Comparisons of Measured and Modeled Aero-thermal Distributions for Complex Hypersonic Configurations

Denton G. Sagerman *
University of Dayton, Dayton, Ohio, 45469, USA

Nastassja Dasque †
Wright-Patterson Air Force Base, Ohio, 45433, USA

Markus P. Rumpfkeil ‡
University of Dayton, Dayton, Ohio, 45469, USA

Barry Hellman §
Wright-Patterson Air Force Base, Ohio, 45433, USA

The ability to quickly and accurately predict the thermal signature of a complex geometry is important in the early design stages for any aircraft. Due to the lack of hypersonic facilities with this capability, a recent effort has been made to quantify the ability of the Mach 6 tunnel at Wright-Patterson Air Force Base (WPAFB) for this task. The Mach 6 High Reynolds Number Facility at WPAFB in Dayton, Ohio, has been non-operational for the past twenty years, but a recent resurgence in the need for hypersonic test facilities has led to the reactivation of the tunnel. With its restoration, the facility is to include new capabilities to assess hypersonic aero-thermodynamic effects on bodies in Mach 6 flow. Using temperature sensitive paint (TSP) and three complex geometries commonly used in the hypersonic community, experimental tests were conducted inside the Mach 6 tunnel to capture the temperature contours and some pressure data for these geometries at various angles of attack. These results were then compared to numerical analyses conducted using the panel code CBAero, the Euler code Cart3D, the coupled Euler/Boundary layer solver UNLATCH, and Navier-Stokes solutions from FUN3D. Due to the experiments in the tunnel never reaching steady state since paint adherence was affected after about 10 seconds in the high-speed flow, the comparison to steady numerical analysis proved difficult. As a result, the capabilities of the Mach 6 tunnel, in terms of having a quantifiable measure between the experimental and numerical temperature distributions, could not be assessed and instead general qualitative comparisons were made.

Nomenclature

\( a \) Speed of sound
\( \alpha \) or \( AOA \) Angle of attack
\( C_D \) Drag coefficient
\( C_L \) Lift coefficient
\( C_M \) Pitching moment coefficient
\( C_p \) Pressure coefficient
\( M \) Mach number
\( T \) Temperature [Kelvin]
\( \rho \) Density

---

* Master’s Candidate, Dept. of Mech. and Aerospace Engineering, sagermand1@udayton.edu, Student Member AIAA
† Aerospace Engineer, Air Force Research Laboratory, Nastassja.Dasque.1@us.af.mil
‡ Associate Professor, Dept. of Mech. and Aerospace Engineering, Markus.Rumpfkeil@udayton.edu, Senior Member AIAA
§ Aerospace Engineer, Air Force Research Laboratory, Barry.Hellman@us.af.mil, Senior Member AIAA

Cleared for Public Release, Distribution Unlimited: 88ABW-2016-6213

1 of 22

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.
I. Introduction

The ability to predict and test surface temperatures at hypersonic conditions in the early design stages can have a significant impact on the success of a design. This research aims to gain confidence in the ability to obtain aero-thermodynamic data from the Mach 6 High Reynolds Number Facility at Wright-Patterson Air Force Base (WPAFB). An initial goal set for the effort includes the analysis of three geometries: the RFSG, a GHV, and the HIFiRE-1 payload geometry. For this effort, confidence is postured through the comparison between TSP temperature distributions and pressure data from wind tunnel experiments and numerical analysis.

All experimental testing for this research was conducted in the Mach 6 High Reynolds Number Facility at WPAFB seen in Figure 1. The test section conditions for the Mach 6 tunnel are given in Table 1.

![Mach 6 High Reynolds Number Wind Tunnel at Wright-Patterson Air Force Base.](image)

Table 1. Mach 6 Wind Tunnel Testing Conditions.

