Performance Improvements in Boeing/AFOSR Mach 6 Quiet Wind Tunnel Based on CFD Predictions

Hadassah Naiman*
Doyle D. Knight**
Selin Aradag†
Thomas J. Juliano‡
Steven P. Schneider§

*Rutgers – The State University of New Jersey
Piscataway, NJ 08854 USA
hnaiman@eden.rutgers.edu

**Rutgers – The State University of New Jersey
Piscataway, NJ 08854 USA
ddknight@rci.rutgers.edu

†US Air Force Academy
Colorado Springs, CO 80840 USA
selin.aradag.ctr@usafa.af.mil

‡Purdue University
West Lafayette, IN 47907 USA
tjuliano@purdue.edu

§Purdue University
West Lafayette, IN 47907 USA
steves@purdue.edu

Abstract

Computations have been performed on the bleed slot lip and the test section of the Boeing/AFOSR Mach 6 Quiet Wind Tunnel. Separation bubbles on the bleed lip and associated fluctuations induced near the bleed lip were identified as the most likely cause of early transition of the nozzle wall boundary layer, resulting in a noisy test section. The existence of separation bubbles was predicted with high resolution CFD simulations, and a new geometry was designed to eliminate these bubbles. The new geometry was implemented and the maximum quiet stagnation pressure of the tunnel improved from 8 psia to 150 psia. Computations were also run on the test section to determine if expanding the diameter would allow starting larger blunt models with stronger shocks. Several cone sizes were considered, with half-angles ranging from 15° to 75°, and the resulting flowfield was examined to see what effect the shocks and shear layers would have on the quiet test section flow.

Key words: laminar, wind tunnel, bleed slot, separation bubble, unstart, test section
**Introduction**

One of the major challenges in hypersonic flow research is the accurate prediction of transition. The location and extent of laminar-turbulent transition is a critical parameter in hypersonic vehicle design. The transition location affects estimates of aerodynamic heating, skin friction drag and other boundary layer properties. Transition experiments have been carried out in conventional ground testing facilities for decades. However, most of the experimental data obtained from these facilities are contaminated by the high levels of noise that radiate from the turbulent boundary layers normally present on the nozzle walls. The effects of this acoustic noise are profound. These high noise levels can cause transition to occur an order of magnitude earlier than in flight [1]. Not only is the location of transition affected, but the parametric trends for transition can also be dramatically different from those in flight [2].

Quiet flow wind tunnels have been developed to simulate hypersonic flow in flight, where the noise levels are very low. A quiet wind tunnel is characterized by laminar boundary layers in the test section. A review of the various efforts worldwide to develop quiet tunnels is provided in Ref. [3]. A Mach-3.5 tunnel was the first to be successfully installed at NASA Langley in the early 80’s. This was followed by a quiet Mach-6 hypersonic facility in the mid-90’s. Unfortunately, this nozzle was removed from service due to a space conflict, and is now being reinstalled at Texas A&M. The Boeing/AFOSR Mach-6 Quiet Tunnel (BAM6QT) at Purdue University was constructed during 1995-2001. It is, at present, the only operational hypersonic quiet tunnel anywhere in the world [4].

![Figure 1. Schematic of the BAM6QT](image)

The BAM6QT is designed as a Ludwieg tube (Figure 1). A Ludwieg tube is a long pipe with a converging-diverging nozzle on the end, from which flow exits into the nozzle, test section and second throat. A diaphragm is placed downstream of the test section. When the diaphragm bursts, an expansion wave travels upstream through the test section into the driver tube. Expansion wave reflections occur for a period of time during which the flow remains quiet. Figure 2 shows the nozzle. The region of useful quiet flow lies between the characteristics marking the onset of uniform flow, and the characteristics marking the upstream boundary of acoustic radiation from the onset of turbulence in the nozzle wall boundary layer. One method to reduce noise is to delay boundary layer transition using a bleed slot before the nozzle throat to remove the contraction-wall boundary layer, beginning a fresh, undisturbed boundary layer for the nozzle wall.

Shocks emanating from a model in the test section interact with the boundary layer on the tunnel wall. While disturbances in supersonic flow can only travel downstream, disturbances in the subsonic boundary layer flow can lead to separated flow upstream in the tunnel nozzle [5]. Laminar boundary layers are more likely to separate than turbulent ones, so shock/boundary layer interactions are more likely to affect upstream flow in a quiet tunnel with laminar boundary layers at high Reynolds numbers. Laminar shock/boundary-layer interactions are thus a critical issue for determining the largest possible model that can be started in the quiet tunnel.

This study explores two cases that combine experimental and computational methods, where CFD was utilized to examine various designs in an effort to improve the performance of the BAM6QT. The first case investigates the
effect of bleed lip geometry on the transition point of the nozzle wall boundary layer. The second case considers the effect of a test section expansion on the ability to test larger, blunter models.

