Computational Challenges in Hypersonic Flow Simulations

Seokkwan Yoon*

NASA Ames Research Center, Moffett Field, CA 94035

Peter A. Gnoffo†, Jeffery A. White‡, and James L. Thomas§

NASA Langley Research Center, Hampton, VA 23681

Computational challenges in hypersonic flow simulations being addressed by the NASA Fundamental Aeronautics Hypersonics Project are reviewed. The purpose of the project is to conduct foundational research for capsule-based entry vehicles as well as air-breathing launch systems enabling human exploration of the Moon, Mars, and beyond. The challenges include accurate simulation of turbulence due to roughness-induced transition, improved understanding of turbulence physics and transition processes, reliable hybrid RANS-LES methods, innovative radiation-chemistry-flow coupling methods, and automatic hybrid continuum-rarefied flow simulation methods. The robust and accurate prediction of aerodynamic heating is a cornerstone enabling requirement. The obstacles to extending previous research on structured grids to unstructured grids, with the ultimate goal of solution-adaptive methodology, are highlighted.

I. Introduction

Recent interest in human exploration of the Moon, Mars, and beyond has revitalized hypersonic flow research. NASA is conducting foundational research for capsule-based entry vehicles as well as air-breathing launch systems through the Hypersonics Project. This project is one of four Fundamental Aeronautics projects, organized by speed regime, created by the Aeronautics Research Mission Directorate. The end of the Apollo Program in the early 1970s brought a decline in research investment in high-speed reentry technology because human spaceflight in that period only needed a transportation system for access to Low Earth Orbit (LEO). Nevertheless, some research progress has been made in supporting unmanned planetary probes and manned reentry vehicles. The unsuccessful National Aerospace Plane Program in the late 1980s spurred fundamental research in air-breathing scramjet propulsion.

Modern design tools for engineering level hypersonic studies rely on fast low-fidelity methods to carry out parametric and configuration-optimization studies. The fidelity of the aerodynamics and aerothermodynamics databases is anchored by using a limited number of Reynolds-Averaged Navier-Stokes (RANS) solutions that attempt to capture the complex physical processes experienced by a vehicle flying at hypersonic speeds. Physics-based models to simulate the processes of turbulence, transition and other complex gas-surface interactions are needed because Direct Numerical Simulation (DNS) methods are not yet feasible for configuration design with today’s computers.

Advanced simulation tools will enable increased spatial and time accuracy, increased geometric complexity, grid adaptation, increased physical-process complexity, uncertainty quantification, and error control. To achieve these goals modern software tools and practices will be applied to implement new algorithms on massively parallel computer architectures. The complex physical phenomena and the wide range of spatial and time scales present in flows containing gas-surface interactions, entropy and shear layers, shock waves, real-gas effects, and turbulence and transition make the development of efficient and accurate numerical simulation methods extremely challenging. For enhanced spatial accuracy, high-order algorithms for both structured and unstructured grids, such as low-

---

* Lead, Hypersonic Aero Team, NAS Applications Branch, AIAA Senior Member
† Senior Research Engineer, Aerothermodynamics Branch, AIAA Fellow
‡ Research Engineer, Computational Aerosciences Branch, AIAA Senior Member
§ Research Engineer, Computational Aerosciences Branch, AIAA Fellow

American Institute of Aeronautics and Astronautics

This material is declared a work of the U.S. Government and is not subject to copyright protection in the United States.
dissipation\textsuperscript{30}, discontinuous Galerkin\textsuperscript{31-33}, and spectral-difference\textsuperscript{34-36} algorithms, need to be developed that operate robustly in the hypersonic regime and that are tested on relevant model problems in order to evaluate their relative strengths and weaknesses.

The accurate prediction of the onset of boundary layer transition is critical for the evaluation of heating and the design of thermal protection systems. Typical work needed in this important area is the development of higher order methods to compute transitional flow by DNS. This work will provide insight for the development of transition models. Uncertainty quantification is needed as a first step towards identifying and managing risk. Sources of uncertainty include model equations, model parameters, geometric representation, flow environment, boundary conditions, and numerical errors. Adjoint methods need to be developed to provide a means to detect and control sources of error, and relate them to engineering outputs of interest such as aerothermodynamic heating and drag. Furthermore, techniques for grid adaptation to capture essential flow features need to be developed using adjoint methods as an improvement to heuristic methods. More general Monte-Carlo methods also need to be developed to handle sources of uncertainty that are not amenable to analytic approaches. Finally, a more comprehensive toolset needs to be developed to capture the interactions of multiple physical effects, such as radiative heat transfer, transition and turbulence, chemical reactions, non-equilibrium thermodynamics, gas-surface interactions, and non-continuum flows.

