# Current Status and Future Prospects for the Numerical Simulation of Hypersonic Flows

Graham V. Candler[1]
*University of Minnesota, Minneapolis, MN, 55455*

Dimitri J. Mavriplis[2]
*University of Wyoming, Laramie, WY, 82072*

Loretta Treviño[3]
*University of Minnesota, Minneapolis, MN, 55455*

**In this paper, we provide a perspective on how the field of computational aerothermodynamics has evolved and make some educated guesses as to where it is headed in the future. Over the past twenty years there have been great improvements in the accuracy, complexity, and reliability of hypersonic flow simulations. However, there is still room to push the boundaries of hypersonic computation with regard to physical modeling capabilities, sensitivity of solutions to the grid, and flexibility for complex geometries. As we move to billion element automated solutions on larger-scale heterogeneous processor systems, methods and solution approaches will have to evolve on many fronts. The continuing increase in computer performance and the improvements to numerical methods will work together to solve many of the multi-disciplinary problems that occur in hypersonic flight.**

## I. Perspectives

WE have come a long way in the simulation of hypersonic flows. Figure 1 shows a simulation of an AOTV geometry. AOTV is long-forgotten abbreviation for an Aero-Assisted Orbital Transfer Vehicle, which was a concept to use aerodynamics to change the orbital plane of a spacecraft – an idea that had currency for about a decade in the late 1970's to 80's. This calculation was state-of-the-art about 20 years ago when it was performed; it used an axisymmetric grid with 120 by 60 points and a nonequilibrium thermo-chemical model with eight chemical species (5-species air with $NO^+$ and electrons) and six temperatures ($T$, $T_e$, $T_{vN2}$, $T_{vO2}$, $T_{vNO}$, $T_{vNO+}$). This was a large calculation at the time and required significant run times on a Cray Y-MP, which was the most powerful supercomputer at the time. This machine had a clock speed of 167 MHz, and had up to 8 vector processors each with two functional units, for a peak theoretical speed of 333 Mflops per vector processor. The Y-MP had up to 512 MB of RAM, and some systems were configured with as much as 4 GB of solid state disk to improve out of core memory performance. It was mandatory to vectorize codes for this machine, because huge performance increases could be obtained by keeping the vector units busy. On a practical code, it was possible to obtain about 150 Mflops sustained performance.

The simulation shown in Fig. 1 provides a number of interesting perspectives on how our field has advanced:

- This image was literally cut and pasted to form a composite image of the simulation and a shadowgraph from a ballistic range experiment. (How quaint!)
- By our present standards, this calculation is almost laughable, as is the performance of the supercomputer used to perform the simulation.
- The thermo-chemical model is not appropriate for this problem; there was no need to use so many internal energies at this 4 km/s flight condition.

---

[1] Professor, Department of Aerospace Engineering and Mechanics, AIAA Fellow.
[2] Professor, Department of Mechanical Engineering, AIAA Associate Fellow.
[3] Graduate Research Assistant Department of Aerospace Engineering and Mechanics, AIAA Student Member.

- We couldn't even think about a three-dimensional problem with a reasonable thermo-chemical model; there was not enough computer performance or memory.
- There were problems with computing the heat flux in the stagnation region even for axisymmetric flows; there were additional problems with obtaining second-order accuracy with strong shocks and expansions.

It is interesting to compare this calculation to a recent simulation of the Space Shuttle Orbiter at a Mach 15 flight condition, shown in Figure 2. This calculation was performed on a shock-tailored grid provided by NASA; here we have increased the wall-normal resolution by a factor of four beyond the baseline so that the present grid has a total of 128 million elements. A five-species, two-temperature thermo-chemical model is used with a radiative equilibrium, finite-rate catalytic surface boundary condition. The problem was run on 175 Sun Microsystems X2200 servers with dual quad-core AMD Barcelona processors running at 2.3 GHz (a total of 1400 cores were used). Each server has a peak theoretical performance of 73.6 Gflops. The system uses a non-blocking QLogic Infiniband interconnect. This calculation requires about 18 hours for convergence to a steady-state on this machine using the US3D code.[1] It should be noted that we plan to perform stability analyses on the wind-side flow field, which requires at least 300 grid points in the wall normal direction. This 128 M element grid has 320 wall-normal points, and should be sufficiently refined for an accurate stability analysis. Thus, there is a valid technical reason for running such a large grid.

In comparison to the calculation shown in Figure 1, this is obviously an impressive increase in solution complexity and degree of difficulty. However, there are some other observations that can be made:

- The chemistry models – reaction rates, vibration-dissociation coupling models, etc. – are largely the same between the two simulations.
- Though not shown here, we still have serious problems associated with predicting the stagnation point heating, particularly for very blunt capsule-like geometries.
- The computational method had to be adapted and the code completely rewritten for the parallel cache-based computer architecture of current machines.
- The increase in computer performance is breath-taking: one off-the-shelf commodity server has about 100 times the theoretical performance of a world-class supercomputer from 20 years ago. (The precise performance increase is highly code and compiler-dependent, and depends on the number of vector units on the Cray Y-MP.) Increases in memory and disk space and speed are even more profound.
- Improvements in numerical methods are at least as important as improvements in computer hardware in increasing the degree of difficulty of present-day simulations.
- We have not made as much progress in the validation of our simulations as we have made in increasing the complexity of the calculations.