Figure 2. Quiet flow region in test section

Bleed Lip Redesign

As of 2005, the BAM6QT had not yet achieved quiet flow for the desired range of stagnation pressures up to 150 psia. Two nozzles were fabricated and tested: the original electroformed nickel nozzle and a surrogate aluminum throat. Early transition of the nozzle wall boundary layer was identified as the most likely cause of the test section noise in the tunnel. Separation bubbles on the bleed lip and associated fluctuations induced near the bleed lip were identified as the most likely source of early transition [6]. The experimental study of Klebanoff and Tidstrom [7] show that the presence of a separation bubble of sufficient size destabilizes the laminar boundary layer downstream of reattachment thereby leading to an earlier transition to turbulence, i.e., the location of transition moves upstream relative to where it would occur without the separation bubble. Moreover, the bleed lip of the electroformed throat of the BAM6QT had a 0.001-inch kink that was not present in the surrogate aluminum throat, and it appears that this kink exacerbated a natural tendency to form a separation bubble near the lip. This separation bubble is possibly highly unsteady, and can lead to early transition downstream [8].

The situation in the hypersonic wind tunnel at Purdue University illustrates the importance of the bleed lip geometry, and the effects of separation bubbles that form around the bleed lip, on the quality of the flow at the test section. The objective of this CFD effort is to demonstrate the effect of separation bubbles on flow structure by numerically investigating the existence of steady and unsteady separation bubbles on the main flow or the bleed flow side of the nozzle lip of the BAM6QT, and to design a new geometry to eliminate or reduce the size of the separation bubbles.

<table>
<thead>
<tr>
<th>Inflow</th>
<th>P&lt;sub&gt;r&lt;/sub&gt;-Riemann subsonic inflow</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bleed slot exit</td>
<td>Forced outflow</td>
</tr>
<tr>
<td>Nozzle exit</td>
<td>Forced outflow</td>
</tr>
<tr>
<td>Solid walls</td>
<td>No slip, adiabatic</td>
</tr>
<tr>
<td>Symmetry Plane</td>
<td>X-axis axisymmetric</td>
</tr>
<tr>
<td>Side walls</td>
<td>Axisymmetric</td>
</tr>
</tbody>
</table>

Table 1: Boundary conditions for bleed slot simulation

Steady and time-accurate computations were performed for both the original geometry and the new designs using GASPex version 4.1.2 [9]. The laminar, compressible Navier Stokes equations were solved. For the modeling of
inviscid fluxes, the third order Roe's scheme with Harten correction was used. The min-mod limiter was employed as a flux-limiter. The boundary conditions are shown in Table 1.

An implicit dual time stepping method was utilized for the time-accurate computations. The time for the flow to go from the bleed lip to the end of the computational domain was calculated to be 0.28 milliseconds. The velocities used for calculating the average velocity are the velocities in the steady state solution. The total simulation time was taken to be four times the time necessary for the flow to go from the bleed lip to the exit of the computational domain, corresponding to 1.1 milliseconds. The values obtained from the steady state solution were used for all the flow parameters as an initial condition for the time accurate computations.

Analysis of the Existing Bleed Lip

A close-up view of the geometry around the bleed lip is shown in Figure 3. The grid used in the computation was generated with GridPro [10] and had a minimum grid spacing around the bleed lip of 0.001 mm with a stretching parameter of 1.105. The total number of grid points was 192,184.

Separation bubbles exist on both the main flow and the bleed flow sides of the bleed lip for a stagnation pressure of 150 psi, according to steady computational results. The lengths of the separation bubbles on the main and bleed flow sides of the bleed lip are 1.15 mm and 2.2 mm, respectively. Streamlines superimposed on the Mach number contours for the flow at 150 psi are shown are Figure 4. The magnified plot around the bleed lip is shown in Figure 5.

Figure 3. Bleed slot geometry (not to scale)

Figure 4. Mach contours for unmodified geometry

Figure 5. Streamlines around bleed slot at 150 psi
The time-dependent computations at 8 psi showed no unsteadiness in the flow. For the flow at 14 psi, unsteadiness was observed in the flow around the separation bubble only on the bleed flow side of the bleed lip. The wall shear stress values were calculated for the points around the bleed slot lip. The shear stress at the wall is defined as:

\[ \tau_n = \mu_v (\overrightarrow{s} \cdot \overrightarrow{v}) / \Delta n \]  

where \( \mu_v \) is the viscosity, \( \overrightarrow{s} \) is the vector parallel to the surface, \( \overrightarrow{v} \) is the velocity vector and \( \Delta n \) is distance between the wall and the next grid point.