This paper discusses the high-priority computational challenges in Entry, Descent, Landing (EDL) technology. The robust and accurate prediction of aerodynamic heating is a cornerstone enabling requirement. The obstacles in extending previous research on structured grids to unstructured grids with the ultimate goal of solution-adaptive methodology are highlighted.

II. Challenges in EDL Technology

A. Turbulence Due to Roughness-Induced Transition

The design of hypersonic vehicles depends critically on the accurate prediction of laminar-turbulent transition and the associated increases of skin-friction drag and wall heat transfer. Such accuracy requires that all relevant physical phenomena be accounted for, in order to improve the predictive and uncertainty quantification techniques that NASA currently relies upon. However, the fundamental physics of transition and turbulence in hypersonic boundary layers are poorly understood. A case in point is the recent discovery of the possibility of transition to turbulence due to transient growth in subsonic and low-Mach number supersonic boundary layers. This transition scenario does not require the classical normal-mode instability mechanism, and the popular $e^N$-method for transition prediction, for example, will not work. To what extent this non-normal mode mechanism plays a role in transition in hypersonic boundary layers is not known.

The same transient growth mechanism also produces so-called optimal disturbances in fully developed turbulent boundary layers. These optimal disturbances are closely related to the self-sustaining mechanism of near-wall turbulence structures, which are known to be responsible for the large skin-friction drag and surface heat transfer in incompressible turbulent boundary layers. There is some evidence that the same turbulence structures are prevalent in low-Mach number supersonic boundary layers, but there also exists contradictory research regarding the effect of Mach number on turbulence structures. Martin\textsuperscript{7} observed that the streamwise extent of near-wall streaky structures decreased with increased Mach number, whereas Coleman et al.\textsuperscript{8} reported that the streak coherence was increased in a supersonic channel flow. A better understanding of the effects of high Mach number on these turbulence structures and of the extent these structures continue to play a critical role in hypersonic boundary layers is an essential design issue for hypersonic vehicles.

Hypersonic vehicles contain various types of roughness elements, ranging from a few isolated large protuberances to a large number of distributed small roughness elements. Surface roughness is inherent to thermal protection materials and associated high-temperature seals. Figure 1a shows the gap fillers in the Space Shuttle Orbiter STS-114. The protrusions might have been tall enough to cause transition at Mach 25. Since the elevated heating associated with turbulent flows would occur so early in the trajectory, the thermal protection system was possibly in jeopardy. Planetary entry vehicles are often designed with protrusions and cavities in their heat shields. Heat transfer rates, and surface recession rates for ablative systems, are severely augmented by turbulent flow conditions, potentially leading to a heat shield failure. Figure 1b shows heating and recession augmentation observed downstream of Apollo compression pads. The effect will be larger for the Orion Crew Exploration Vehicle (CEV) since the recession rate of Phenolic Impregnated Carbon Ablator is higher than that of Avcoat, Apollo’s thermal protection material. A Reynolds-Averaged Navier-Stokes (RANS) solution\textsuperscript{9} in Fig. 2 shows significant turbulent heating augmentation near the compression pads on the heat shield of Orion CEV because of a forced transition.
A future human Mars mission may require a very large hypersonic inflatable decelerator (Fig. 3a) to land a large mass onboard an ellipse sled (Fig. 3b) after passing through the thin Martian atmosphere. A Ballistic Range experiment\textsuperscript{10} for a pre-ablated hemispherical graphite nose tip in Fig. 4 shows a rapid increase in surface temperature due to roughness-induced transition. The extent to which turbulence induced by surface roughness will interact with the outer part of a turbulent boundary layer, and hence to what extent the presence of surface roughness affects the overall boundary-layer characteristics is not very well understood even for subsonic boundary layers. Improved understanding of transition processes triggered by such roughness elements and turbulence characteristics in the presence of such roughness elements in hypersonic boundary layers are also critical issues in the design of hypersonic vehicles.
B. Hybrid RANS-LES