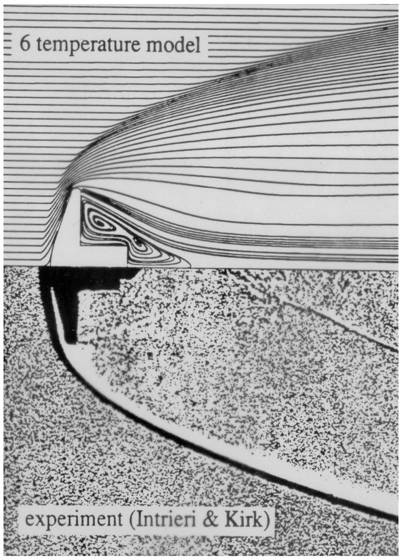

**Figure 1: Simulation of an AOTV geometry circa 1988.**

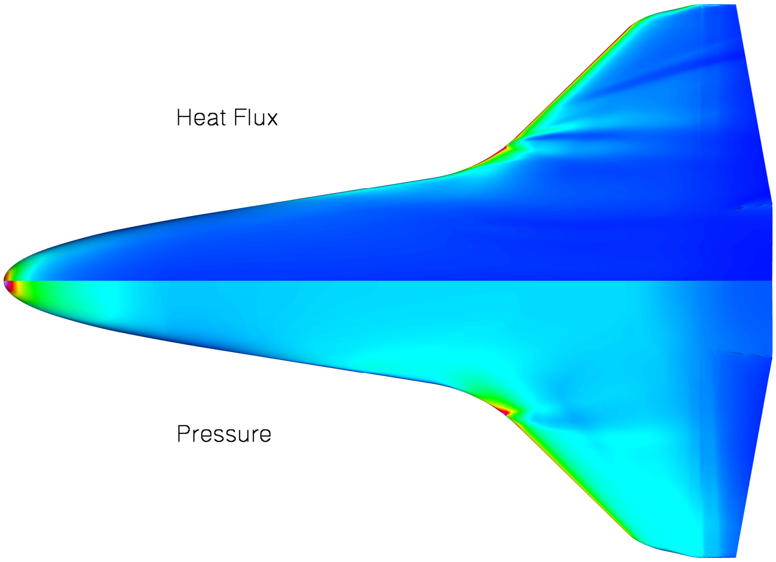

**Figure 2: Windside heat flux and pressure on the Space Shuttle Orbiter computed on a 128M element grid.**

- Grid generation remains a major problem, often consuming considerably more time than actually running a simulation. Commercial grid generators do not work well for 100 million element grids, and worse, it is very difficult to manage the grid generation process for massive grids – it takes just one poor element to ruin a simulation, and subtle grid problems can damage a solution.
- Data management and flow visualization are very time consuming and can be difficult.
- This is a steady-state solution, but many interesting problems are unsteady, resulting in increased run times and severe solution analysis problems.

We think that this type of improvement in solution fidelity, complexity, and overall degree-of-difficulty is going to continue and even accelerate over the next decade. The field is going to make major advances in multi-physics simulations of complete vehicles undergoing maneuvers; the accuracy of physical models will increase, and the sensitivity of the design to model uncertainties will become an integral part of the solution; numerical methods will continue to improve and will dramatically increase the fidelity, reliability and robustness of hypersonic flow simulations.

In this paper, we highlight some of the current weaknesses of present simulation methods and make some predictions about likely near-term advances in our field.

## II. Validation and Unknown Physics

Consider Fig. 3, which plots the measured ultraviolet emission from the nose region of a hypersonic ($M \cong 12$) blunted cone as a function of altitude for a photometer centered at 230 nm. The baseline curve represents the predicted emission from the flow field prior to the flight experiment; the prediction is not too bad at low altitude, but rapidly degrades until it is eight orders of magnitude too small at 70 km altitude.

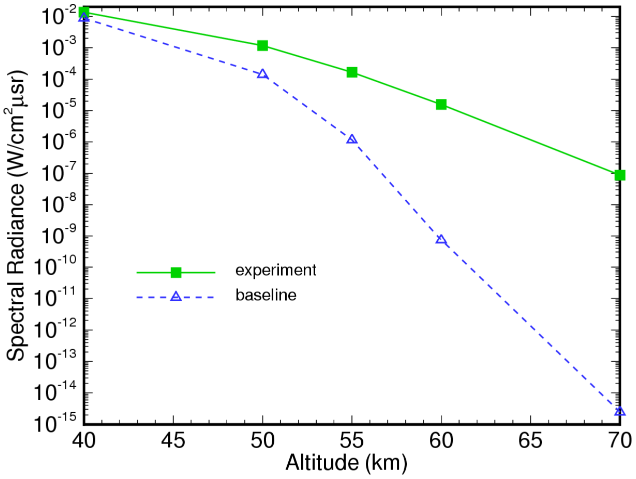

**Figure 3: Comparison of measured and pre-flight prediction of flow field emission in the 230 nm spectral region for the Bow Shock Ultraviolet-1 experiment.**[2]

The radiative emission is primarily from the nitric oxide gamma bands (transition from the first electronically excited state to the ground state). Comparisons with spectrally-resolved data show that the model does not represent the vibrational energy state of the excited state NO correctly, and the rate of formation of ground-state NO is also under-predicted. Quasi-classical trajectory (QCT) simulations were performed in an attempt to understand this discrepancy, and it was found that the literature values for the critical NO formation reaction was an order of magnitude too low at high temperature (typical temperatures are 14,000 K in this flow field). Furthermore, the internal energy state of the reactants affects the reaction rate. A new model was developed using the data obtained from the QCT study, and it gives excellent agreement with the flight data.[3]

This study illustrates a number of key lessons associated with accurate modeling of hypersonic flows:

- Hypersonic flows are characterized by complicated physics at extreme conditions for which fundamental data often do not exist. Extrapolations of models to hypersonic conditions can produce worthless predictions.
- Simulations of hypersonic flows are inherently multi-disciplinary.
- Validation of simulations with high-quality data is absolutely mandatory.
- There are many interesting and unknown physics in hypersonic flows; simulations can be used to improve our understanding of experiments and discover new physics.

