



AFRL-AFOSR-VA-TR-2022-0392

Boundary Layer Transition (BOLT)- Experiments and Simulations (Low Concave Curvature and Leading-Edge Sweep Effects)

**Reed, Helen
TEXAS ENGINEERING EXPERIMENT STATION COLLEGE STATION
1470 WILLIAM D FITCH PKY
COLLEGE STATION, TX,
US**

**07/15/2022
Final Technical Report**

DISTRIBUTION A: Distribution approved for public release.

Air Force Research Laboratory
Air Force Office of Scientific Research
Arlington, Virginia 22203
Air Force Materiel Command

REPORT DOCUMENTATION PAGE

PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ORGANIZATION.

1. REPORT DATE 20220715	2. REPORT TYPE Final	3. DATES COVERED <table style="width: 100%;"> <tr> <td style="width: 50%;">START DATE 20180215</td> <td style="width: 50%;">END DATE 20210930</td> </tr> </table>		START DATE 20180215	END DATE 20210930	
START DATE 20180215	END DATE 20210930					
4. TITLE AND SUBTITLE Boundary Layer Transition (BOLT)- Experiments and Simulations (Low Concave Curvature and Leading-Edge Sweep Effects)						
5a. CONTRACT NUMBER	5b. GRANT NUMBER FA9550-18-1-0010	5c. PROGRAM ELEMENT NUMBER 61102F				
5d. PROJECT NUMBER	5e. TASK NUMBER	5f. WORK UNIT NUMBER				
6. AUTHOR(S) Helen Reed, Rodney Bowersox						
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) TEXAS ENGINEERING EXPERIMENT STATION COLLEGE STATION 1470 WILLIAM D FITCH PKY COLLEGE STATION, TX US			8. PERFORMING ORGANIZATION REPORT NUMBER			
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) Air Force Office of Scientific Research 875 N. Randolph St. Room 3112 Arlington, VA 22203		10. SPONSOR/MONITOR'S ACRONYM(S) AFRL/AFOSR RTA1	11. SPONSOR/MONITOR'S REPORT NUMBER(S) AFRL-AFOSR-VA-TR-2022-0392			
12. DISTRIBUTION/AVAILABILITY STATEMENT A Distribution Unlimited: PB Public Release						
13. SUPPLEMENTARY NOTES						
14. ABSTRACT The TEES/CUBRC team actively engaged in the BOLT flight test program. The Team participated in BOLT reviews and weekly telecons conducted by Dr. Brad Wheaton (Lead; Applied Physics Lab) and AFRL/AFOSR. 1) The computational TEES team led by Dr. Reed provided numerical stability simulations to help identify instability mechanisms and advise on sensor placement both in the ground tests and on ascent and descent in flight. Using our in-house nonlinear parabolized stability code EPIC, Alexander Moyes (graduated Ph.D. student) characterized the region away from the centerline and swept leading edge in flight and suggested the heat streaks observed behave like stationary crossflow instability. Daniel Mullen (graduated Ph.D. student) conducted further basic-state studies with a focus on verification (including successful comparisons with other codes) and investigated the instabilities on the flight configuration and the wind-tunnel subscale model. Flight off-design conditions of pitch and yaw were also modeled. Mr. Mullen submitted his predictions to the Transition Prediction Challenge. He was a Summer 2019 AFOSR Intern to SpaceX. Andrew Riha (Ph.D. student) also worked with Mr. Scott Berry (NASA Langley Research Center) on the roughness experiments planned for the opposite side on the BOLT II (companion project, FA9550-19-1-0154) flight model. 2) The approach taken by the TEES/CUBRC team led by Dr. Bowersox has been to identify transition instabilities and mechanisms both on and off the surface of the BOLT geometry at hypersonic speeds through ground test and simulation.						
15. SUBJECT TERMS						
16. SECURITY CLASSIFICATION OF: <table style="width: 100%;"> <tr> <td style="width: 33%;">a. REPORT U</td> <td style="width: 33%;">b. ABSTRACT U</td> <td style="width: 33%;">c. THIS PAGE U</td> </tr> </table>		a. REPORT U	b. ABSTRACT U	c. THIS PAGE U	17. LIMITATION OF ABSTRACT UU	18. NUMBER OF PAGES 104
a. REPORT U	b. ABSTRACT U	c. THIS PAGE U				
19a. NAME OF RESPONSIBLE PERSON SARAH POPKIN			19b. PHONE NUMBER (Include area code) 000-0000			

Final Report:
FA9550-18-1-0010

**Boundary Layer Transition (BOLT) - Experiment and Simulation (Concave
Curvature and Leading Edge Sweep Effects)**

Helen L. Reed, Rodney D.W. Bowersox
Texas A&M Engineering Experiment Station (TEES), College Station, Texas
CUBRC, Buffalo, New York