Understanding of the complex flow phenomena that occur at hypersonic speeds can best be obtained through a coordinated study involving both computational and theoretical analyses. Relatively few numerical simulations of a turbulent hypersonic boundary layer are available in the literature and most of these studies were conducted at relatively low Mach numbers with perfect gas models. Furthermore, most studies were primarily focused on the effects of compressibility on turbulence statistics, and very few studies aiming at an improved understanding of the fundamental physics of hypersonic boundary layers have been conducted. DNS is uniquely suited for such studies because the complete physics can be simulated with minimum simplifying assumptions. Development of high-order spatial and temporal algorithms for non-equilibrium hypersonic flows is a key element for successful utilization of DNS. Using DNS for transitional and turbulent flows involving a realistic geometric configuration, however, is beyond the capability of practically-available computational resources, and it would be limited to flows involving simple configurations at low Reynolds numbers.

Large-Eddy Simulation (LES) is a more affordable tool for studying complex flow phenomena as well as for the actual design of hypersonic vehicles. Complex unsteady flows due to a window in the afterbody of Orion CEV are shown in Fig. 5 from predictions with a RANS model; there is considerable uncertainty with these predictions because most of the RANS modeling developments have been for steady flows. There are many issues that need to
be addressed before LES can be used reliably in computing hypersonic flows. For example, most, if not all, subgrid-scale models, including those accounting for compressibility effects, have not been validated in hypersonic flows. They must be validated first, and improved if necessary, before they can be used in studying flow physics of hypersonic flows. Considering difficulties in conducting experiments at a hypersonic speed, DNS will play a crucial role in providing data and developing a subgrid-scale model suitable for hypersonic flows. Although LES is much less computationally intensive than DNS, and been used successfully in many applications, it is still computationally too expensive for certain applications.

Hybrid schemes, such as Detached-Eddy Simulation (DES), in which RANS is used in flow regimes where the more affordable RANS approach can be used reliably and LES is used in flow regimes where more accurate flow dynamics are required, lead to more practical approaches to computations of complex turbulent flows. In the foreseeable future, a hybrid scheme is probably the only practical means that can be used routinely to compute transitional and turbulent flows around a complete hypersonic vehicle. Although some progress has been made over the past decade, much more work is required to make this relatively new and promising approach a reliable design tool.

Most current approaches do not adequately treat the interface between RANS and LES. Ideally, the decision whether a particular region is handled by RANS or LES should depend on the local grid resolution and flow conditions. This boundary, whether it is a sharp or buffered interface, is a function of the numerical grid (i.e., once the grid is chosen, the boundary between RANS and LES is chosen and fixed). Also, while there are very few or no explicit turbulence fluctuations in a RANS region, sufficient turbulence fluctuations must enter into the LES region through the interface for a consistent level of turbulence to be maintained. Improper handling of such interfaces often leads to non-converging results as the grid is refined. Sinha and Candler\textsuperscript{11} reported that a grid independent solution was not obtained for a low Reynolds number flow in spite of using a large number of grid points (Fig. 6). Improved handling of the RANS-LES interface is essential before current hybrid RANS-LES methods can be used for design in a reliable manner.

![Figure 5. RANS solution of CEV afterbody flows at Mach 27 with and without a window (McDaniel\textsuperscript{9})](image)

![Figure 6. DES of Fire II wake at Mach 16 shows lack of grid convergence. (Sinha and Candler\textsuperscript{11})](image)
C. Multi-Physics Coupling

The accurate prediction of aerothermal environments is of crucial importance for supporting future NASA missions by reducing uncertainties in design. Simulations in the hypersonic regime require inclusion of many physical effects such as thermal and chemical nonequilibrium including gas-phase chemical kinetics in both N₂/O₂ and CO₂/N₂ atmospheres, radiation generated behind strong shock waves, gas-surface material interactions, and dynamic ablation. During the past couple of decades, Park’s intuition-based two-temperature model has most commonly been used for thermochemical nonequilibrium flow simulations. To improve such simulations, development is underway of a new chemistry model based on ab initio quantum mechanical calculations.