## III. Some Current Problems

There remain a number of outstanding numerical issues in the simulation of hypersonic flows. A particular difficulty is associated with the simulation of high Mach number blunt capsule geometries that have a very large region of subsonic flow near the stagnation point. This class of flow magnifies numerical error generated at the strong shock wave; this error then accumulates in the stagnation region and corrupts the solution. The main remedies for this problem are:

- The grid must be aligned with the bow shock.
- A grid with a nose patch must be used, as opposed to a grid generated by revolving a 2D grid around the axis.
- Eigenvalue limiters must be used judiciously; other forms of dissipation can also be used to counter-act the error generated by the strong gradients across the bow shock.

None of these “fixes” actually solve the underlying problem, rather they reduce its magnitude and mask its effects with additional dissipation. Clearly, fundamental work needs to be done to reduce the sensitivity of the solution to the grid and specific details of the numerical method. Recent work by Gnoffo[4] on a multi-dimensional flux reconstruction is encouraging, also recent results with discontinuous Galerkin discretizations[5] show promise.

It is interesting to consider how a standard second-order accurate method functions on a high Mach number capsule flow. Figure 4 shows the surface grid used for a generic spherical capsule shape similar to the Apollo and CEV forebody. This figure illustrates the use of a grid patch in the stagnation region, which moves the grid singularity away from the axis. Figure 5 plots the computed heat flux and pressure using a wall-normal grid of 200 points that is not shock-adapted. The conditions correspond to a Mach number of 21, Reynolds number of $Re_D$ = 4.32 x $10^5$, and angle of attack of $12^o$; a cold-wall fixed wall temperature is used. The pressure distribution looks reasonable, but the heat flux is obviously completely wrong. Changing the flux method and other numerical method parameters do not improve the situation, and the error generated as the flow crosses the bow shock dominates the flow. This type of problem was discussed in Refs. 6, 7 in the context of using tetrahedral elements for aerothermal simulations. It was shown that when the grid is poorly shock aligned, the momentum balance across the shock produces a spurious post-shock component of velocity in the shock-tangential direction. This results in artificial vorticity in the post-shock flow, and once this vortical motion has been produced, it is very difficult to remove. Thus, future methods must focus on preventing the generation of this vorticity at the shock, rather than trying to mitigate its effects inside the shock layer.

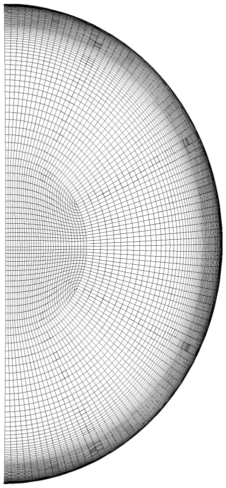

**Figure 4: Surface grid for capsule simulations.**

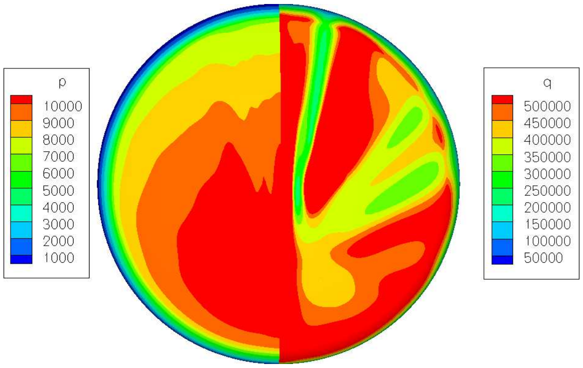

**Figure 5: Computed surface pressure (left) and heat flux (right) on a capsule forebody using a non-shock-aligned grid.**

Figure 6 plots the pressure and heat flux for the same case as Fig. 5, but with a shock-aligned grid. Here we have used the grid tailoring method in the DPLR code;[8,9] simulations were then performed with the US3D code.[1] The solutions were obtained with 4 values of the eigenvalue limiter; the exact details of this expression are not important here (see Ref. 7 for details; this calculation used a MUSCL approach to obtain second-order accuracy using the primitive variables and the Osher limiter). Rather, the sensitivity of the solution to this numerical method parameter is of primary importance. The solutions improve with increasing values of the eigenvalue limiter, which is essentially a measure of the level of dissipation in the calculation. It is important to note that this type of limiter is not applied in the computation of the wall-normal fluxes, because it causes extreme levels of dissipation and inaccurate heat fluxes.

This level of sensitivity to the grid alignment and numerical method parameters is unacceptable at this stage in the development of CFD methods.