February 15, 2018 – November 30, 2021

ABSTRACT

The TEES/CUBRC team actively engaged in the BOLT flight test program. The Team participated in BOLT reviews and weekly telecoms conducted by Dr. Brad Wheaton (Lead; Applied Physics Lab) and AFRL/AFOSR. **1)** The computational TEES team led by Dr. Reed provided numerical stability simulations to help identify instability mechanisms and advise on sensor placement both in the ground tests and on ascent and descent in flight. Using our in-house nonlinear parabolized stability code EPIC, Alexander Moyes (graduated Ph.D. student) characterized the region away from the centerline and swept leading edge in flight and suggested the heat streaks observed behave like stationary crossflow instability. Daniel Mullen (graduated Ph.D. student) conducted further basic-state studies with a focus on verification (including successful comparisons with other codes) and investigated the instabilities on the flight configuration and the wind-tunnel subscale model. Flight off-design conditions of pitch and yaw were also modeled. Mr. Mullen submitted his predictions to the Transition Prediction Challenge. He was a Summer 2019 AFOSR Intern to SpaceX. Andrew Riha (Ph.D. student) also worked with Mr. Scott Berry (NASA Langley Research Center) on the roughness experiments planned for the opposite side on the BOLT II (companion project, FA9550-19-1-0154) flight model. **2)** The approach taken by the TEES/CUBRC team led by Dr. Bowersox has been to identify transition instabilities and mechanisms both on and off the surface of the BOLT geometry at hypersonic speeds through ground test and simulation. On-surface measurements of the geometry are compared among Texas A&M University, Purdue University, NASA Langley Research Center, and CUBRC hypersonic wind tunnel facilities. The freestream environment within each facility ranges from conventional to quiet (flight-like) to high-enthalpy flow. Measurements on the surface include surface pressure fluctuations measured by PCBs and Kulites as well as the surface heat flux viewed by infrared thermography and TSP and measured directly with Schmidt Boelter gauges. The heat flux results from the quiet wind tunnel facilities were compared with

computations performed by Texas A&M University and the University of Minnesota. The TAMU heat flux data reduction code became the project standard, and is used across all facilities. The focus this past year has been threefold. First, the team continued to analyze the ground test data from the BOLT experimental campaign. Second, the team completed the off-body results and subsequent analyses. Third, the team focused on the development of the BOLT II (companion project, FA9550-19-1-0154) flight test article design with CUBRC, AFRL, AFOSR, NASA and NSROC. Heather Kostak (Ph.D. student) completed a summer internship at APL in 2018. She not only led the TAMU experiments, but she also participated in the experimental campaigns at NASA Langley (20-inch Mach 6 tunnel) and closely collaborated with the AFRL/AFOSR BOLT team.

A. ACCOMPLISHMENTS

1. RESEARCH OBJECTIVES

The project objective was to perform a scientific examination of the transition mechanisms, working with the AFOSR BOLT team, that utilized wind tunnel testing and numerical analyses to characterize transition physics. An integrated research program to achieve the following overarching goals (and BAA tasks) was proposed: Goal 1 – Understand design requirements as defined by AFOSR; Goal 2- Design and analyze a configuration meeting those requirements. In the process, consider the BOLT geometry as provided by AFOSR; Goal 3 – Analyze the configuration under duplicated flight conditions; and Goal 4 (Years 2 and 3) – Study fundamental transition mechanisms and variants.

2 ACCOMPLISHMENTS

A brief synopsis of the accomplishments to date is provided below.

- **Goal 1 Understand design requirements.** The TEES/CUBRC team actively engaged in the BOLT flight test program. The Team participated in the CoDR, PDR, CDR, Science Team Meeting & Hardware Send Off, and the Transition Prediction Challenge as well as other reviews and weekly telecoms.
- **Goal 2 Design and Analyze the BOLT Configuration.** 1) The computational TEES team led by Dr. Reed provided numerical stability simulations to help identify instability mechanisms and advise on sensor placement both in the ground tests and on ascent and descent in flight. Using our in-house nonlinear parabolized stability code EPIC, Alexander Moyes (graduated Ph.D. student) characterized the region away from the centerline and swept leading edge in flight and suggested the heat streaks observed behave like stationary crossflow instability. Daniel Mullen (graduated Ph.D. student) conducted further basic-state studies with a focus on verification (including successful comparisons with other codes) and investigated the instabilities on the flight configuration and the wind-tunnel subscale model. Flight off-design

