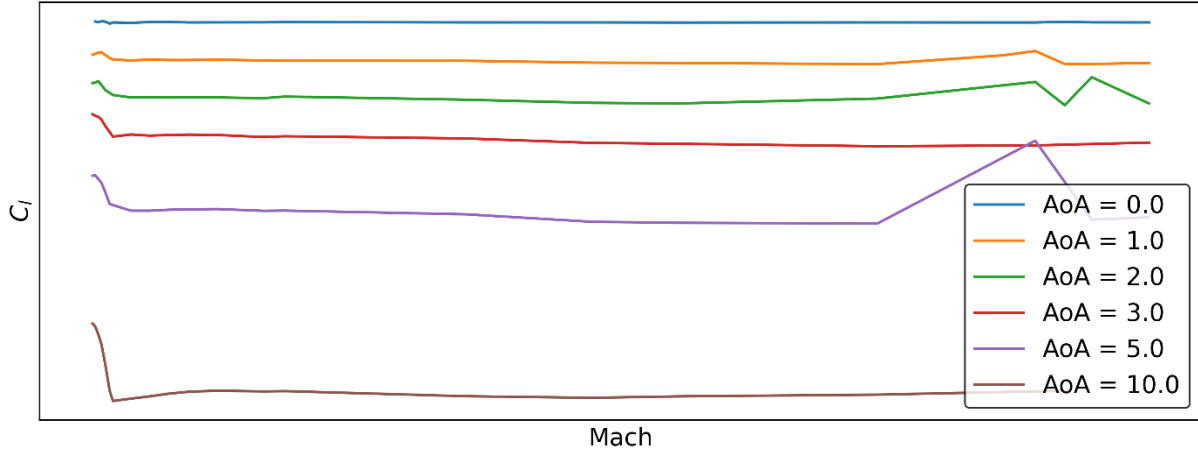


Fig. 11 Lift Coefficient vs Mach of Carryall Block 1.

Cases in this lower transonic region as well as the four cases at the beginning of the trajectory showed the most numerical instability and exhibited scalloping in the shock front downstream of the vehicle. In some cases, this did not affect surface values or aero coefficients and in others it did. It is likely that increasing the mesh complexity during the refine cycles as well as optimizing iterations through each cycle would have solved these issues.

B. DPLR Validation

Aerodynamic coefficients were also validated against DPLR cases. The drag coefficient errors at angles of attack of 0, 1, and 5 degrees for a specific condition are shown below in **Table 1**. Errors are less than 1% for all angles of attack.

Angle of Attack [°]	Error (%)
0	0.17%
1	0.04%
5	0.70%

Table 1 Errors in Drag Coefficient at Angles of Attack of 0, 1, and 5 degrees between DPLR and FUN3D.

V. Fluid Structure Interaction

To predict the deflected state of the deployable heat shield during reentry, static fluid-structure interaction (FSI) analysis is performed by coupling the nonlinear finite-element solver LS-Dyna [Ansys Inc., 2025] and FUN3D's mesh movement capabilities [Biedron, 2009]. The finite-element model contains the spars as well as the carbon-based fabric under high tension in the deployed configuration. Due to the nonlinearity in the structure required to model the fabric in the aeroshell, FUN3D's modal structural solver used in other high-speed applications [Kessler, 2021] is not applicable for this static FSI analysis. Instead, the loose coupling process is based on file I/O. Surface pressure distributions written from a steady FUN3D solution are mapped to the structural finite-element mesh. LS-Dyna then computes deflections which are mapped back to the FUN3D surface mesh. FUN3D reads the new surface mesh and deforms the volume mesh using a linear-elasticity analogy. This coupling loop is repeated until the deflections of the structure and aerodynamic quantities of interest converge. The base case for the nonlinear solver must be created and validated manually, along with translation scripts, which is ongoing work.

VI. Conclusions and Future Work

Analysis of the Carryall Block 1 heatshield was conducted using both FUN3D and DPLR. FUN3D's mesh adaptation and HLLC++ Riemann solver were compared to DPLR solutions and FUN3D GPU acceleration was used to develop an aerodynamic database which was also compared to DPLR solutions at certain conditions. Overall, differences between the two solvers are less than 1% and values for heating, surface pressure, and aerodynamic coefficients are well within expected norms.

As vehicle development progresses, further aerodynamic databases and analyses will be conducted. Increasing mesh density through more adaptation cycles and improving adaptation through increased iterations will likely be used to improve database case stability. Also, a hybrid C2S approach will be implemented in future solutions rather than a fixed surface grid and boundary layer.

Once deflected shapes are obtained through static FSI analysis, the structure will be linearized about the nonlinear equilibrium condition to compute mode shapes. Flutter analysis will then be performed with FUN3D's internal modal solver to help determine if the edges of the flexible TPS are at risk of damage or damaging the structure through flutter.