Radiative heat transfer becomes significant relative to convective heat transfer when the size and speed of the entry vehicles become large. Shock-layer radiation has not been a critical issue for the Space Shuttle Orbiter due to its relatively low reentry velocities on the order of 7.5 km/s, nor for high-speed entry probes such as Genesis and Stardust due to their small size. However, the Orion CEV’s large size (the diameter is about 30% larger than Apollo) and high reentry velocities (10.5 km/s for lunar return or 12-14 km/s for Mars return missions) make it imperative to study the phenomenon of shock layer radiation. A comparison of stagnation heat flux for lunar and Mars missions in Fig. 7 shows that the heating for Mars return can be up to five times higher than that for lunar return because the dominant heating mode shifts from convection to radiation. Such high levels imply that the current practice of calculating the radiation in a post-process manner, with no coupling back to the flow, is inadequate. It is imperative to develop methods for calculating the radiative heat transfer throughout the flowfield with sufficient efficiency that fully coupled radiation-hydrodynamics may be used for design purposes. Such calculations are made especially challenging due to the very complex structure of the radiative spectra of high temperature air.

![Figure 7. Comparison of total heat flux (W/cm²) for lunar and Mars return conditions (Kinney and Bowles)](image)

![Figure 8. Uncoupled solution of convective, radiative, and total heating for CEV at Mach 41 Mars return condition (Kinney)](image)
Figure 8 shows convective and radiative heat fluxes for a Mars return condition. The level of uncertainty for radiative heating is so high that the current margin applied to Orion CEV is 200%, compared to 120% for laminar convective heating. Radiation, similarly to chemical reactions, absorbs energy from and deposits energy into the flow. Coupling the radiative transfer equation with the flow solution is required to avoid the usual overprediction of radiative heating in uncoupled simulations. Modeling is required to do this coupling because the computational cost of a fully-resolved solution is too high for coupled simulations. Quantities dependent on the frequency variable cannot be fully resolved, but their average effects can be included through various mathematical approximations, such as opacity binning and macroscopic models, which will be implemented and compared to assess their effectiveness. Similarly, fully-resolved angular dependence of the radiation is also expensive to treat directly but can be handled through various modeling approaches.

For the heating rates encountered during hypersonic reentry, the surface is ablating and the ablation products blowing into the boundary layer induces new interactions that have strong impacts on surface heating rates and integrated heating loads. Therefore, it is important to couple computational methods for radiation, chemistry, ablation and fluid dynamics into a more comprehensive toolset to capture the interactions of multiple physical effects.

Hypersonic flows around reentry vehicles at high altitudes involve a mixture of continuum and rarefied regimes that no single existing code can simulate accurately. Aerocapture enables future missions to Titan and Neptune, and substantially enhances missions to Venus and Mars by decreasing the required propellant for orbit insertion. It is important to assess the effect of non-continuum flows on heat loads for single-pass aerocapture or multi-pass aerobraking. Both continuum and rarefied regions may be involved when using ballutes. Accuracy of continuum based CFD methods is poor in rarefied flow regions while the computational cost of particle-based DSMC methods is unacceptably high in continuum regions. A hybrid code switches between these techniques as local conditions dictate. Figure 9 shows contours of temperature for the Mach 10 flow of nitrogen around a cylinder. The lower half shows the CFD results. The upper half shows both the DSMC results and the hybrid simulation results. The hybrid code was initialized to the CFD results and corrected them to provide new results that provide almost perfect agreement with the much more expensive pure DSMC results.

![Figure 9. Temperature contours (K) of Mach 10 flow of nitrogen over a cylinder: Lower half from Navier-Stokes and upper half from hybrid DSMC/CFD (Schwartzentruber, Scalabrin, and Boyd)](image)

American Institute of Aeronautics and Astronautics
III. Heating Prediction on Unstructured Grids

Regardless of the underlying grid type (structured, unstructured mixed elements, unstructured Cartesian, overset structured or unstructured) a robust scheme must either: (1) be free from any requirement of special topological grid constraints in resolving all flow structures; or (2) be able to automatically adapt specially required grid topologies wherever they are needed in the flow field. In the context of the simulation of hypersonic, blunt body, stagnation region heating, three-dimensional simulations using high aspect-ratio tetrahedra to resolve the boundary layer and the bow shock produce poor heating. 19