conditions of pitch and yaw were also modeled. Mr. Mullen submitted his predictions to the Transition Prediction Challenge. 2) The approach taken by the TEES/CUBRC team led by Dr. Bowersox has been to identify transition instabilities and mechanisms both on and off the surface of the BOLT geometry at hypersonic speeds through ground test and simulation. On-surface measurements of the geometry are compared among Texas A&M University, Purdue University, NASA Langley Research Center, and CUBRC hypersonic wind tunnel facilities. The freestream environment within each facility ranges from conventional to quiet (flight-like) to high-enthalpy flow. Measurements on the surface include surface pressure fluctuations measured by PCBs and Kulites as well as the surface heat flux viewed by infrared thermography and TSP and measured directly with Schmidt Boelter gauges. The heat flux results from the quiet wind tunnel facilities were compared with computations performed by Texas A&M University and the University of Minnesota. The focus this past year has been twofold. First, the team continued to analyze the ground test data from the BOLT experimental campaign. Second, the team completed the off-body results and subsequent analyses. Heather Kostak (Ph.D. student) completed a summer internship at APL in 2018. She not only led the TAMU experiments, but she also participated in the experimental campaigns at NASA Langley (20-inch Mach 6 tunnel) and closely collaborated with the AFRL/AFOSR BOLT team.

- **Goal 3 Analyze the configuration under duplicated flight conditions.** 1) The full-scale geometry was tested in the CUBRC LENSII facility at flight duplicated conditions. 2) The TEES computational team has analyzed the stability of flight ascent and descent conditions using our in-house code EPIC on the main test side of the model. Off-design conditions of pitch and yaw have also been modeled.
- **Goal 4 Study fundamental transition mechanisms and variants.** The whole TEES/CUBRC team worked with the AFOSR-APL team to provide analyses and data on the flowfield and instability mechanisms. As a variant, Dr. Bowersox and his team also focused on the development of the BOLT II (companion project, FA9550-19-1-0154) flight test article design with CUBRC, AFRL, AFOSR, NASA and NSROC. Also, as a variant, Dr. Reed and her computational team have also worked with Mr. Scott Berry (NASA Langley Research Center) on the roughness experiments planned for the other side of the flight model for BOLT II (companion project, FA9550-19-1-0154).

3. DISSEMINATION OF RESULTS

- The TEES/CUBRC team actively engaged in the BOLT flight test program, working with the AFOSR-APL team. Our Team participated in the CoDR, PDR, CDR, Science Team Meeting & Hardware Send Off, and the Transition Prediction Challenge as well as other reviews and weekly telecoms, and provided analyses and data on the flowfield and instability mechanisms.
- Our Team has published at peer-reviewed conferences and in journals, and has graduated 2 Ph.D students and 1 Masters student.
- Results from BOLT are feeding forward to the BOLT II program.

B. IMPACTS

- The principal objective of this work was to increase the state-of-the-art in the scientific knowledge base associated with boundary-layer stability and transition mechanisms on low-curvature concave surfaces with swept leading-edges at high Mach number through an integrated numerical and ground test program, so as to understand how these geometries perform in the flight environment. A detailed research approach including extensive wind tunnel testing and modern stability simulation was conducted. New numerical and ground test databases, which are of great importance for future predictions of transition behavior under hypersonic flight conditions, were created, including:
 - 3-D LPSE and NPSE Stability Simulation Databases and Analyses
 - Mach 6 Quiet Tunnel Sub-scale Databases and Analyses
 - Full-scale Ground Test Databases and Analyses
- The TEES/CUBRC team applied these results through active engagement with the AFOSR-APL team in the BOLT flight test program. Our Team participated in the CoDR, PDR, CDR, Science Team Meeting & Hardware Send Off, and the Transition Prediction Challenge as well as other reviews and weekly telecoms, and provided analyses and data on the flowfield and instability mechanisms.
- Moreover, results from this work on BOLT are feeding forward to the BOLT II flight-test program.

C. CHANGES

None to report.

D. TECHNICAL UPDATES: COMPUTATIONAL RESULTS (REED)

1. NOMINAL FLIGHT-TEST CONDITIONS

1.1 Stability Calculations

For all of the primary instability calculations in this work, the linear stability theory (LST) and parabolized stability equation (PSE) code, EPIC (Euonymous Parabolized Instability Code) [1]-[2], is used. EPIC is a well-documented, verified, and validated in-house code to study boundary-layer stability and laminar-to-turbulent transition. It has been used successfully for many flow fields in different speed regimes. It is a basic research code that continues to be developed and its applicability continues to be expanded.