<table>
<thead>
<tr>
<th>Test Section</th>
<th>12 in. diameter open jet, 17 to 28 inches long</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach Number</td>
<td>6.0 (5.85 measured during facility calibration)</td>
</tr>
<tr>
<td>Temperature Range</td>
<td>440 to 640 degrees F</td>
</tr>
<tr>
<td>Reynolds Number Range</td>
<td>10 to 30 million/ft.</td>
</tr>
<tr>
<td>Run Time</td>
<td>Continuous 5 minutes at 2000 psi</td>
</tr>
<tr>
<td>Density Altitude</td>
<td>Simulates flight from 30,000 ft. to 60,000 ft.</td>
</tr>
<tr>
<td>Stagnation Temperature Range</td>
<td>900 to 1100 Rankine</td>
</tr>
<tr>
<td>Stagnation Pressure Range</td>
<td>700 to 2100 psia - exhausting to atmosphere</td>
</tr>
</tbody>
</table>
The outline of this paper is as follows. Section II presents the three employed geometries and gives a quick overview of the performed computational analyses. Section III presents inviscid results for the RFSG and GHV models whereas Section IV shows viscous results for a coupled Euler/Boundary layer solver as well as a RANS solver for the HIFiRE-1 geometry. Section V shows the experimental results obtained in the Mach 6 tunnel for all three geometries. Finally, in Section VI a comparison between experimental and numerical results for the HIFiRE-1 and RFSG model are shown and Section VII concludes this paper.

II. Geometries and Computational Analysis Considerations

The three geometries chosen for benchmarking are the RFSG reusable booster system, the generic hypersonic vehicle (GHV), and the HIFiRE-1 all shown in Figure 2. Once again, the goal of this research is to assess the aero-thermal capabilities of the Mach 6 wind tunnel for complex geometries. The comparisons can give confidence in a new capability of the wind tunnel to measure wall temperatures at Mach 6. Such measurements can also give insight into the thermal response of bodies in Mach 6 flow, which in turn can inform subsequent structural analyses of future design iterations. If performed early in the design cycle, insights from the measurements may be able to guide designers to avoid unacceptable outer moldline (OML) decisions that can hinder success of the design later in the design process.

II.A. RFSG

The Reference Flight System model G (RFSG) was developed under the Air Force Research Laboratory (AFRL) Future responsive Access to Space Technologies (FAST) program. This vehicle design was developed based on lessons learned from the FAST program’s three ground technology development efforts. The vehicle is a reusable first stage which would be paired with an expendable upper stage to provide responsive and affordable space access. This model was chosen for this research because of the extensive aerodynamic database compiled for this model as well as its unique geometric features such as the wing-body combination and wing tip vertical fins. An instrumented drawing of the RFSG geometry as well as the wind tunnel model itself can be seen in Figures 3 and 4, respectively. As seen in Figure 4, the paint adhesion on the winglets was severely degraded after some time in the high-speed flow. This was deemed acceptable since the camera mounted to capture the change in lumination of the paint was above the model and the leading edge of the winglets was not captured.

II.B. Generic Hypersonic Vehicle

In order to foster collaboration between the government and other hypersonic stakeholders, AFRL created a family of Generic Hypersonic Vehicles (GHV) that are publicly released. This allows for data to be more easily disseminated within the hypersonic community in order to advance the field. The GHV is a slender, low aspect ratio wing-body configuration enclosing a supersonic combustion ramjet (SCRAMjet) engine with an inward-turning inlet. The vehicle is conceptually designed to have a metallic structure with a thermal protection system capable to withstand Mach 6 cruise conditions. The model was chosen for its complexity as well as its ease of distribution within the community. The GHV model used for this research has an
added stingmount to allow for compatibility with the sting available in the Mach 6 tunnel. The stingmount adaptation to the GHV was incorporated in the computational modeling for this research. The top and side views of the model with instrumentation are shown in Figure 5. The wind tunnel model mounted in the Mach 6 tunnel can be seen in Figure 6 with the TSP paint applied.

II.C. HIFiRE-1

The Hypersonic International Flight Research Experimentation (HIFiRE) program is a bi-lateral hypersonic flight test program between AFRL and the Australian Defense Science and Technology Group (DTSG),
Figure 5. GHV wind tunnel model with instrumentation.

Figure 6. GHV wind tunnel model.
formerly the Australian Defense Science and Technology Organization (DSTO). HIFiRE Flight 1 took place on March 22, 2010 to measure boundary layer transition and shock boundary layer interaction at Mach 7 flight conditions. Figure 7 exhibits the dimensioned HIFiRE-1 wind tunnel model to be used in this research. The actual wind tunnel model used in testing can be seen in Figure 8 mounted in the Mach 6 tunnel with TSP paint applied.