Problems with heating can be overcome if one uses semi-structured grid (prisms) across the boundary layer and adapts the grid to the shock. Nompelis et al. 20 show excellent heating results for the cylinder test problem 21 on families of grids where a prismatic grid was adapted to the shock and acceptable heating results when an unbiased (random face orientation), tetrahedral grid was adapted to the shock. Prismatic elements are relatively easy to generate on blunt body geometries and their superior performance with respect to aeroheating predictions have led to their use as standard practice in unstructured simulations. There is no better alternative with today’s algorithms than the use of prismatic elements to capture the boundary layer and bow shock.

If one is doing a simulation without any free shear layers or internal shocks the specialized application of prismatic elements at the body and (possibly) bow shock is perfectly acceptable. If such flow structures exist - as they do in almost all interesting problems - then the accuracy of any algorithm that ignores the special topological grid requirements where the features are harder to enforce must be questioned.

The position advocated in Ref. 19 is that unstructured grids provide the greatest flexibility to adapt to evolving flow structures (both viscous and inviscid) and to complex, deforming bodies in a hypersonic flow simulation without a requirement for significant user intervention. More simply put, unstructured elements provide the greatest opportunity to create a robust aerothermodynamic simulation. A discussion of the simulation needs for hypersonic flow over an inflated aerobrake in the next section provides a context for this position. The problem with the simulation of heating on tetrahedral grids is currently one of the biggest obstacles to realization of such a simulation capability.

A. Spacecraft-Tether-Ballute System Simulation

Large, inflatable ballutes (balloon--parachutes) have been proposed as hypersonic decelerators for planetary aerocapture applications. 22-24 A simulation of a spacecraft -- tether -- ballute system using the unstructured grid, Navier-Stokes flow solver FUN3D is presented in Figs. 10-11. These results, repeated from an earlier work 25, are presented here to provide context for a position advocating use of an adaptive, tetrahedral grid system for hypersonic simulations.

The ballute has a 52 m ring diameter and 13 m cross-sectional diameter. Conditions of the simulation are for a Titan Organics Explorer with velocity equal to 8550 m/s and density equal to 1.9 x 10^{-7} kg/m^3. All surface temperatures are set to a constant value equal to 500 K. The gas model includes molecular nitrogen and atomic nitrogen in thermochemical nonequilibrium. The towing spacecraft is a Pathfinder shape - 70 deg spherically capped cone with a 6 m base diameter. The simulation domain encompasses a 90 degree wedge about the system axis. The spacecraft looks sharp at the nose in the figures because of the view angle combined with the 90 degree domain cutting planes. The simulation assumes symmetric flow with four 0.3 m diameter compressive tethers attached to the toroid so that the leading edge of the tether is tangent to the toroid outer surface. A compressive tether is a flexible cylinder which can be inflated to withstand compressive loads and is used to position the toroid in space prior to entry. Tensile load tethers would likely be encased within the compressive tether. At present, only a continuum simulation is enabled. Flow over the compressive tethers is deep in the transitional flow domain in which the validity of Navier-Stokes analyses is inaccurate. Ideally, a fully coupled continuum-rarefied analysis would be brought to bear on this complex system of disparate length scales.

All of the flow features illustrated in Figs. 10-11 are simulated using only tetrahedral cells. The flow conditions are at a relatively low Reynolds number; consequently the aspect ratio of cells in the boundary layer does not need to be very large. There is a spacecraft bow shock that interacts with the toroid bow shock. The toroid bow shock creates a focused compression in the center of the toroid that drives a reverse flow upstream toward the base of the towing spacecraft. A shock forms further upstream, diverting flow around the core. The compressive tether passes through the spacecraft bow shock and reenters the toroid bow shock. A merged shock layer about the tether itself interacts with these external features. The shocked flow off the tether spills out onto the ballute surface and a local hot spot is formed near the impingement point as seen in Fig. 11. The shear layer around the recirculating core regulates energy exchange between flow processed by the spacecraft bow shock and flow processed behind the spacecraft bow-shock and toroid bow-shock interaction. As Reynolds number increases with deeper descent into the
planetary atmosphere the reverse flow moves further upstream and becomes unsteady. The simulation here assumes a rigid body but a robust simulation must also handle the multi-physics interactions associated with an inflatable structure that undergoes significant deformation.