In summary, EPIC is formulated with primitive variables for 3-D compressible flows and uses an orthogonal curvilinear coordinate system. EPIC is currently setup for thermally perfect gases, assumes Sutherland's law for viscosity, and has options for thermal conductivity. This research will utilize the calorically perfect gas assumption and a constant Prandtl number $Pr = 0.72$. It includes a robust shock and boundary-layer height finder. The shock is calculated utilizing the first and second wall-normal derivatives of pressure and the boundary layer is calculated in a variety of ways, depending on the wall-normal profiles of the flow variables. This shock and boundary-layer height finder is utilized to create a high-quality stability grid that can change in the streamwise direction. The shock and boundary-layer heights are directly input into Malik's [3] wall-normal grid clustering scheme, which leads to a physical grid that maps analytically to an orthogonal equally spaced computational grid. All of the original basic-state data is interpolated to this physical grid using a cubic spline interpolation with not-a-knot boundary conditions. By using an orthogonal equally spaced computational grid, a standard finite difference technique is used for all derivatives. Differentiation matrices are formed that contain the coefficients of the finite difference stencils of any order of accuracy. A variety of boundary conditions at the wall and the end of the wall-normal profile can be applied, including Dirichlet and Neumann, as well as a porous wall option. The only limited boundary conditions are the density disturbance at the wall, which utilizes a compatibility condition (global mass continuity is always used), and the wall-normal velocity disturbance at the end of the wall-normal profile of the mean-flow distortion for nonlinear parabolized stability equation (NPSE) calculations (Neumann is always used). The LST solution can be calculated using a full generalized eigenvalue QZ algorithm, the shift-invert Arnoldi algorithm, or a Newton-Raphson technique (this technique is detailed below). The linear parabolized stability equation (LPSE) and NPSE calculations are initialized with an LST eigensolution and a normalization condition is applied to update the marching-direction wavenumber and determine convergence [4]. An additional nonlinear convergence criterion is required for NPSE to ensure the higher order inhomogeneous terms converge. NPSE also includes a relaxation parameter that helps converge the nonlinear terms in the highly nonlinear region. Unique to EPIC, this relaxation parameter features a robust schedule that changes the parameter

based on the required number of iterations at a station. Finally, parallelization, as well as efficient third-party libraries, have been included in important spots to drastically speed up EPIC (MUMPS for LU decomposition).

Since EPIC is a marching code, a marching path is necessary. There are techniques currently to adjust the path direction at each streamwise location to follow the disturbance downstream (group velocity [3],[5]), but it is less efficient than providing an approximate disturbance propagation path. A 3-D path extraction code [6] has been developed to follow various paths only using the basic-state data. Two example path types are straight paths that march straight down a geometry and grid line paths that march along constant spanwise index grid lines. For flow fields with 3-D boundary layers, a streamline is a convenient path as it follows the resultant velocity vector in the flow field. Streamlines are good first approximations to the correct disturbance direction and are straightforward to extract and follow downstream. The only difficulty of using a streamline is deciding where in the wall-normal profile to follow the local velocity vector. The path extraction code has many options, including the height at which 99.5% or 100.1% of the freestream total enthalpy occurs. For stationary crossflow, a more accurate path to follow than a streamline has been developed. The vortex path, or generalized inflection-point path [6]-[7], uses the basic-state flow variables to approximate the direction of the stationary-crossflow development. It is found by using an inflection-point method which locally aligns the marching path such that the generalized inflection point of ρw mass flux occurs where the mass flux equals zero. This path direction has been verified and validated in depth [6]-[7].

Defining ω in the various stability formulations is relatively straightforward. For a given frequency in spatial stability analysis, this disturbance will grow, decay, or do both. It is important to determine amplification of each frequency disturbance independently. However, for α and β , it is much less straight forward and the community is still developing techniques to model β . The disturbance formulation needs to satisfy the conservation laws of kinematic wave theory [8]. Mack [9] discusses that the streamwise and spanwise wavenumbers are not independent, leading to the irrotationality condition. For a flat geometry, a constant spanwise wavelength has been applied to satisfy this condition. On conical geometries, the common technique to satisfy the irrotationality condition is to set constant the total number of waves surrounding a geometry (a constant spanwise wavelength has been applied before, but it is the least physical). This means that as the path develops downstream, the corresponding spanwise wavelength would change based on the local spanwise radius of curvature. This works well for axisymmetric configurations at 0° angle of attack (AoA) and yaw or situations with constant pressure gradients along a span. For more complex flow fields with stationary crossflow, the previous technique has been modified. For a non-axisymmetric configuration or a cone at AoA or yaw, the streamlines (or vortex paths) converge or diverge differently around the azimuth. Various stationary-crossflow experiments and DNS calculations have shown the total number of waves remains constant, but the wavelength per crossflow vortex varies axially and around the azimuth. To account for this variation, a model for the spanwise wavelength evolution (χ_2) has been developed and discussed by Kocian et al. [6]-[7]. It is a generalization of the previous technique, the spanwise wavenumber evolution (χ_1) [6]. Both

techniques make an additional adjustment to the local spanwise radius of curvature to satisfy the irrotationality condition.

1.2 Geometry and Basic-State Overview with Convergence Study

See Wheaton et al. [10] for details on the BOLT project and geometry. The BOLT geometry has spanwise concave curvature and a highly swept leading edge (Figure 1.1). The nose and leading-edge radii are 5 millimeters. The geometry is 0.866 meters in length, 0.438 meters in width at the base, and 0.256 meters in height at the base. There are two main regions on this geometry, the top surface and the “gutter”, which are symmetric on the opposite sides of the geometry. The top surface is the focus of the flight experiment and these calculations. The main purpose of the gutter is to completely separate the flow fields on the top and bottom surfaces of the geometry by preventing flow leakage between them.

