Analysis of Heating Rates on the Conical Surface of Apollo Command Module flying AS-202 Flight

By Jim L. Wong

Abstract

The aerodynamic and aerothermal heating at the leeward surfaces of the Apollo capsule flying high angle of attack were found difficult to simulate using conventional tools. Due to the large subsonic region in the proximity of the shoulder of the base shield, correlation-based tools were found to be inadequate. CFD tools are too time consuming for conceptual design purposes, and cannot account for the transient effects of material response such as wall temperature and blowing. An accurate and timely simulation is essential to effectively size the thermal protection system (TPS), to enhance its performance, and to ensure the safety of the crew.

Northrop Grumman Corporation applied a modified version of the MASCC/ATAC program, an inviscid flowfield code with boundary layer solver to simulate this scenario. The MASCC/ATAC code is believed to be the only non-CFD code that can rigorously perform the simulations on the windward surface. The predictions on the windward side of the conical surface were found to be in good agreement with flight data over a wide range of environments. The results are presented in the paper.

Introduction

The leeward surfaces of hypersonic reentry vehicles (RV) such as the Apollo command module flying at high angle of attack are often difficult to simulate. The configuration of the Apollo capsule is straightforward, comprised of a base shield with large radius, a conical surface, and a toroidal shoulder to blend these two surfaces. Over the conical side of the capsule when flying at high angle of attack, the streamlines may separate and reattach, causing heating augmentation. Even at the windward side of this conical surface where the flowfield remains attached, the thick boundary layer due to geometry may allow the flowfield to be tripped easily, and to become turbulent. Yet an accurate and timely simulation of the reentry body is essential to trim the non-value adding thermal protection system (TPS) mass, to enhance its performance, and to ensure the safety of the crew.

Though the Apollo Command Module has a relatively simple configuration, the high-heat-flux and high-shear flowfield near the shoulder of the base shield causes great difficulty for the heating to be determined, in particular with angle of attack. Conventional approaches to perform the aerothermal simulations include correlation-based tools and Computational Fluid Dynamic (CFD) codes.

Correlation-based codes such as MINIVER/EXITS approximate the heating near the shoulder region of the capsule as the stagnation point of a simple object such as a sphere, and downstream locations are approximated as simple objects such as cone, wedge, and cylinder. These less than rigorous heating predictions are then correlated through the use of wind tunnel data. A set of correlation multiplying factors is built to correct these simple models to the ground test data at limited conditions. After finishing a trajectory, the computed heat transfer coefficient and recovery enthalpy histories are then used in a thermal analyzer to determine the material response.

The weaknesses of the correlation-based approach include

1. Because of the large subsonic region on the base shield, the location of stagnation point on the windward surface of the blunt capsule is not obvious.
2. The radius of the equivalent sphere at the stagnation point is difficult to calculate.
3. The correction multiplying factors used to correlate heating are often based on wind tunnel data collected at cryogenic freestream temperature and a relative low speed of Mach 6 to 10. All the aerochemistry effects are ignored.
4. Uncoupling the aerothermal and thermal prediction processes ignores the effects of wall temperature on flowfield. Such effects may be important in leeward surfaces where flowfield is turbulent while temperature is modest.

A potentially more accurate approach is to apply various computational fluid dynamic (CFD) codes to perform the simulations. Recently Wright et. al. (AIAA 2004-2456) of NASA Ames used the Data Parallel Line Relaxation (DPLR) Navier-Stokes code to predict the flowfield with good success. However, the CFD tools are too time consuming for up-front design purposes, and do not account for the transient effects of material response such as wall temperature and blowing.

Computation Algorithm

Northrop Grumman Corporation applied a modified version of the Maneuvering ABRES (Advanced Ballistic Reentry Systems) Shape Change Code/ aeroheating and thermal analysis code (MASCC/ATAC) program, an inviscid flowfield code with boundary layer solver to simulate this scenario. The MASC Code[1] was developed for USAF/BMO (Ballistic Missile Office) by Aerotherm with Dr. Tony Lin and Jim Wong as the technical monitors and was based on the ASC Code[2]. Dr. Alvin Murray has made substantial improvements since the 90s[3] and renamed this code to ATAC on 2001.