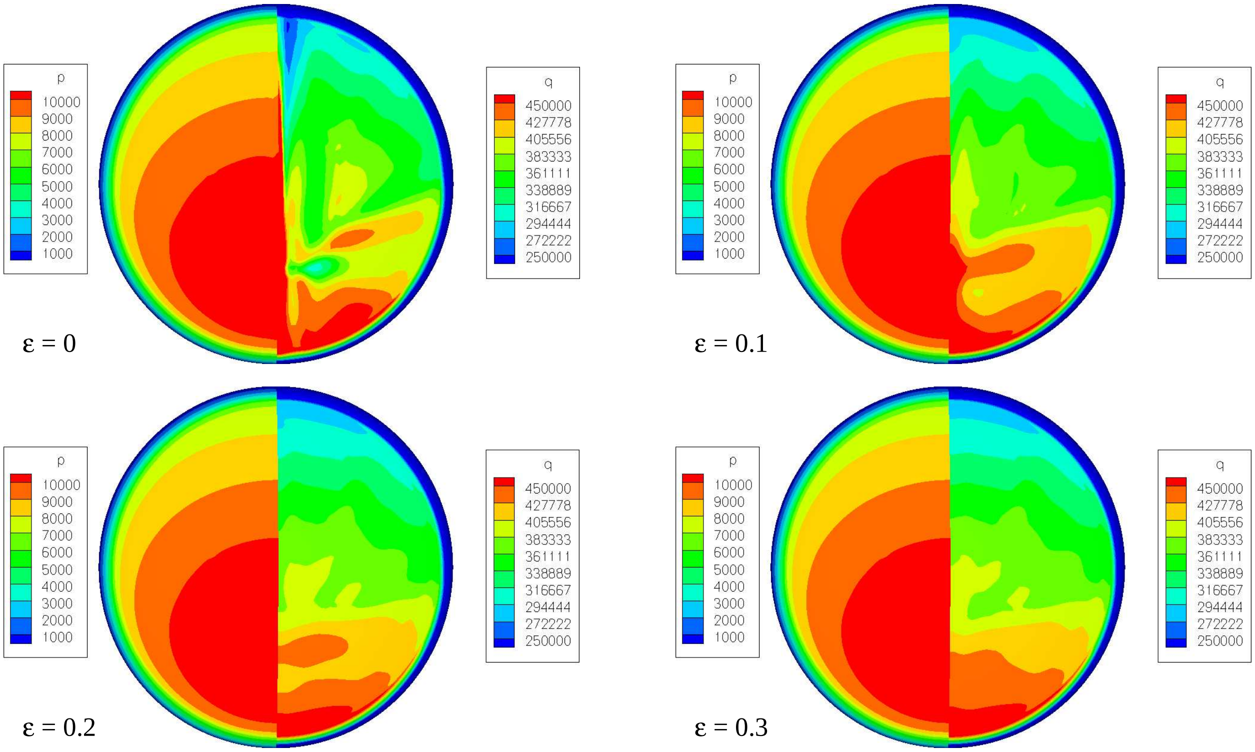

**Figure 6: Computed surface pressure and heat flux for the blunt capsule geometry, using different values of the eigenvalue limiter.**

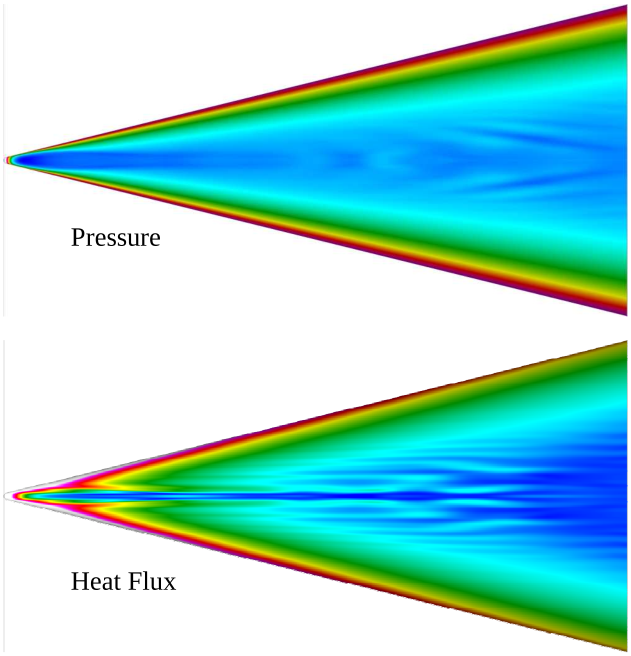

**Figure 8. Surface pressure and heat flux on the HIFiRE-5 2:1 elliptic cross-section blunt cone.**

It is hard to understate the difficulties of grid generation for large-scale problems. Consider Fig. 8, which plots the surface pressure and heat flux for the HIFiRE-5 2:1 elliptical cross-section code at representative Mach 7 flight conditions. The variations in surface properties are consistent with previous flow visualization experiments.[10,11] This calculation was performed on a 38 million element grid for the ¼ geometry (angle of attack and yaw are zero).

A non-obvious grid generation approach was taken here – first a moderately refined grid was generated with Gridgen and a solution computed. The bow shock envelope was then constructed by extracting the M = 7-ε isosurface in Tecplot. This surface is not adequately smooth, so it was numerically smoothed with a set of multi-dimensional sine and cosine basis functions. Finally, a high-resolution grid was obtained by algebraically generating a surface grid and intersecting wall-normal lines with the triangulated and smoothed shock surface. This process is much faster, less tedious, more flexible, and orders of magnitude less frustrating than using Gridgen for the high-resolution grid. Such an approach can be applied to new problems, however it would be very difficult or impossible to use for complex geometries such as the Shuttle Orbiter.