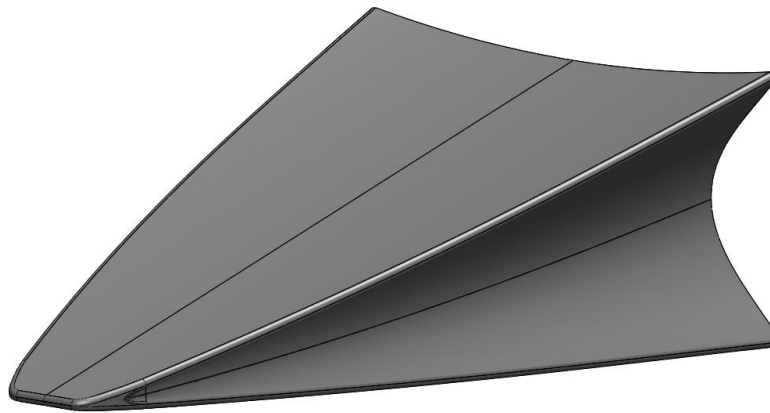


Figure 1.1. Geometric configuration.

Two flight times are computationally analyzed. Each analysis needs a steady, laminar, basic-state solution. These were computed using the Data Parallel Line Relaxation (DPLR) NASA CFD code [11]. The geometry is assumed to be at zero degrees angle of attack and yaw angle. Therefore, due to the model’s symmetry, a quarter section is simulated. The different flight times and flow conditions for the various basic-state configurations can be seen in Table 1.1. All computations were performed assuming air as an ideal gas using Sutherland’s law for viscosity and a constant Prandtl number $Pr = 0.72$.

Table 1.1. Flow conditions of two cases.

Case	Flight Time (s)	M_∞	Re^t (1/m)	T_w (K)	T_∞ (K)	P_∞ (Pa abs.)
Ascent	24.5 (Predicted)	5.2	11.52×10^6	400	217	6641
Descent	485.9 (Predicted)	7.4	11.52×10^6	400	217	4813

In order to fully analyze the boundary-layer stability on a flow field, a convergence study is necessary. Reed et al. [12] suggests looking at the basic state is not enough to consider a solution converged and that converging the boundary-layer stability is more important. Previous work for the yawed straight circular cone [12] and HIFiRE-5 38.1% scale model [6] shows the boundary-layer stability converges with less than one hundred million grid points distributed on a half cone. Using this insight, a convergence study is completed for the BOLT geometry ascent case, with various grid densities and topologies. The process includes creating multiple grid architectures, calculating the basic-state solutions, extracting streamlines, and performing boundary-layer stability analysis for the key instability mechanisms in the flow field. It should be noted that Wheaton [13] provided a grid he created and a basic-state solution he calculated that is included in the convergence study.

Five meshes are investigated with four different grid densities and three different topologies, and the details are shown in Table 1.2. The meshes are multi-domain structured grids. The Wheaton grid and grid #1 are completely unique from each other and the other grids. Grids #2 – #4 originate from the same grid and have the same topology. Grid #3 is half the grid density in the off-wall direction as grid #2, and grid #4 has the same number of grid points as grid #2 but has some different wall spacing. For all of the grids except grid #3, the wall spacing on the top surface is 1×10^{-7} meters. In the gutter region, there is more variation. The Wheaton grid and grid #4 have wall spacing of 1×10^{-7} meters and grids #1 and #2 have wall spacing of 1×10^{-5} meters. Grid #3, as mentioned above, has half the number of off-wall points with twice the wall-spacing height as grid #2.

Table 1.2. Various I-J-K structured grid indices for convergence study.

Grid	I	J	K
Wheaton	436	251	166
Grid #1	751	701	302
Grid #2	901	701	301
Grid #3	901	351	301
Grid #4	901	701	301

A full quarter-section view of grid #2 is shown in Figure 1.2. From this view, all of the grids look very similar. A unique feature of this flow field that makes all of the grids more complicated is that the geometry has concave curvature in the spanwise direction, but the shock has convex curvature in the azimuthal direction (shown later). This requires extra care when blending approximate wall-normal and shock-normal grid cells. Furthermore, the geometry features a very sudden change in surface curvature moving spanwise from the top surface to the gutter, while the shock development azimuthally remains gradual. Once again, this mismatch of the geometry shape and the shock shape requires extra care. The other key flow feature that adds complexity to the grid is the highly swept leading edge that minimizes azimuthal flow leakage from the gutter to the top surface. This creates an extremely small boundary layer (shown later)

that requires very small wall-spacing heights. This wall spacing has to blend with the quickly growing boundary layer that develops near the centerline on the top surface and the centerline in the gutter.