MASCC/ATAC Algorithm

The MASCC/ATAC code solves the integral mass conservation equation between the shock and vehicle body to determine the bowshock geometry. The boundary layer flowfield and heat transfer are solved with the Momentum Energy Integral Technique (MEIT). The wall temperatures at every body point are accurately predicted by running a large number of indepth conduction simulations at every time-cut using the Charring Material Ablator (CMA) code, a one dimensional thermal conduction code with ablation and surface chemistry capabilities. Details of this series of codes are well documented in the corresponding users manuals, and are only briefly summarized here.

The MASCC/ATAC code has been extensively validated against many USAF flight vehicles. For the windward surface of the Apollo capsule, Murray[4] found that this algorithm could accurately and timely predict the complex flowfield and heating, and validated the code against Apollo wind tunnel and flight data.

The MASCC/ATAC predictions were compared with the surface pressure and heat flux distributions on a sub-scale model of the Apollo capsule as presented in Ref. 5. A comparison with Marvin's tests in helium at Mach 20 is presented in Figure 1 to demonstrate the capabilities of the code to simulate the 3D flowfield, in particular the high-heat-flux-high-shear shoulder region. Murray also validated the algorithm with the Apollo AS-202 flight data. Trajectory information was taken from the Post-launch Report[6] and is presented in Fig 2. This flight comprised of an initial high-speed entry, a skip back to vacuum, and followed by a second entry. This trajectory was a relatively non-severe convective heating environment with negligible radiation heating.