The shear stress variation for the upper and lower bleed lip surfaces is shown in Figure 6 for several time values. Unsteadiness in the shear stress can be seen along the upper surface of the lip at 14 psi. The first location where the shear stress is negative on the upper surface (Figure 6a) corresponds to the separation bubble on the bleed flow side of the lip. The second location where the shear stress has negative values corresponds to the recirculation region at the corner of the bleed lip where the flow is turned upward (Figure 3). The unsteadiness of the first bubble may cause the stagnation point to fluctuate, which can affect the boundary layer on the main flow side. Figure 6b confirms the existence of a separation bubble on the lower surface of the lip. This bubble is assumed to be responsible for the early transition of the boundary layer.

![Figure 6. Shear stress variation for the upper (a) and lower (b) bleed lip surfaces at 14 psi](image)

Separation bubbles can induce earlier transition to turbulent flow by destabilizing the boundary layer. Steady and time-accurate simulation results at several stagnation pressures reveal separation bubbles of varying size on both the main flow and bleed flow sides of the bleed lip of the original electroformed nozzle for all stagnation pressures tested.

**Bleed Lip Modification**

Several modifications were made to the geometry by remachining the bleed lip over an axial region that covers less than 0.1 inches (2.54 mm). An adverse pressure gradient is present just aft of the blunt nose on a flat plate in uniform flow. As a semi-elliptical nose becomes more slender, this gradient is reduced [11]. The basic idea in modifying the bleed lip is to make the lip more slender to eliminate the separation bubbles. Several different geometries were designed for the bleed lip of the tunnel [12, 13]. The computational results obtained with the most successful geometry will be summarized.

The original and new geometries are displayed in Figure 7. To obtain the new geometry, the nozzle coordinates after point (-22.9826817, 17.4838357) mm were not altered. The coordinates of the upper portion of the bleed lip were not changed after point (-22.5, 18.44801907) mm. Three arbitrary points were put between these two unaltered,
original geometry points and four different cubic splines were fit to these five points to create the new geometry. Also, in order to remove the scratches on the existing lip surface and to eliminate the offset of 0.002 inches between the aluminum surrogate nozzle and the original geometry, the tip point of the lip was moved 0.005 inches inside.

Steady and unsteady computations were performed on the new geometry at a stagnation temperature of 433 K for three different pressures: 50, 150 and 300 psi. The Mach number contours for the steady simulations of the new geometry at 300 psi are shown in Figure 8. The results for 50 and 150 psi stagnation pressures are similar to those at 300 psi. The separation bubbles on both the lower and upper parts of the bleed lip are eliminated up to a stagnation pressure of 300 psi.

The wall shear stress plots are shown for the upper and lower sides of the stagnation point at a stagnation pressure of 150 psi in Figure 9. There is no unsteadiness in wall shear stress at 150 psi. The shear stress at the location of the unsteady separation bubble, which previously existed on the upper side of the bleed lip, is high and positive as seen in Figure 9a. The separation bubble on the lower surface (main flow side) of the bleed lip has also been eliminated.

Steady and unsteady computations with this nozzle lip geometry show that the separation bubbles on both the main and bleed flow sides of the nozzle lip are eliminated with this new geometry up to a stagnation pressure of 300 psi. The separation bubbles, which cause earlier transition in the test section of a hypersonic wind tunnel, can be eliminated with a slight change in the geometry of the bleed lip.
Test Section Expansion

Although slender vehicles are the primary concern in many transition experiments, blunt vehicles are also affected by transition [14]. Computations have been performed for the test section of the BAM6QT [15] to determine if expanding this section would allow larger blunt objects to be tested. The shock waves from the nose of the test cone and bow shocks from blunt models interact with the nozzle wall boundary layer. This shock/boundary layer interaction could cause disturbances to propagate upstream and disturb the uniform, quiet flow. The further away from the model this interaction occurs, the less likely are the disturbances to reach the area upstream of the test cone. Furthermore, when strong bow shocks from blunt models impinge on a shear layer before reaching the wall boundary layer, the disturbances from the shock/shear layer interaction appear less likely to propagate upstream and separate the nozzle wall boundary layer [16]. On the other hand, the shear layer generated by expanding the test section may grow and effectively reduce the useful test cross-section.

The purpose of this analysis is to predict what sorts of shocks and shear layers would result from such an expansion, and if this new design would allow larger blunt models to be tested. Several cone sizes at zero angle of attack with a 5.5-inch base diameter are considered, with half-angles ranging from 15° to 75°, in order to determine how large a cone could fit in the test section before the tunnel unstarts.