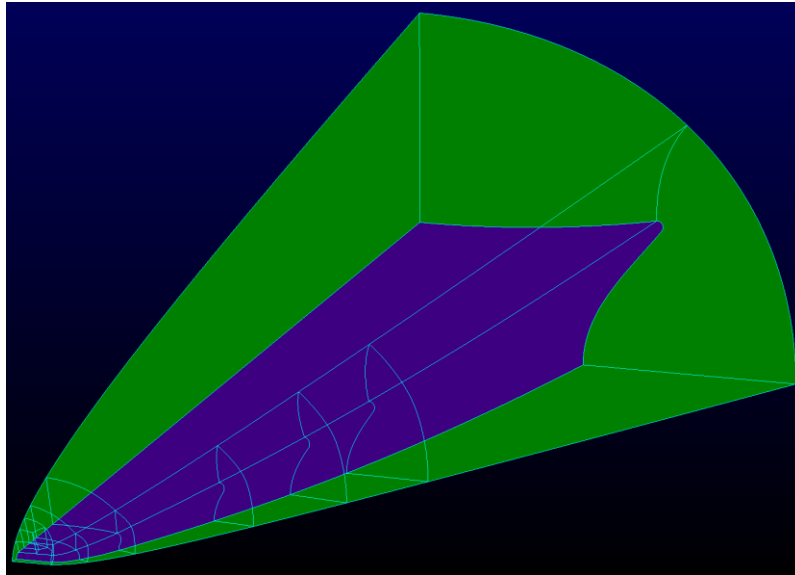


Figure 1.2. Full quarter-section view of grid #2.

When zooming into the region near the nose and swept leading-edge intersection, there are a few different options for grid topologies. Wheaton's grid includes a singularity near where the nose and swept leading edge meet. A zoomed-in view of grid #1 near this region is shown in Figure 1.3a. It shows some highly skewed grid cells in the top-left corner but no singularity. As previously mentioned, grids #2 – #4 share the same topology, and a zoomed-in view of grid #2 near the nose and swept leading-edge intersection is shown in Figure 1.3b. This topology pushes the singularity away from the intersection and toward the centerline. This idea comes from Dinzl & Candler's [14] work on the HIFiRE-5 38.1% scale model. They found that moving the singularities away from the leading edge and stationary-crossflow region numerically stabilized the basic-state calculations, removing the growth of stationary-crossflow disturbances seeded by the grid.

The basic-state solutions produced from grid #2 for all of the cases are shown in Figure 1.4. The fully developed flow fields with Mach contour slices at various axial locations for the ascent and descent cases are shown in Figure 1.4. In all cases, the shock appears azimuthally to have convex curvature. Also, an extremely thin boundary layer appears on the highly swept leading edge and a clear complex azimuthally varying flow structure appears near the top surface centerline and the gutter centerline. The formation of this complex flow structure is a product of the steady, undisturbed, laminar-flow basic-state solution, and no intentional disturbances or excitations are input to cause its manifestation.