In our experience, no single commercial grid generator is adequate for large-scale grid generation. Furthermore, because of the demonstrated grid sensitivity of current methods, grid generation is a pacing item for large-scale complex geometry simulations.

## IV. Future Prospects

The numerical simulation of hypersonic flows can be expected to play an increasingly important role in the engineering analysis and design of hypersonic vehicles, as simulation capabilities become more complex and sophisticated, through the inclusion of more physics, a general drive towards higher accuracy, and the extension beyond traditional analysis problems to the development of a more comprehensive engineering design tool. This is driven in large part by rapidly advancing computational hardware which enables improved simulation capabilities. As demonstrated above, more powerful computers are enabling the use of much finer grids and smaller time steps, for increased accuracy. At the same time, additional available computational power is making feasible the incorporation of added physical models into production level engineering simulations, such as the use of coupled flexible structures and fluid flow, coupled radiation for aerothermodynamics, increasingly elaborate chemistry models, and a general trend towards more expensive large-eddy simulations (LES) and hybrid RANS-LES approaches in the place of RANS models. Additionally, numerical optimization techniques for multidisciplinary optimization are becoming feasible for hypersonic problems, and extended simulation objectives, such as the incorporation and propagation of uncertainties into the simulation process are being considered. However, in spite of the availability of increasingly capable hardware, in order to realize the full benefit of these new simulation capabilities, there are several issues which must be addressed. The first issue relates to the need to improve the reliability and robustness of current analysis capabilities. Secondly, improved automation of production level simulations will be imperative, and finally, the development of strategies for effectively harnessing the capabilities of rapidly advancing and changing hardware architectures will be required.

### A. Approaches for Robust and Reliable Simulations

As hardware and simulation capabilities advance, the need for ensuring reliable and robust simulation outcomes becomes increasingly important. For more complex simulations which incorporate increasingly complicated physical models, the failure of any of the growing number of simulation components has the potential to invalidate the entire simulation. For example, the importance of transition location for determining hypersonic heating rates is well known. However, it is unlikely in the near future that first-principles simulations of transition (as opposed to stability analyses based on a perturbation analysis) will become incorporated in production engineering hypersonic simulations. Therefore, elaborate and computationally expensive hypersonic simulations incorporating highly resolved meshes and sophisticated physical models (i.e. chemistry, radiation, ablation, fluid-structure interaction) may easily be invalidated if the transition model fails to provide accurate transition locations. One obvious approach to this problem is to invest more effort into the development of reliable transition models, although this will also be complicated by the increasingly complex flow physics being considered by such simulations. An alternate approach which can be considered in the presence of additional computational power is the quantification of the uncertainty in the simulation outputs due to the transition model. This can be achieved through brute-force Monte Carlo techniques, running a large number of possible scenario simulations, or through the incorporation of sensitivity analysis techniques, which enable the propagation of input or model parameter uncertainties through the simulation process in order to obtain uncertainties in the simulation output objectives. Here the effect of the transition model has been taken as an example, but this approach can be used for any of the variety of physical models which are part of the hypersonic simulation process. For example, the effect of uncertainties in chemical reaction rate constants on performance objectives can be assessed in this manner, or even the uncertainty entailed through the omission of specific reaction rates (or any other physical phenomena) can be assessed. The development of these types of uncertainty quantification methods should prove to be a useful tool for guiding future investment decisions on which physical models need to be incorporated and/or refined for improving our predictive ability of specific engineering objectives. The work of Wright, Bose, and Chen[12] provides an excellent example of the use parametric uncertainty analysis to determine the sensitivity of a flow field to the model uncertainties.

### B. Advanced Methods Leading to Automated Solutions

The increased productionalization of hypersonic flow simulations made possible though more capable computer hardware for tasks such as data-base filling, parameter sweeps, or multi-objective optimization, puts additional emphasis on automation and means that user intervention in selected cases will no longer be feasible. As shown above, one of the well known difficulties with high-speed flow simulations is in the sensitivity of the surface heating results to details of the shock capturing process and the grid design/shock-alignment. Current wisdom holds that shock capturing using tetrahedral elements leads to poor surface heating predictions, thus requiring the use of hexahedral elements in these regions, as well as user knowledge for aligning these mesh elements with the shock position.[6,7] However, because hexahedral elements are not well suited for meshing complex geometries, the most effective approach for hypersonic simulations of complicated configurations relies on the use of hybrid hexahedral-tetrahedral unstructured meshes constructed with a great deal of user intervention and expertise for locating the different types of elements, and for aligning the mesh with strong shock features. Cleary, full automation of the grid generation process is desirable if not necessary in order to enable ubiquitous simulations of various geometrical configurations over a wide range of flow conditions. There are currently two possible approaches for addressing this problem. The first approach concentrates on devising discretizations which are less sensitive to the type and alignment of the mesh in the shock region. One approach in this direction is found in the development of truly multi-dimensional approximate Riemann solvers for use on unstructured meshes.[4,13] Another possibility involves the use of high-order methods, such as discontinuous Galerkin discretizations, which are capable of sub-cell shock resolution, thus minimizing the shock / mesh face interaction region.[5,14,15] However, both of these approaches are still in their infancy and require substantial further development in order to be deployable to production engineering simulations. An alternate approach can be found in the development of mesh optimization procedures, whereby the mesh is locally modified in a solution-adaptive manner to align with the developing shock front. Although such techniques have been demonstrated in two dimensions,[16,17] for three-dimensional problems, in the event tetrahedral elements are initially located in shock regions, additional techniques will be required to automatically remove or de-emphasize diagonal faces which intersect the shock profile, resulting in the formation of prismatic or hexahedral aligned cells in these regions. A variant strategy consists of computing the sensitivity of the principal simulation objective (e. g. surface heating) with respect to the mesh point coordinates, through an adjoint calculation, and attempting to optimize the mesh configuration in order to reduce the error in the objective. This approach, which offers prospects for increased automation, has been demonstrated in two dimensions, although it does not address the elimination of diagonal faces as required in three-dimensions. A longer term and more ambitious goal is the complete automation of the grid generation process, through adaptive mesh refinement (and movement) techniques driven by adjoint-based functional output error estimation techniques. In this scenario, an initial generic coarse grid with minimal user input (and no knowledge of anticipated flow features) is used as a starting point, and is adaptively refined throughout the flow solution process, until the desired accuracy in the simulation objective of interest is achieved, as determined by the adjoint-error estimation procedure. This approach has been demonstrated successfully for other demanding aerodynamic flow problems such as three-dimensional sonic-boom applications,[18,19] and is currently being investigated in the context of discontinuous Galerkin discretizations for hypersonic flows in two dimensions.[20,21] However, such an approach will most likely require a numerical method that is less sensitive to the grid alignment than current methods.