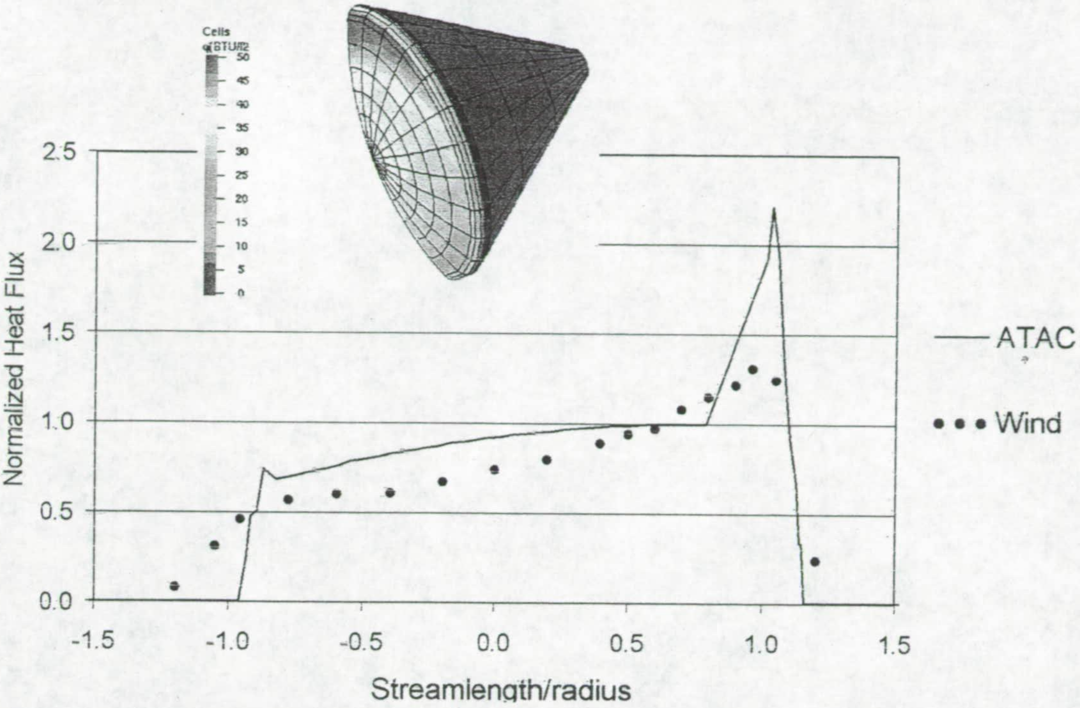

Figure 1: Comparison of ATAC with Heat Flux Data at Mach 20 in Helium, α=25

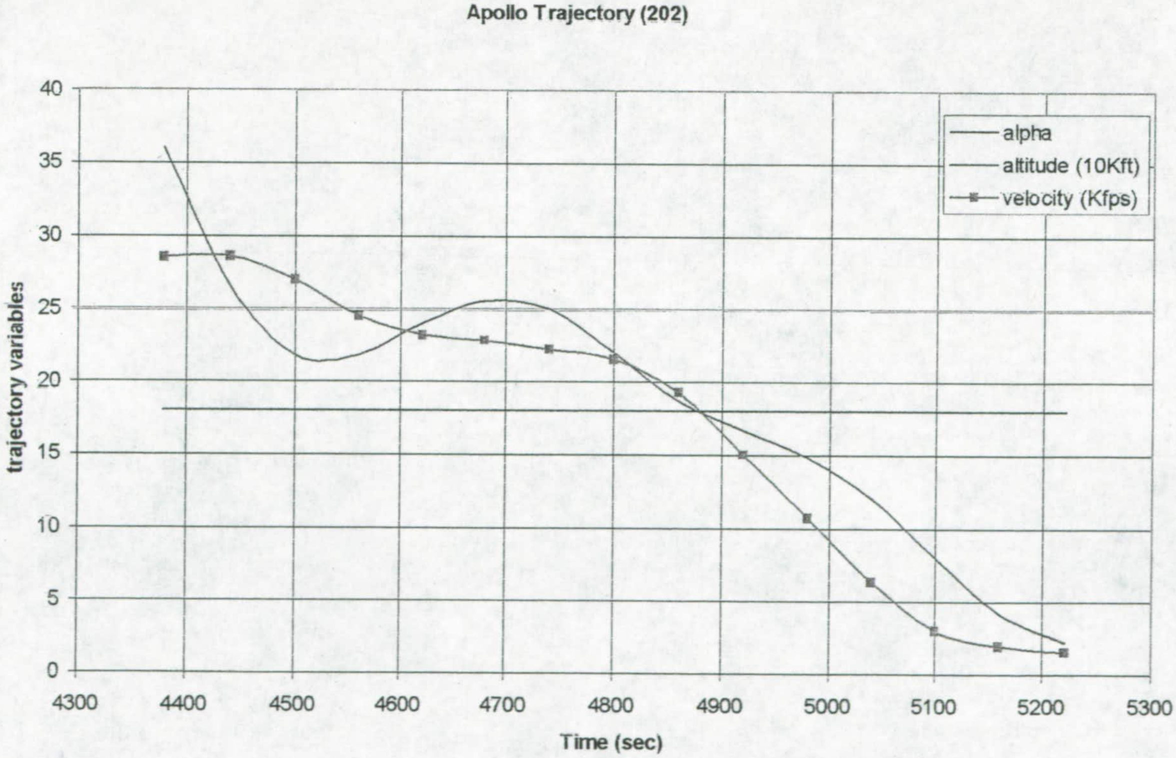

Figure 2: Apollo A202 Flight trajectory

Murray made comparisons with AS202 thermocouple measurements, char depth measurements and char density measurements, and the results were documented in his publication. In his calculations, he approximated the pressure coefficient (Cp) on the leeward (conical) surface to be zero, or the static pressure to be the ambient pressure. For most hypersonic vehicles, this assumption is reasonable as the leeward heating is mild, and abort scenarios dictate the TPS sizing in this region.

Overall, Murray performed an excellent demonstration of the capabilities of algorithm to simulate the complex flowfield over the Apollo capsule, in particular the high-heat-flux-high-shear shoulder region. This code is believed to be the only non-CFD code that can rigorously perform such task. Furthermore, the simulation of the coupled aerodynamic, aerothermal and indepth thermal conduction takes roughly one minute for the entire trajectory on a modern PC. However, the built-in geometry input module restricts the angle of attack such that the stagnation point must reside on the base shield with the big blunt radius, and the entire conical surface to have pressure coefficient Cp = 0, including the windward portion of the conical surface. Both of these restrictions or assumptions are not acceptable to the design of a modern reentry capsule.