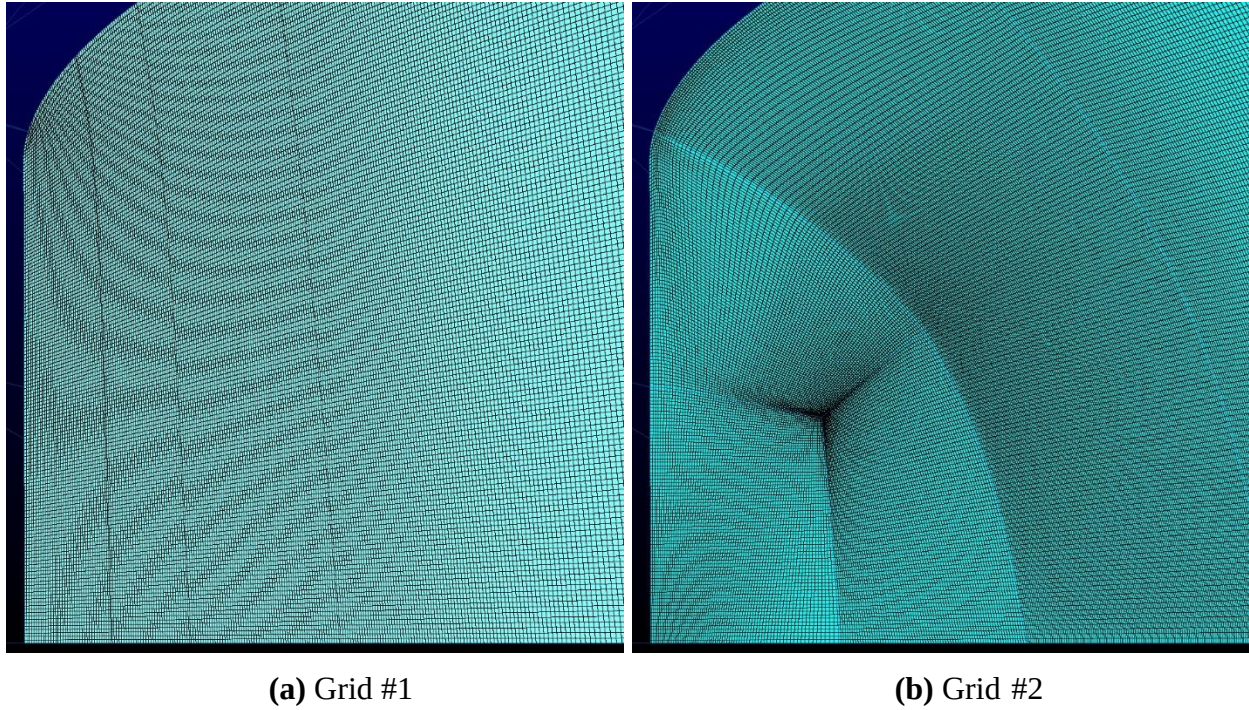


Figure 1.3. Zoomed-in top-view of grid topologies.

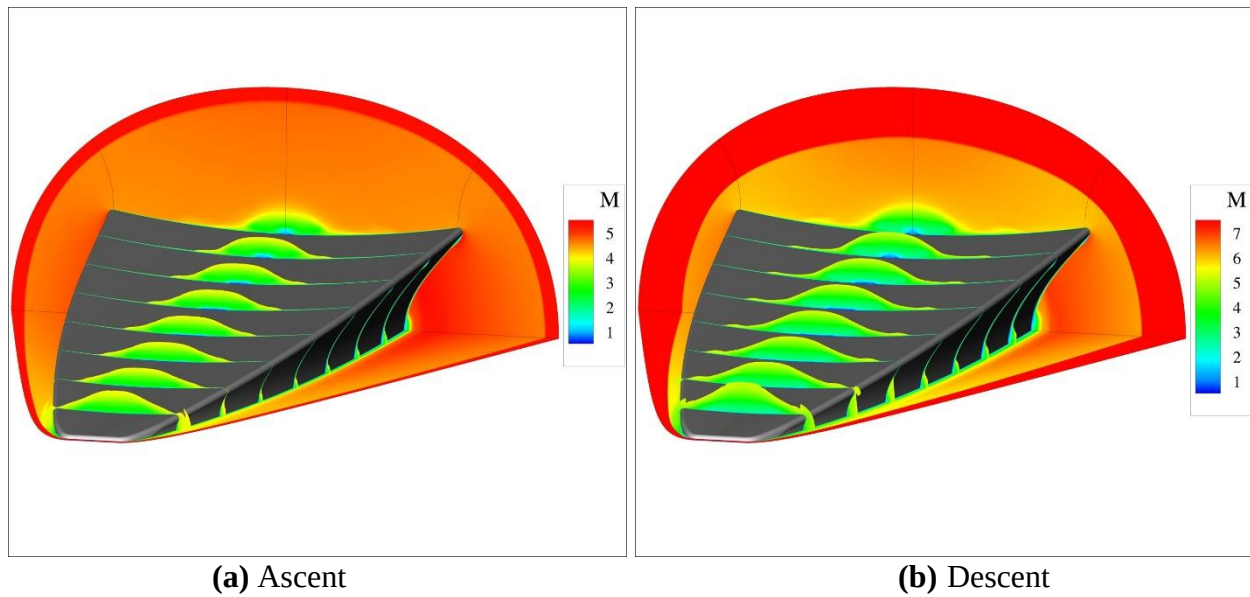


Figure 1.4. Mach contour slices at axial locations 0.1, 0.211, 0.322, 0.433, 0.543, 0.654, 0.765, 0.866 m.