![Figure 7. HIFiRE-1 wind tunnel model with instrumentation.](image)

![Figure 8. HIFiRE-1 wind tunnel model.](image)

To gain confidence in the experimental results obtained from the Mach 6 wind tunnel, four different computational analysis tools were employed: namely the Configuration Based Aerodynamics (CBAero) tool set, the Euler solver Cart3D, Cart3D coupled with the Unstructured Langley Approximate Three-Dimensional Convective Heating (UNLATCH) code and the Fully-Unstructured Navier-Stokes (FUN3D) solver.
flow solver. All analysis tools were developed by the National Aeronautics and Space Administration (NASA). Prior to reporting results from any of these codes grid convergence studies were conducted on computational meshes built specifically for each of the analysis tools. Final meshing strategies were chosen from the grid convergence studies that incurred acceptable discretization errors in the integrated quantities of interest (lift and drag coefficients, $C_L$ and $C_D$) derived from flow solutions. Once appropriate meshes were chosen for each tool, attempts were made to obtain surface wall temperatures as well as pressure distributions for each geometry. Computational setup and results for all three models using the low-fidelity CBAero software package have already been reported and will thus not be repeated here. However, details about the other computational methods and their results for the different geometries will be given in the next two sections.

III. Inviscid Computational Results

Medium-fidelity computational flow simulations were performed using Cart3D, a software suite employing automated and error-adapted Cartesian volume mesh generation to solve the Euler flow equations. While monitoring a user-specified functional, such as lift, drag, or pitching moment, Cart3D uses “adjoint-weighted” error estimation to cluster nodes in regions of the flow-field where error in the flow solution degrades the accuracy of the chosen functional. In this effort, the value of pitching moment coefficient, $C_M$, was monitored to drive adjoint-weighted mesh refinement for the off-body flow-field for the HIFiRE-1 payload. A combined functional $0.6C_D + 0.4C_L$ was used to drive the adaptations for the RFSG and GHV geometries. Using the mesh adaptation capability of Cart3D, grid convergence studies were already performed for the RFSG, GHV, and HIFiRE-1 models in previous work. Thus, only the grid converged results for the RFSG and GHV will be briefly discussed in the next two subsections.

III.A. RFSG

Grid convergence is reached with about one million cells, corresponding to roughly seven to nine adaptation cycles for initial meshes with about 5,500 cells. A more in depth analysis for the RFSG geometry using Cart3D, including surface and volume mesh structure and a grid convergence study can be found in Sagerman et al. Stagnation regions can be readily seen at the nose and wing leading edges in the surface $C_p$ contours while strong flow interactions are apparent near the nose in the Mach contour, see Figures 9 and 10, respectively.

![RFSG Cp contours](image)

Figure 9. RFSG $C_p$ contours.

A comparison between Cart3D and CBAero results is performed next which can give confidence that both analysis codes are producing acceptable results. For comparison, a centerline cut of the RFSG was taken to extract the pressure coefficient value found along the cut for flow conditions at Mach 5.85 and angle-of-attack values of 0, 5, and 10 degrees. Very acceptable agreement between upper and lower surface $C_p$ distributions from Cart3D and CBAero along the centerline can be seen in Figure 11. The cut-plane itself is also shown in the same figure.
\( M = 6, \alpha = 0 \) degrees  \( M = 6, \alpha = 5 \) degrees  \( M = 6, \alpha = 10 \) degrees

Figure 10. RFSG Mach contours.