Obtaining a grid converged simulation on this problem is challenging. Obtaining the solution under constraints that prismatic grids must adapt and align to all of these developing features (shocks and shear layers) on deforming bodies is even more difficult. That is not to say such constraints could not be realized in a robust simulation capability -- only that an algorithm without a requirement for such constraints is more likely to be robust.

Figure 10. Unstructured grid used in simulation of spacecraft-tether-ballute system. Colors correspond to nondimensionalized pressure levels.

Figure 11. Surface heating in W/cm² on ballute in vicinity of tether attachment point
B. Hypersonic Flow over Cylinder -- A Simple Test for Heating

Accurate simulation of stagnation region heating in hypersonic flows is a key requirement for acceptance of any algorithm proposed for use in aerothermodynamic analyses. A structured grid solution generated with LAURA is used both as a benchmark and to generate initial grids for use in FUN3D. The test problem uses \( V_\infty = 5000 \text{ m/s} \), \( \rho_\infty = 0.001 \text{ kg/m}^3 \), and \( T_\infty = 200 \text{ K} \). Sutherland's law for air is used to define transport properties in all perfect-gas cases. As noted below, this simple problem provides insight into the ability of a scheme to cleanly capture the bow shock, smoothly resolve the post-shock stagnation region flow and predict a smoothly varying heating distribution around the stagnation point. These flowfield characteristics are particularly sensitive to the inviscid flux reconstruction algorithm and problems that are not evident in well aligned, structured (hexahedral) grids are exposed in the unstructured (tetrahedral) environment. The goal of these tests is to develop a flux reconstruction algorithm that is applied identically to structured and unstructured grid environments with minimal discrepancies between the simulations and to test grid adaptation algorithms that are required to address any remaining accuracy issues.

Grids

A structured grid, adapted to the shock and boundary layer is tested. It is then converted from hexahedral elements to tetrahedral elements by adding diagonal edges consistently from minimum index to maximum index corners. A comparison of the grids in a plane of nodes perpendicular to the cylinder axis is presented in Fig. 12. The structured grid has 65 nodes from the body to the inflow boundary ahead of the bow shock and 61 nodes from the left to right outflow boundaries. Node placement is identical between the two grids. The placement of additional edges in the unstructured grid is the only difference. The strong biasing of diagonals in the grid is an intentional characteristic to expose algorithm deficiencies that may otherwise be averaged out in the simulation.

A key element of this test problem is the addition of ten spanwise cells, shown in Fig. 13, across the cylinder, providing additional degrees of freedom in the simulation to allow asymmetries to develop. Earlier tests (when the code was referred to as High Energy Flow Solver Synthesis -- HEFSS) have shown that the single spanwise cell grids show good agreement with the structured code results in heating. (Fig. 14a) The strongly biased grids produce a streamwise asymmetry in the shear distribution and overpredict the LAURA results throughout even in the case of a single spanwise cell. (Fig. 14b) The contours of heating and shear should be constant (straight lines) in the spanwise direction. Asymmetries introduced using the biased grid are easily observed in these tests; consequently, the simulation of symmetric results is a measure of solution quality.

Figure 12. Structure grid (LAURA) and biased, unstructured grid (FUN3D) in plane orthogonal to surface
Hexahedra vs. Tetrahedra

A perfect gas case ($\gamma = 5/3$, $MW = 2$, $M_u = 4.25$) is tested on both structured and unstructured grids with identical location of nodes. The structured and unstructured results for heating and shear are compared in Fig. 15. An approximate spanwise variation of $\pm 10\%$ about the mean is observed in peak values of heating and shear in the unstructured case. There is approximately 10% difference in the peak values of shear on the right and left sides of the cylinder for the unstructured grid. The spanwise mean value of the unstructured heating variation appears to be in good agreement with the spanwise constant, structured grid results. The spanwise mean value of the unstructured shear is about 20% to 25% higher than the structured grid result. These results suggest that the development of the velocity profile is more sensitive to the strong bias in the grid than is the development of the enthalpy profile.