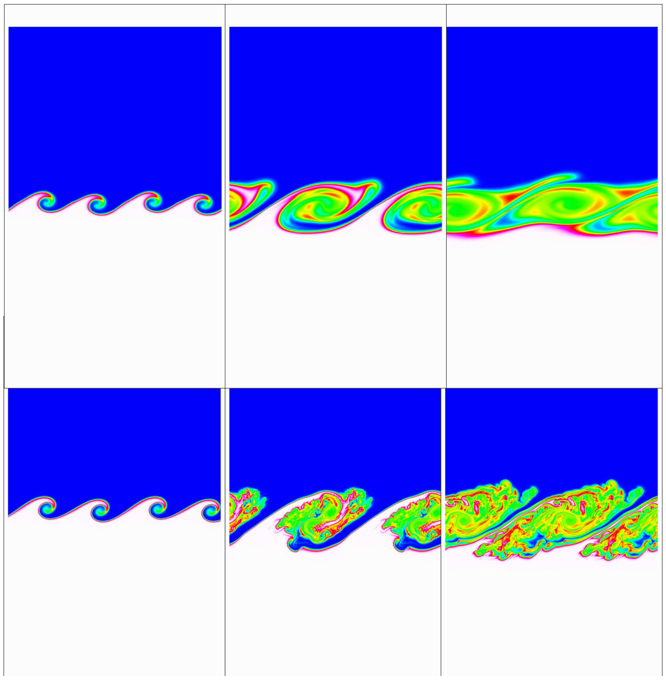

**Figure 9: Evolution of a compressible shear layer computed with a standard 3rd order accurate upwind biased method (top) and the low-dissipation method (bottom); density is shown.**

As our field moves toward unsteady hybrid RANS-LES and LES approaches, a greater premium will be placed on solution accuracy.

Current practices involve the use of high-order compact differences with some form of limiting, filtering, or weighted ENO smoothing to control high frequency error near strong gradients. Impressive results have been obtained with this approach, but it is necessarily limited to simple geometries. Furthermore, the large non-localized stencil used in these methods requires that limiting is applied in large fractions of the flow (and precisely in the regions of interesting flow physics). Great care must be taken in the development and testing of the methods to ensure high accuracy solutions.[22] Discontinuous Galerkin discretizations offer an alternate strategy for achieving high order accuracy based on a compact nearest neighbor stencil, offering the possibility of sub-cell shock resolution, and thus reducing the extent of accuracy reduction due to limiting[15,20,26] or added artificial dissipation,[5,14] although robustness and efficiency concerns remain with these methods.

A different approach is taken in Ref. 23, in which a high accuracy, low dissipation method is derived by imposing a secondary conservation constraint on the derivation of the numerical flux function. The resulting method is second-order accurate in space and time (fourth-order extensions are possible), but more importantly many practical problems can be solved without any flux limiting. Figure 9 shows a compressible mixing layer computed with a standard third-order upwind-biased flux and the new kinetic-energy consistent flux. See Ref. 24 for application of this approach to the hybrid RANS-LES of scramjet combustor flow. Clearly in Fig. 9, there is a huge increase in the range of length scales resolved by the new approach, and this is an illustration that accuracy and order of accuracy are two different measures of a method's quality.

The ultimate utility of these approaches will depend on detailed code-to-code comparisons and validation with high-quality experimental data. Some promising results on simple hypersonic problems have been obtained, but it will be some time until the new methods have been evaluated and proven on complex geometry flows.