<table>
<thead>
<tr>
<th>Inflow</th>
<th>User specified pointwise data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outflow</td>
<td>Forced outflow</td>
</tr>
<tr>
<td>Solid wall</td>
<td>No slip, adiabatic</td>
</tr>
<tr>
<td>Cone surface</td>
<td>No slip, adiabatic</td>
</tr>
<tr>
<td>Centerline</td>
<td>Symmetry Plane</td>
</tr>
</tbody>
</table>

Table 2. Boundary conditions for the test section with cones

The grids were generated with GridPro [10] and contained approximately 39,000 points. Grid clustering was performed with a stretching parameter of 1.105 and a first cell height of $10^{-5}$ ft along the wall and cone surfaces in order to resolve the boundary layers. The inflow boundary condition was a user-specified flow with a boundary layer thickness corresponding to a laminar boundary layer that had been developing since the bleed lip near the tunnel throat, and a uniform Mach 6.15 freestream flow. The boundary layer profile contained 40 points and was calculated with EDDYBL [17] assuming a stagnation pressure and temperature of 90 psi and 433 K. The boundary conditions are listed in Table 2. The initial conditions were chosen such that the flow was accelerated from rest by an incoming shock wave. These conditions do not simulate the tunnel startup process, in which a ruptured diaphragm generates an expansion fan. Rather, a conventional “impulsive” start is used to arrive at a steady solution. Numerical simulations were performed using GASPex Version 4.1.0 [8], with an implicit dual time stepping algorithm with a time step of $10^{-6}$ sec. Each case was run until unstart occurred or until it appeared steady (10-20 ms).

In order to distinguish between shocks, shear layers and expansion waves, it is helpful to examine various contour plots. Entropy changes across a shear layer but not across an expansion, and there is a noticeable change across a normal shock. Pressure changes across expansion fans and shocks, but not across shear layers. Mach number changes across all three. The structure of the flowfield is illustrated in Figures 10-12 with contour plots superimposed on numerical schlieren images for the 15° half-angle cone at 20 ms.

The flow is unable to make the sharp 45° turn so a shear layer is formed and a recirculation region exists in the extended region. A series of shocks gradually turn the flow at the 10° compression corner so that it exits normal to the outflow boundary. When the flow reaches the cone, a shock forms, which interacts first with the shear layer and then with the boundary layer along the wall. When it hits the wall it reflects off, providing much of the compression needed for the flow to turn the 10° corner. At the base corners of the cone there is a small expansion fan which
causes the flow to expand to $M \sim 8$ (red regions in Figure 12). The recirculating flow behind the cone gives rise to the barrel shock, which slows the flow to stagnation conditions at the cone base.

Figure 10. Entropy contours and numerical schlieren for 15° half-angle cone at 20 ms

Figure 11. Pressure (lb/ft$^2$) contours and numerical schlieren for 15° half-angle cone at 20 ms

Figure 12. Mach number contours and numerical schlieren for 15° half-angle cone at 20 ms

The region of interest is that upstream of the cone. These computations do not reveal any disturbances to the nozzle wall boundary layer in this region. All the shock/shear layer/ boundary layer interactions affect the flow downstream of the cone, but do not appear to cause the boundary layer upstream to become unsteady. Figures 13 and 14 show a blunter 75° half-angle cone with a 5.5-inch base. A separation bubble forms at the expansion corner and grows until it reaches the inflow boundary, unstarting the tunnel.

* The range of pressures in this plot is limited to pressures below 25 lb/ft$^2$ for the purpose of distinguishing the pressure variation in the wake of the cone. The pressures in the red area downstream of the compression corner exceed 70 lb/ft$^2$ and a different choice of contours would illuminate variation in this region as well.
Figure 13. Schlieren snapshot of 75° half-angle cone unstarting the tunnel

Figure 14. U Velocity (ft/sec) contours with streamlines of 75° half-angle cone unstarting the tunnel

Conclusion

Two examples have been presented of design modifications based on CFD analysis. The existence of steady and unsteady separation bubbles on the bleed slot lip of the BAM6QT were investigated numerically. A new geometry was designed to eliminate these separation bubbles. The bleed lip of the BAM6QT was remachined in 2006 and less than 0.010 inches of material was removed over a small region of the tip. Quiet flow was subsequently achieved to a freestream unit Reynolds numbers of more than 3.5 x 10^6/ft. The performance of the tunnel has improved to a maximum of $p_t < 153$ psia.

Nozzle wall boundary layer separation is more likely for laminar boundary layers. Bow shocks from the models interact with the wall boundary layer and can cause the flow upstream to separate. Consequently, there is an interest in expanding the diameter of the test section so that this separation does not reach the upstream quiet flow region. CFD was carried out on this section of the BAM6QT, and several cone sizes were examined to see if they would unstart the tunnel. It was found that a 15° half-angle cone fit into the modified section without causing any upstream separation. This is an improvement over the 7° half-angle cone currently used. These are preliminary computations for the expanded test section, which is to be installed over the summer. Investigation into this issue will continue during the fall as the new section is tested.

References