The first reentry of a Carryall Block 1 is expected in late 2027 and will be instrumented to validate the aerodynamic coefficients as well as the surface heating rates and pressures.

References

- Anderson, W.K., et. Al. FUN3D Manual 14.2, NASA TM 20250002308, 2025.
- Ansys, Inc. *Ansys LS-DYNA User's Guide*, Release 2025 R1. Canonsburg, PA: ANSYS, Inc., 2025.
- Biedron R.T., Thomas, J.L., "Recent Enhancements to the FUN3D Flow Solver for Moving-Mesh Applications" AIAA Paper 2009-1360, 2009. <https://doi.org/10.2514/6.2009-1360>
- Coblisch, J.J., Smith, M.S., Hand, T., Candler, G.V., and Nompelis, I., "Double-Cone Experiment and Numerical Analysis at AEDC Hypervelocity Wind Tunnel No. 9," AIAA Paper 2005-0902, 2005, <https://doi.org/10.2514/6.2005-902>
- Dutta, S., Karlgaard, C., Korzun, A., Green, J., Tynis, J., Williams, J., Yount, B., Cassel, A., Wercinski, P. "Adaptable Deployable Entry and Placement Technology Sounding Rocket One Modeling and Reconstruction", *Journal of Spacecraft and Rockets*, 2022, 59(1), 236-259.
- Gnoffo, P.A., Gupta, R. N., and Shinn, J.L. "Conservation Equations and Physical Models for Hypersonic Air Flows in Thermal and Chemical Nonequilibrium" NASA TP-2867, 1989.
- Hollis, B., Wise, A., Liechty, D., Korzun, A., Thompson, K., Rodrigues, N., Rieken, E. "Aerothermodynamic Analyses for the LOFTID Technology Demonstration Mission," *Journal of Spacecraft and Rockets*, 2026, 63:1, 113-133, <https://doi.org/10.2514/1.A36009>
- Kessler, D., Hess, A., Obenschain, K., Eder, D., Koniges, A., Roe, K. McDaniel, D., Bond, R., Nielsen, E., Walden, A. Nastac, G., Jacobson, K., Knutson, A., Candler, G., Johnson, H., "Performance of Coupled Physics Solvers for Multidisciplinary Hypersonic Flow Simulations on Several Classes of Computer Architectures", AIAA Paper 2022-0973, 2022, <https://doi.org/10.2514/6.2022-0973>
- Nastac, G., Forrest, M., Schneider, D., et Al., "Toward Adaptive Mixed-Element Unstructured Grids for Simulations of Viscous Flows", AIAA Paper 2025-0642, 2025, <https://doi.org/10.2514/6.2025-0642>

Nastac, G., Nielsen E., and Tramel R., “Improved Heat Transfer Prediction for High-Speed Flows over Blunt Bodies using Adaptive Mixed-Element Unstructured Grids,” AIAA Paper 2022-0111, 2022, <https://doi.org/10.2514/6.2022-0111>

Nastac, G., Walden, A., et Al., “A Multi-Architecture Approach for Implicit Computational Fluid Dynamics on Unstructured Grids,” AIAA Paper 2023-1226, 2023, <https://doi.org/10.2514/6.2023-1226>

Nastac, G., Walden, A., Nielsen, E., and Frendi, A., “Implicit Thermochemical Nonequilibrium Flow Simulations on Unstructured Grids using GPUs,” AIAA Paper 2021-0159, 2021, <https://doi.org/10.2514/6.2021-0159>

Wright, M. W., White, T., and Mangini, N., "Data Parallel Line Relaxation (DPLR) Code User Manual Acadia – Version 4.01.1," NASA/TM-2009-215388, October 2009.

Wright, M.J., Milos, F.S., and Tran, P., “Afterbody Aeroheating Flight Data for Planetary Probe Thermal Protection System Design,” *Journal of Spacecraft and Rockets*, Vol. 43, No. 5, May 2006, pp. 929-943, <https://doi.org/10.2514/1.17703>

Wright, M.J., Olejniczak, J., Brown, J.L., Hornung, H.G, and Edquist, K.T., “Computational Modeling of T5 Laminar and Turbulent Heating Data on Blunt Cones, Part 2: Mars Applications,” AIAA Paper 2005-0177, January 2005, <https://doi.org/10.2514/6.2005-177>

Wright, M.J., Prabhu, D.K., and Martinez, E.R., “Analysis of Apollo Command Module Afterbody Heating Part I: AS-202,” *Journal of Thermophysics and Heat Transfer*, vol. 20, no. 1, 2006, pp. 16-30, <https://doi.org/10.2514/1.15873>