### C. Toward Billion Element Simulations

The second principal requirement for advancing hypersonic simulation capabilities rests on devising strategies for harnessing the power of rapidly advancing computer architectures. By all current accounts, the path towards increasingly high performance computing involves radically higher levels of parallelism. This is evident in the rapid growth in the number of cores in current and planned leadership class machines, many of which comprise from 100,000 to 1 million cores, due to the emergence of multicore architectures, as well as the appearance of GPUs and Cell processors which achieve high performance through extensive multithreading. At a minimum, the current message-passing parallel programming paradigm will be severely challenged by this rapidly expanding level of parallelism, and hybrid or multi-level parallelism programming models (for example using OpenMP-MPI or GPU-interface-MPI models) will likely emerge as new standards. At the same time, the availability of substantially more capable hardware will enable much higher resolution simulations for increased accuracy, leading to billion mesh cell production calculations compared to today's tens of million cell simulations. However, the generation, partitioning, handling, archiving and maintenance of such large meshes promises to pose a formidable simulation bottleneck, largely due to the slower progress of supporting software and hardware, such as parallel grid generation capabilities, disk I/O, and internet connectivity. In fact, the increasing scale of such simulations poses problems not only for the grid generation and management portion of the simulation, but for many other aspects as well, and the full end-to-end simulation, from geometry definition to flow visualization and engineering design, must be automated in a comprehensive approach. This will necessarily include not only parallel mesh generation and adaptive refinement, but also parallel access to detailed geometry (CAD) description, parallel simulation and optimization, and in-situ visualization techniques, all of which combine to obviate the need for large data migration between disparate computer architectures.

On the other hand, the advent of rapidly expanding levels of parallelism for hypersonic flow simulations poses additional challenges for problems in which extreme grid resolution may not be required, but where simulation costs remain high, as in the case of unsteady simulations or multi-objective design optimization problems. In such cases, spatial decomposition alone may not provide sufficient concurrency to adequately scale on tens or hundreds of thousands of cores, and additional opportunities for parallelism will need to be investigated. This may be achieved through parallel coupled physics solvers, where different physics simulation components are solved simultaneously in a loosely coupled fashion, through the use of space-time parallelization strategies for long-time integration unsteady problems, or through managing multiple instances of the simulation code for multi-objective optimization problems.

The use of alternate discretizations offers another approach for dealing with radically higher levels of parallelism. Higher-order methods, such as discontinuous Galerkin discretizations (DG) have been shown to scale extremely well on large numbers of processors or cores, even for relatively coarse meshes.[25] This is due to their use of a compact stencil with relatively expensive but local dense matrix operations within each mesh element or cell. The superior accuracy afforded by the asymptotic properties of these discretizations should also prove to be beneficial for complex hypersonic simulations. At the same time, DG methods offer the potential for relieving many of the mesh generation and data-handling bottlenecks mentioned previously, due to the fact that they achieve equivalent or superior accuracy on much smaller meshes, effectively replacing billion cell second-order approximations with million cell high-order approximations. Additionally, these methods can be thought of as data-compression techniques, where the final solution is represented by a relatively small number of modal coefficients which are used to reconstruct a high-order polynomial solution, rather than the large number of sampling points required for accurate low-order solution representations, thus simplifying the data transfer problem for highly resolved simulations. However, the success of high-order discretizations is predicated on a smooth behavior of the solution, which is often not the case for hypersonic flow problems, which naturally involve near discontinuous phenomena such as shock waves and slip lines. Therefore, much effort has been devoted to enabling limiting, filtering, or diffusive approaches for feature capturing for DG discretizations in high-speed flows.[5,14,26] The long term success of these methods will ultimately hinge on prospects for efficient and robust shock capturing and solution, including automated adaptive procedures, accurate geometry definition and querying, and postprocessing capabilities which capitalize on the efficient compact data-representation inherent in higher order methods.

## V. Conclusion

Our field is highly multi-disciplinary and our problems are dominated by tightly coupled non-linear effects – perhaps more so than in any other engineering discipline. This makes the numerical simulation of hypersonic flow very demanding in terms of computational resources and the required sophistication of the numerical methods. We have made great progress in the development of physical models and robust simulations for the numerical simulation of hypersonic flows, but the best is yet to come and there are many outstanding puzzles to be solved. Advanced numerical simulations are going to play a major role in the solution of these problems and in the discovery of new hypersonic flow physics.

The increase in computer performance has obviously driven much of our advances, but the continuing development of numerical methods has certainly been at least as important. As we move to new even larger-scale computers, numerical method development is going to play an increasingly important role in extracting reasonable fractions of the theoretical performance from these machines. We must work on methods that produce high-quality solutions that are less sensitive to the alignment of the grid with the bow shock and other grid imperfections, particularly as we require automated solutions for optimization. We can debate about which particular approaches are most likely to succeed, but ultimately the best will emerge, and it is likely and healthy that there will be a mix of approaches.

Finally, although not specifically emphasized in this paper, an important consideration for the continued improvement of hypersonic flow simulations involves the need for high-quality experimental data-sets designed specifically for code validation.

## Acknowledgments

We would like to thank Drs. Meelan Choudhari and Seokkwan Yoon, NASA, for providing the impetus to write this paper. This research is supported by the Air Force Office of Scientific Research (AFOSR) under Grant Nos. FA9550-08-1-0397 and FA9550-07-1-0164 and NASA under Grants NNX07AC28A and NNX07AC31A. The views and conclusions contained herein are those of the authors and should not be interpreted as necessarily representing the official policies or endorsements, either expressed or implied, of the AFOSR or the U.S. Government.

## References

[1] Nompelis, I., T.W. Drayna, and G.V. Candler, “A Parallel Unstructured Implicit Solver for Hypersonic Reacting Flow Simulation,” AIAA-2005-4867, June 2005.