\( \alpha = 0 \)

\( \alpha = 5 \)

\( \alpha = 10 \)

Figure 11. CART3D vs. CBAero for RFSG along y-cutplane also shown.
III.B. GHV

The conclusion from the GHV grid convergence study was that grid-converged integrated quantities are reached when at least seven adaptation cycles are used on an initial Cartesian volume grid with around 5400 cells.\textsuperscript{6} Resulting flow features are illustrated for $C_p$ and Mach contours, in Figures 12 and 13, respectively. Again, a comparison is made between Cart3D and CBAero for a $C_p$ distribution along a cutplane of the geometry. A cut was made along the port (left) wing of the GHV (see Figure 14), and the $C_p$ distribution was extracted for 5 and 10 degrees $\alpha$ as shown in Figure 15. There is discrepancy between the distributions from both analysis codes. However, trends are quite similar. At these higher angles of attack, the flow can be highly separated, which is not modeled by either code. Cart3D can capture three-dimensional effects, but cannot accurately model separation since separation is a viscous phenomenon. CBAero has no ability to model three-dimensional flow effects or even communicate flow information from one point to the next. Therefore, even though there is discrepancy in the two distributions, there is still confidence that both codes exhibit the proper trends.

![GHV Cp contours](image1)

$M = 6, \alpha = 0$ degrees  \hspace{1cm} $M = 6, \alpha = 5$ degrees  \hspace{1cm} $M = 6, \alpha = 10$ degrees

Figure 12. GHV $C_p$ contours.

![GHV Mach contours](image2)

$M = 6, \alpha = 0$ degrees  \hspace{1cm} $M = 6, \alpha = 5$ degrees  \hspace{1cm} $M = 6, \alpha = 10$ degrees

Figure 13. GHV Mach contours.

![GHV cutplane along port side wing](image3)

Figure 14. GHV cutplane along port side wing.
The ability to predict convective heating over a three-dimensional body at hypersonic speeds is very important in the early stages of hypersonic vehicle design. These simulations cannot be performed accurately with an inviscid flow assumption and two different approaches utilized in this work to account for viscous effects as well as their results for the HIFiRE-1 geometry are discussed in the next two subsections.

### IV. Viscous Computational Results

The UNLATCH (Unstructured Langley Approximate Three-Dimensional Convective Heating) engineering code can be used to calculate the heating rates on general three-dimensional bodies using inviscid flow field results from an unstructured flow solver, such as Cart3D. This is done by reducing the three-dimensional boundary layer equations to the same form as the axisymmetric boundary layer equations. UNLATCH will be used in conjunction with the inviscid flow field results from Cart3D to approximate the surface temperatures on the body of the HIFiRE-1 geometry. The cases shown in Table 2 were run using UNLATCH in order to approximate the viscous effects of the boundary layer on the body with a constant wall temperature of 300 K. These are the same conditions as the ones used during the wind tunnel experiments except for the constant wall temperature.

#### IV.A. UNLATCH Boundary Layer Solver

There was considerable difficulty in reaching converged flow solutions from Cart3D that could be successfully input into UNLATCH. Stagnation regions and areas of high flow interaction must be resolved to a significant degree in order for UNLATCH to begin its viscous approximation calculations. As a result, a very large number of trade-offs in Cart3D’s adaptation strategies were attempted in order to find a combination of adaptation strategy and resolution that would be acceptable for UNLATCH. A detailed overview of the grid generation process for the volumes meshes used for the following UNLATCH analysis can be found in Sagerman et al. In general, flow residuals were only reduced by approximately two orders of magnitude. Although the residuals are higher than usually preferred in CFD studies (desiring reductions of at least four orders of magnitude for hypersonic flows), the integrated quantities do not appear to suffer from this. In fact, the monitored coefficients reach values that experience negligible changes from the eighth to the ninth adaptation cycle for initial volume grids with cell counts greater than roughly 40,000 cells.

Resulting flow features for the first case in Table 2 are shown in Figures 16 and 17, respectively.

#### Table 2. UNLATCH Run Conditions.

<table>
<thead>
<tr>
<th>Case</th>
<th>$V_\infty \ [\text{m/s}]$</th>
<th>$P_\infty \ [\text{N/m}^2]$</th>
<th>$T_\infty \ [\text{K}]$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>936</td>
<td>3570</td>
<td>63.7</td>
</tr>
<tr>
<td>2</td>
<td>986</td>
<td>3570</td>
<td>70.8</td>
</tr>
<tr>
<td>3</td>
<td>936</td>
<td>10200</td>
<td>63.7</td>
</tr>
<tr>
<td>4</td>
<td>986</td>
<td>10200</td>
<td>70.8</td>
</tr>
</tbody>
</table>
The heating rate contours predicted by UNLATCH for the HIFiRE-1 geometry are shown in Figure 18. Unfortunately, due to a bug in the UNLATCH source code, output of surface wall temperatures from UNLATCH is currently unavailable.
Figure 18. UNLATCH solution for surface heating rates $[\frac{W}{m^2}]$ for HIFiRE-1 Geometry.
IV.B. FUN3D RANS Solver