New Computation Algorithm and Validation of Results

One simple fix to solve these two restrictions is to rotate the grid by the angle of attack. The author repeated the works by Murray with the grid rotation and modified the stagnation point location based on CFD results. The Northrop Grumman team performed many CFD simulations to determine the flowfield over the capsule, and found that the pressure over the conical surface had an influence over the flowfield on the windward surface because of the large subsonic region. The current simulations focus on the windward portion of the conical surface. Because the flowfield must go through a sharp shoulder, the validity of existing boundary layer transition (BLT) criteria are questionable. For simplicity, the boundary layer flow is assumed to be turbulent for the entire flight.

The results at the windward side of the conical surface using this new algorithm were compared with the Apollo A202 flight data. The locations of the calorimeters were obtained from Wright's paper and are presented in Fig 3. The prediction vs. flight data for sensor A is delineated in Fig 4. The predictions showed that the surfaces observed high heating during first entry, a benign skip followed by the second entry with relatively modest heating. Prior to the 4500 seconds, the agreement needs additional work as the turbulent assumption may not be valid. Between the 4500 and 4850 seconds, the agreement is remarkable for a wide range of Mach and Reynolds Numbers. The prediction deviated from the flight data during the second peak. One theory of this disagreement is that the thermal soak-back during the benign skip caused the phenolic filler of the AVCOAT TPS to ablate while the honeycomb structure remained intact in the proximity of the shoulder. The exposed honeycomb increased the surface roughness, causing additional heating augmentation.