[2] Levin, D.A., G.V. Candler, R.J. Collins, P.W. Erdman, E. Zipf, P. Espy, C. Howlett, “Comparison of Theory with Experiment for the Bow Shock Ultraviolet Rocket Flight,” *Journal of Thermophysics and Heat Transfer*, 7, 1, 30-36, 1993.

[3] Bose, D., and G.V. Candler, “Simulation of Hypersonic Flows Using a Detailed Nitric Oxide Formation Model,” *Physics of Fluids*, 9, 4, 1171-1181, 1997.

[4] Gnoffo, P.A., “Multi-Dimensional Flux Reconstruction for Simulation of Hypersonic Heating on Tetrahedral Grids,” AIAA-2009-0599, Jan. 2009.

[5] Barter, G.E., and D.L. Darmofal, “Shock Capturing with Higher-Order PDE-Based Artificial Viscosity,” AIAA-2007-3823, June 2007.

[6] Gnoffo, P.A., and J.A. White, “Computational Aerothermodynamic Simulation Issues on Unstructured Grids,” AIAA-2004-2371, June 2004.

[7] Candler, G.V., M.D. Barnhardt, T.W. Drayna, I. Nompelis, D.M. Peterson, P. Subbareddy, “Unstructured Grid Approaches for Accurate Aerothermal Simulations,” AIAA-2007-3959, June 2007.

[8] Anon., “DPLR Package Users Guide,” NASA Ames Research Center.

[9] Wright, M.J., D. Bose, and G.V. Candler, “A Data-Parallel Line Relaxation Method for the Navier-Stokes Equations,” *AIAA Journal*, Vol. 36, No. 9, 1988, pp. 1603-1609.

[10] Huntley, M.B., P. Wu, R.B. Miles, and A.J. Smits, “MHz Imaging of Boundary Layer Transition on Elliptic Cones at Mach 8,” AIAA-2000-0379, Jan. 2000.

[11] Huntley, M.B., and A.J Smits, “Transition Studies on an Elliptic Cone in Mach 9 Flow Using Filtered Rayleigh Scattering,” *European Journal of Mechanics B – Fluids*, Vol. 19, 2000, pp. 695-706.

[12] Wright, M.J., D. Bose, and Y.-K. Chen, “Probabilistic Modeling of Aerothermal and Thermal Protection Material Response Uncertainties,” AIAA Journal, Vol. 45, No. 2, Feb. 2007, pp. 399-410.

[13] Kitamura, K., P. Roe, and F. Ismail, “Evaluation of Euler Fluxes for Hypersonic Flow Computations,” *AIAA Journal*, Vol.47, No.1, 2008, pp. 44-53.

[14] Persson, P., and J. Peraire, “Sub-Cell Shock Capturing for Discontinuous Galerkin Methods,” AIAA-2006-0112, Jan. 2006.

[15] Krivodonova, L., J. Xin, J.-F. Remacle, N. Chevaugeon, J. Flaherty, “Shock Detection and Limiting with Discontinuous Galerkin Methods for Hyperbolic Conservation Laws,” *Appl. Numer. Math.*, Vol. 48, 2004, pp. 323–338.

[16] van Rosendale, J., “Floating Shock Fitting Via Lagrangian Adaptive Meshes,” AIAA-1995-1721, June 1995.

[17] Paraschivoiu, M., and J. -Y. Trepanier, M. Reggio, and R. Camarero, “A Conservative Dynamic Discontinuity Tracking Algorithm for the Euler Equations,” AIAA-1994-0081, Jan. 1994.

[18] Wintzer, M., M. Nemec, and M. Aftosmis, “Adjoint-Based Adaptive Mesh Refinement for Sonic Boom Prediction,” AIAA-2008-6593, Aug. 2008.

[19] Casper, J., S. Cliff, S. . Thomas, M. Park, M. McMullen, J. Melton, D. Durston, “Assessment of Near-Field Sonic Boom Simulation Tools,” AIAA-2008-6592, Aug. 2008.

[20] Wang, L., and D.J. Mavriplis, “Adjoint-Based h-p Adaptive Discontinuous Galerkin Methods for the Compressible Euler Equations,” AIAA-2009-0952, Jan. 2009.

[21] Hartmann, R. and P. Houston, Adaptive Discontinuous Galerkin Finite Element Methods for the Compressible Euler Equations,” *Journal of Computational Physics*, Vol. 183, No. 2, 2002, pp. 508-553.

[22] Taylor, E.M., Wu, M., and Martin, M.P., “Optimization of Nonlinear Error for Weighted Non-Oscillatory Methods in Direct Numerical Simulations of Compressible Turbulence,” *Journal of Computational Physics*, Vol. 223, 2007, pp. 384–397.

[23] Subbareddy, P., and G.V. Candler, “A Fully Discrete, Kinetic Energy Consistent Finite-Volume Scheme for Compressible Flows,”*Journal of Computational Physics*, available online Nov. 2008.

[24] Peterson, D.M., G.V. Candler, and T.W. Drayna, “Detached Eddy Simulation of a Generic Scramjet Inlet and Combustor,” AIAA-2009-0130, Jan. 2009.

[25] Nastase, C., and D.J. Mavriplis, “A Parallel hp-Multigrid Solver for Three-Dimensional Discontinuous Galerkin Discretizations of the Euler Equations,” AIAA-2007-0512, Jan. 2007.

[26] Rudin, L.I., S. Osher, and E. Fatemi, “Nonlinear Total Variation Based Noise Removal Algorithms,” *Physica D*, Vol. 60, 1992, pp. 259-268.