FUN3D is an unstructured finite-volume viscous flow solver.\textsuperscript{11} For the HIFiRE-1 geometry, three unstructured meshes were made using Numeca Hexpress, and the finest mesh is shown in Figure 19. The freestream conditions from Case 1 in Table 2 were used to match the experimental testing of the model leading to a Reynolds number of roughly 43 million/m.

![Figure 19. HIFiRE-1 volume grid for FUN3D, Fine Mesh, $N_{\text{cells}} = 21,815,736$.](image)

No mesh adaptation was used for the various angles of attack. The increasing clustering of cells in the boundary layer of the various meshes can be seen in Figure 20.

![Figure 20. HIFiRE-1 volume grid for FUN3D around nose.](image)
A typical convergence history of mass, momentum, energy and turbulence working variable for the Spalart-Allmaras model exhibiting good convergence is shown in Figure 21.

Figure 21. HIFiRE-1 FUN3D Convergence History for Fine mesh.

The integrated quantities of $C_L$ and $C_D$ for the various meshes are plotted in Figure 22 showing grid convergence.

Figure 22. HIFiRE-1 FUN3D Grid Convergence Study for $C_L$ and $C_D$.

Some results for the fine mesh are displayed below in Figures 23 – 27. The surface $C_p$ contours can be seen in Figure 23, which show similar trends to that of the inviscid solutions using Cart3D and CBAero with the stagnation locations being at the nose and the high-inclination region.
The off-body Mach contours can be seen in Figure 24, showing the formation of the shock and expansion at positive angles of attack.

Using the density contours displayed in Figure 25, one can infer similarities to the inviscid Cart3D results shown in Sagerman et al.6

The $y$-plus values for the volume mesh in the boundary layer can be seen in Figure 26 to help judging the quality of the mesh. A $y$-plus value on the order of 1 was maintained throughout and was deemed sufficient for initial flow results.

The distribution of heating, identified through the heating rate shown in Figure 27, closely follows that of the results computed with UNLATCH (compare with Figure 18). A more concentrated heating at the stagnation point at the nose and overall higher magnitude for the heating was seen in the UNLATCH cases,
\( M = 5.85, \alpha = 0 \text{ degrees} \) \( M = 5.85, \alpha = 5 \text{ degrees} \) \( M = 5.85, \alpha = 10 \text{ degrees} \)

\textbf{Figure 26.} HIFiRE-1 \( y^- \) plus contours.

\( M = 5.85, \alpha = 0 \text{ degrees} \) \( M = 5.85, \alpha = 5 \text{ degrees} \) \( M = 5.85, \alpha = 10 \text{ degrees} \)

\textbf{Figure 27.} HIFiRE-1 surface heating rates \( \frac{\dot{W}}{m^2} \).

which is expected when comparing an approximation (UNLATCH) to fully viscous solutions.
V. Mach 6 Wind Tunnel Experimental Results

With the Mach 6 wind tunnel being unused for the past 20 years, a resurgence in the need for accurate high speed testing facilities has resulted in the wind tunnel becoming operational once again. A point of interest in hypersonic testing as previously noted, is the surface temperature distribution which can be obtained by using Temperature Sensitive Paint (TSP). The paint is applied evenly as a thin layer on the surface and a Charge Coupled Device (CCD) array can capture the instantaneous change in color of the paint which can then be correlated to local temperatures. The TSP is calibrated by Innovative Scientific Solutions Incorporated (ISSI) for certain temperature ranges with a second-order accurate curve fit applied to the calibration data based on luminosity. The temperature value is a function of the ratio of wind-off to wind-on luminosity.

V.A. HIFiRE-1 Temperature Sensitive Paint Results

The test matrix shown in Table 3 was used in testing the HIFiRE-1 geometry in the Mach 6 wind tunnel. This matrix was first completed without TSP to have clean readings of the pressure taps and thermocouples without the interference of the paint in the instrumentation holes and then repeated with the use of TSP while continuing to take readings from the instrumentation.