Figure 13. Surface mesh on cylinder with ten rows of spanwise cells

Figure 14. Surface heating and shear over cylinder and standard test conditions with 5-species reacting air model and fully catalytic wall
Note that according to the criteria developed by Barth for critical Mach number as a function of $\gamma$ for shock instability\textsuperscript{27} (and confirmed by the structured grid result of the previous section) the present test case does not produce a carbuncle. Nevertheless, the current case shows a spanwise variation in heating which indicates this behavior is not solely a function of the tendency of the approximate Riemann solver to admit a carbuncle.

The large value of $\gamma = 5/3$ and low molecular weight MW=2 produce a relatively large sound speed and moderate supersonic Mach number $M_\infty = 4.25$. Selecting a perfect gas with $\gamma = 7/5$ and MW =28 yields a lower sound speed and higher free stream Mach number $M_\infty = 17.34$. The shock capturing process now requires a shock sensor to prevent catastrophic undershoots in the temperature. It is immediately evident that the spanwise variation in heating and shear shows significant increase in Fig. 16a with $\pm 30\%$ about the mean -- an unacceptable result for aerothermodynamic analyses. The LAURA result in this case is 52 W/cm$^2$ which is in good agreement with the spanwise mean of the heating at the stagnation point. Fig. 16b shows an overlay of the Mach number contours in the front and back planes. It is evident that the captured shock is offset by one cell between the two planes. The surface heating contours in Fig. 16c indicate that a slight drift in the velocity -- probably induced by grid bias -- produces a higher heating toward the front ($y = 0$) plane. The jags in the temperature contours in Fig. 16d indicate a small overshoot in temperature in the post shock region.

Using a metric of spanwise variation of heating, the present hypersonic result is approximately a factor of three worse than the previous supersonic result. Sensitivity to Mach number (through a variation of $\gamma$ and molecular weight) is evident in these results.

Detailed views of the grid (Fig. 17) clearly show that the captured shock is offset between the front and back planes.
Figure 16. Simulation results for $\gamma = 7/5$, pressure ratio switch = (2,3), and no grid adaptation
IV. Summary

High-priority challenges for hypersonic research in EDL technology for future NASA missions have been presented. The challenges are: (1) accurate simulation of turbulence due to roughness-induced transition, (2) improved understanding of turbulence physics and transition processes, (3) reliable hybrid RANS-LES methods, (4) innovative multi-physics coupling methods, and (5) automatic hybrid continuum-rarefied flow simulation methods. A key component of these challenges is reliable and robust numerical prediction that includes error prediction and control. Thus, solution-adaptive methods are needed that are able to begin from relatively coarse grids and end with an estimate of the error.

The goal of solution adaptivity for aerothermodynamic analyses places requirements on the numerical scheme that it (1) be free from any requirement of special topological grid constraints in resolving all flow structures and (2) be able to automatically adapt specially required grid topologies wherever they are needed in the flow field. The grid adaptation mechanisms are most well developed for tetrahedral grids, and it is noted herein and elsewhere that three-dimensional simulations using high aspect-ratio tetrahedra to resolve the boundary layer and the bow shock produce poor heating. This problem is more severe as the Mach number is increased. Consequently, standard practice has evolved to use special topological elements (prisms) aligned with the shock and boundary layer to overcome the greatest source of difficulties in hypersonic heating predictions and to accept the errors (or the need for a finer grid) that are necessarily introduced by such practice. This is a significant obstacle to the eventual routine usage of solution-adaptive methodology.

Figure 17. Original grid for \( \gamma = \frac{7}{5} \) and pressure ratio switch = (2,3)
Acknowledgments

The first author thanks Iain Boyd, Graham Candler, Winifred Huo, John Kim, Dave Kinney, Dochan Kwak, Ryan McDaniel, Nagi Mansour, Dinesh Prabhu, James Reuther, Chun Tang, Kerry Trumble, Mike Wilder, Alan Wray, and Mike Wright for providing helpful information.

References

9. McDaniel, R. and Trumble, K., Private Communications
10. Wilder, M. and Reda, D., Private Communications
15. Kinney, D. and Bowles, J., Private Communications