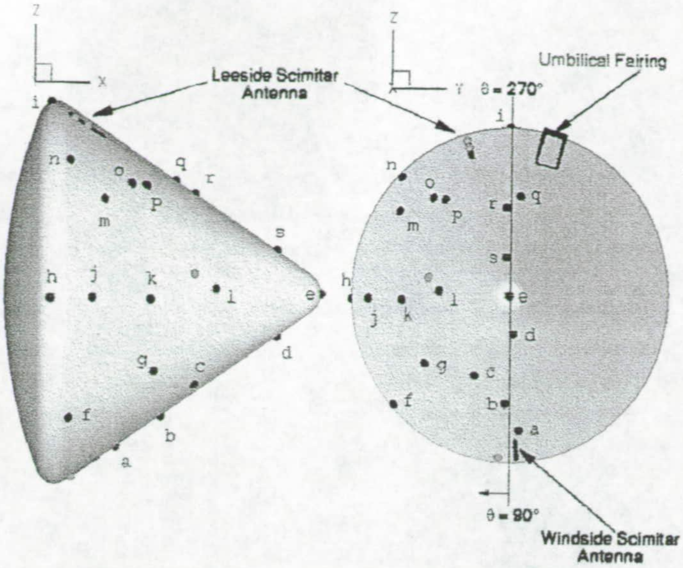

Figure 3: Locations of Apollo sensors

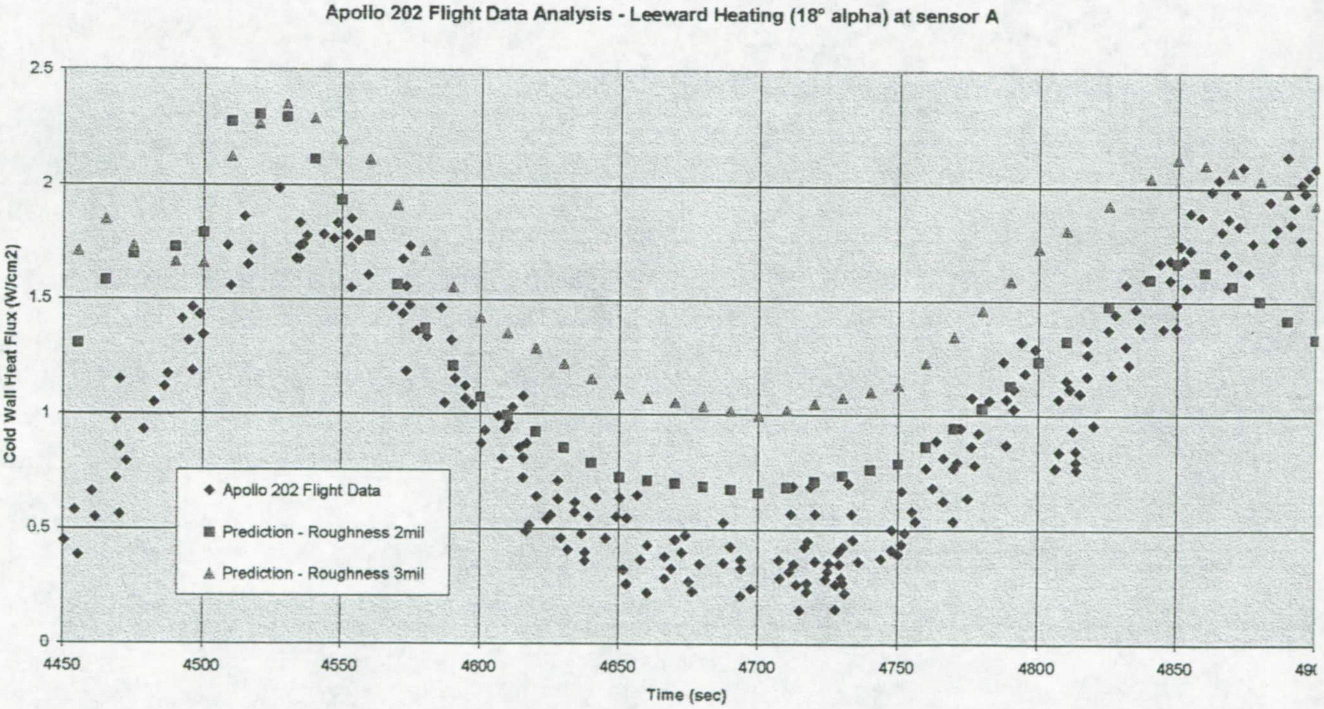

Figure 4: Comparison of ATAC with Apollo A202 Flight Data at Sensor A

Conclusion

The aerodynamic and aerothermal heating at the conical surfaces of the Apollo capsule flying high angle of attack were found difficult to simulate. Due to the large subsonic region in the proximity of the shoulder of the base shield, the correlation-based tools were found to be inadequate. However, the CFD tools are too time consuming for design purposes, and cannot account for the transient effects of material response such as wall temperature and blowing.

The MASCC/ATAC code is shown to be a non-CFD code that can rigorously perform the simulations on the windward base shield. The geometry module of that code was modified to accommodate the capsule configuration, and the predictions on the windward side of the conical surface were found to be in remarkable agreement with flight data over a wide range of environments.

References

1. T. H. Squire and A. L. Murray. *Advanced Reentry Effects Technology (ARET) Program.* Technical Report TR-89-11, BSD, August 1988. Final Report and User's Manual for the Maneuvering ABRES Shape Change Code (MASCC88), Task Order 0003.
2. King, H. C., Muramoto, K. K., Murray, A. L., and Pronchick, S. W., "ABRES Shape Change Code (ASCC86): Technical Report and User's Manual," Acurex Corporation, FR-86-24/ATD, Mt View, CA, Dec 1986.
3. Murray, A. L., "User's Manual for the Aeroheating and Thermal Analysis Code (ATAC3D)", ITT Aerotherm, FR 0915-03-001, Huntsville, AL, May 2003.
4. Murray, A., "Coupled Aeroheating/Ablation Analysis for Re-entry Vehicles", TFAWS 2003.
5. Marvin, J., Tendeland, T., and Kussoy, M., "Apollo Forebody Pressure and Heat-Transfer Distributions in Helium at $M_\infty=20$," NASA Ames Research Center, NASA TM X-854, Moffett Field, CA, Nov 1963.
6. "Postlaunch Report for Mission AS-202 (Apollo Spacecraft 011)," NASA Manned Spacecraft Center, Report MSC-A-R-66-5, Houston, TX, Oct 1966.