<table>
<thead>
<tr>
<th>Total Pressure ($P_0$)</th>
<th>Total Temperature ($T_0$)</th>
<th>Orientation</th>
<th>AOA [Deg]</th>
</tr>
</thead>
<tbody>
<tr>
<td>700</td>
<td>900</td>
<td>Notch Starboard</td>
<td>-10,-5,0,5,10</td>
</tr>
<tr>
<td>700</td>
<td>1000</td>
<td>Notch Starboard</td>
<td>-10,-5,0,5,10</td>
</tr>
<tr>
<td>900</td>
<td>900</td>
<td>Notch Starboard</td>
<td>-10,-5,0,5,10</td>
</tr>
<tr>
<td>900</td>
<td>1000</td>
<td>Notch Starboard</td>
<td>-10,-5,0,5,10</td>
</tr>
</tbody>
</table>

Sample TSP results for the HIFiRE-1 for $P_0 = 700 \text{ psi}$ and $T_0 = 900 \text{ R}$ are shown in Figure 28.

![Figure 28. HIFiRE-1 Temperature Contour.](image-url)
V.B. RFSG Temperature Sensitive Paint Results

For the RFSG model, findings from the testing of the HIFiRE-1 model were used to influence the test set up of the RFSG. First the need to run tests without the paint to obtain clean readings with the instrumentation was found unnecessary and therefore the instrumentation readings were taken with the paint applied. Also, the RFSG was tested at the wind tunnel conditions that resulted in the best paint adhesion and response. Therefore the low pressure and high temperature wind tunnel conditions were chosen for both the RFSG and GHV, shown in Tables 4 and 5, respectively. Due to the RFSG and GHV being asymmetric, the models were also mounted inverted to capture the paint response for both the windward and leeward sides for each angle of attack.

As seen in Figure 29, the temperature contour for the RFSG shows higher temperature at the stagnation points around the leading edge of the wings and nose compared to the HIFiRE-1 model. One can note the circular hole section on the lower surface of the RFSG where epoxy was added to fill in the screw holes connecting the two pieces of the RFSG. The epoxy has different thermal properties than the 17-4 stainless steel of the rest of the model and therefore explains the higher temperature shown due to conduction through the material.

Table 4. Mach 6 Wind Tunnel Testing Matrix for RFSG Geometry.

<table>
<thead>
<tr>
<th>Total Pressure ($P_0$)</th>
<th>Total Temperature ($T_0$)</th>
<th>Orientation</th>
<th>AOA [Deg]</th>
</tr>
</thead>
<tbody>
<tr>
<td>700</td>
<td>1000</td>
<td>Normal</td>
<td>-10,-5,0,5,10</td>
</tr>
<tr>
<td>700</td>
<td>1000</td>
<td>Inverted</td>
<td>-10,-5,0,5,10</td>
</tr>
</tbody>
</table>

Figure 29. RFSG Temperature Contour.
V.C. GHV Temperature Sensitive Paint Results

The experimental test matrix for the GHV model is given in Table 5 where again the low pressure ($P_0 = 700 \text{ psi}$) and the high temperature ($T_0 = 1000 \text{ R}$) wind tunnel conditions were evaluated. The temperature contour from the TSP is shown in Figure 30 where a high temperature was seen at the inlet and along the leading edge of the wing and vertical fins. Due to high conduction through the model, especially through the very thin wings, determining where the heat was coming from was very difficult once the model was in the flow for longer than a few seconds.

![Figure 30. GHV Temperature Contour.](image)

**Table 5. Mach 6 Wind Tunnel Testing Matrix for GHV Geometry.**

<table>
<thead>
<tr>
<th>Total Pressure ($P_0$)</th>
<th>Total Temperature ($T_0$)</th>
<th>Orientation</th>
<th>AOA [Deg]</th>
</tr>
</thead>
<tbody>
<tr>
<td>700</td>
<td>1000</td>
<td>Normal</td>
<td>-10,-8,-5,0,5,8,10</td>
</tr>
<tr>
<td>700</td>
<td>1000</td>
<td>Inverted</td>
<td>-10,-8,-5,0,5,8,10</td>
</tr>
</tbody>
</table>
VI. Comparisons between Experimental and Computational Pressure Results

The comparison for the pressure coefficient values from the wind tunnel experiments for the HIFiRE-1 model and the various computational results are shown in Figure 31. When the notch is at 90 degrees starboard, the instrumentation is directly in the flow direction, resulting in higher pressure readings. The trends for pressure tap 1 and 2 match well among the experimental and various computational values. At pressure tap 3, shock induced boundary layer separation causes a drop in surface pressure at the leeward 5 degree case, which, as expected, is not captured in the inviscid CART3D solution. The panel method, CBAero, with viscous approximations accurately predicts the drop in pressure at 5 degrees but does not predict the pressure recovery at 10 degrees when compared to the the experimental results. FUN3D seems to predict the experimental trends the best. The results for $P_0 = 2000 \text{ psi}$ and $T_0 = 1000 \text{ R (HT/HP)}$ as well as $P_0 = 700 \text{ psi}$ and $T_0 = 900 \text{ R (LT/LP)}$ are shown for comparisons since they represent the extreme ends of the test matrix.

![Figure 31. Comparison between experimental and computational $C_p$ results for HIFiRE-1 geometry.](image)

The experimental vs. numerical results for the RFSG’s eight pressure ports are shown in Figure 32. The trends between the experimental value for $C_p$ and the numerical results from Cart3D and CBAero seem to match at least in overall trend across the angle of attack sweep.

![Figure 32.](image)
Figure 32. Comparison between experimental and computational $C_p$ results for RFSG geometry.
VII. Conclusion

Although one-to-one comparisons of experimental and computational surface temperature distributions at Mach 6 have not been accomplished, TSP data was obtained from the Mach 6 High Reynolds Number Facility at Wright-Patterson Air Force Base for the HIFiRE-1 payload geometry, the RFSG model and the GHV stingmount vehicle at Mach 5.85. The mere demonstration of TSP capability at Mach 6 is a significant step forward for experimental hypersonic vehicle design. Such a capability can lead to augmented design studies in the future such that insight into the thermal response of a geometry can be realized much earlier in the design process for hypersonic vehicles. Furthermore, though output temperature distributions could not yet be obtained from the UNLATCH boundary layer solver, the solver itself accepted flow solutions from Cart3D using a unique mesh adaptation strategy developed specifically for this effort. Once we can demonstrate valid wall temperature results from UNLATCH, there are powerful implications that can dramatically influence hypersonic vehicle design. The most notable implication is that the cost of reaching a wall temperature solution is effectively reduced to the cost of a single Euler flow solution. Breaching the barrier of the wall temperature problem of complex geometries in hypersonic flow using a boundary layer approximation to Euler solutions, through the use of Cart3D and UNLATCH, provides strong motivation to continue such work and to identify the problems slowing the progress in the application and implementation of UNLATCH.

The difficulty found in this research was the one-to-one comparison of the surface temperature contour, for a set wind tunnel condition, between the numerical and experimental analysis with TSP. Since the wind tunnel testing could not be done to steady state, quantifying the magnitude for the temperature on the model from computational analysis proved difficult in that the conduction into the model would have to be incorporated into the simulation. This would lead to the need for modeling the fluid-structure interaction which was outside the scope of this project. In terms of recommendations, in order to test the capability of the Mach 6 tunnel for obtaining aero-thermal data, the models should have been constructed as thermal models where the thinness of the wings did not result in almost immediate conduction. This condition makes it difficult to determine where the heat, shown in the rise of luminosity of the paint, came from. Also with the way the models were built, the heat transfer coefficient could not be obtained and would have been a better comparison for the numerical analysis. Although there were some shortcomings, the fact that TSP contours were obtained for all three experimental models and numerical results of varying fidelity were computed to quantify the abilities of the tunnel are hopefully useful to the hypersonic community.

References

1Hayes, J. and Altman, S., “AFRL/RQ Mach 3 and Mach 6 High Re Facility,” AIAA Dayton-Cincinnati Section Lunch and Learn.