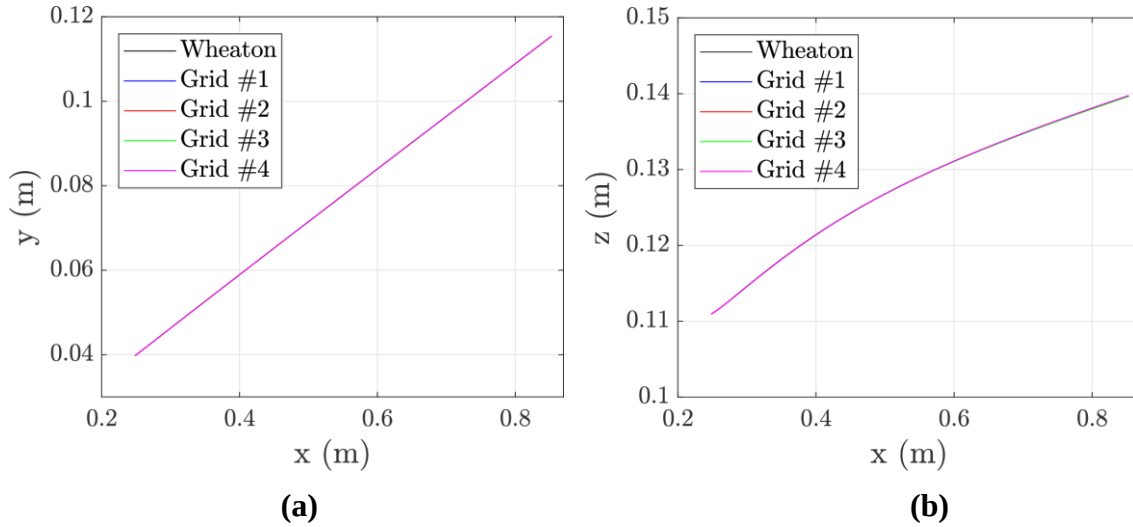


Figure 1.5. Ascent streamline global coordinates convergence study for five grids.

A single streamline that is initialized at the same global x , y , and z position is extracted for all of the grids for the ascent case. The global coordinates are shown in Figure 1.5. The streamlines are very similar to each other and the largest percentage difference at the end is 0.01% and 0.08% for the y and z positions, respectively.

LPSE calculations were completed on this streamline for all of the grids. The second-mode boundary-layer stability results are shown in Figure 1.6a and four different frequencies are investigated. The N-factors here and for all other LPSE calculations represent the mass-flux (ρu) linear amplification. The stationary- crossflow boundary-layer stability results are shown in Figure 1.6b and four different spanwise wavelengths are investigated. The maximum N-factor percentage difference is 3.97% and 1.21% for the second mode and stationary crossflow, respectively.

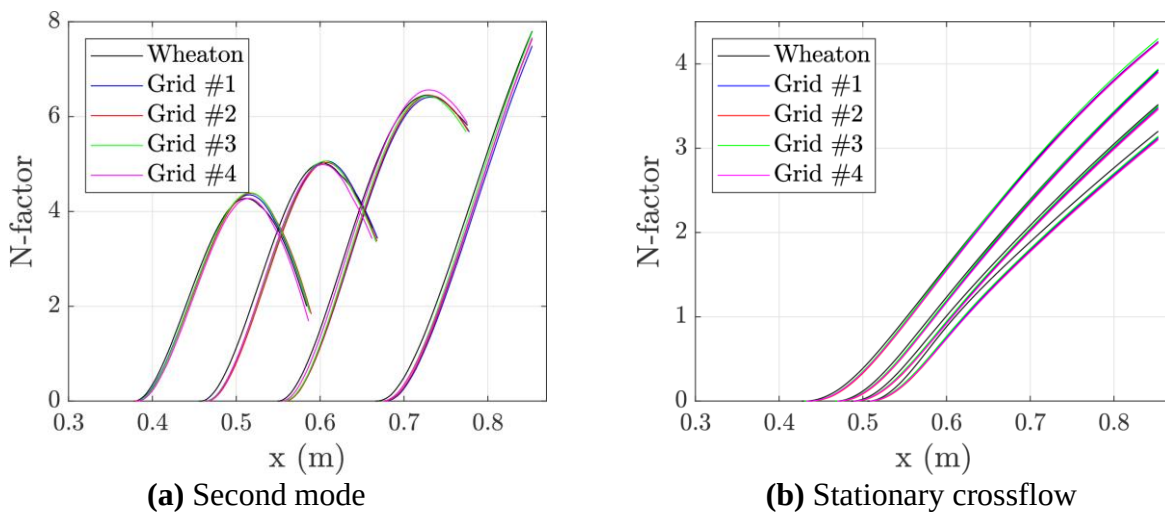


Figure 1.6. Ascent streamline LPSE N-factor convergence study for five grids.

The boundary-layer stability results demonstrate Wheaton's grid, which is the lowest resolution, is fully resolved for this streamline for the second-mode and stationary-crossflow mechanisms. For this work, the highest resolution grid, grid #2, is used, as the flow features are most clearly captured.

1.3 2-D and 3-D Linear Primary Instability Analysis on Streamlines

This section focuses on boundary-layer stability mechanism identification. Specifically, LPSE is performed on the ascent case and the descent case for the various instability mechanisms. In order to look at linear amplification of the various mechanisms on the entire top surface, many streamlines need to be extracted. The ascent case features 15 streamlines and the descent case features 17.

The second-mode instability on the top surface is the first to be identified. Figure 1.7 shows the development of the second-mode instability for the ascent and descent cases. Specifically, top-view contours of the LPSE N-factor envelope are portrayed, where $z = 0$ represents the geometric centerline on the top surface and the negative "z" corresponds to reflected results from the positive "z". The contour colors along the streamlines shown represent the largest LPSE N-factor at each location considering all frequencies of the second-mode disturbance.

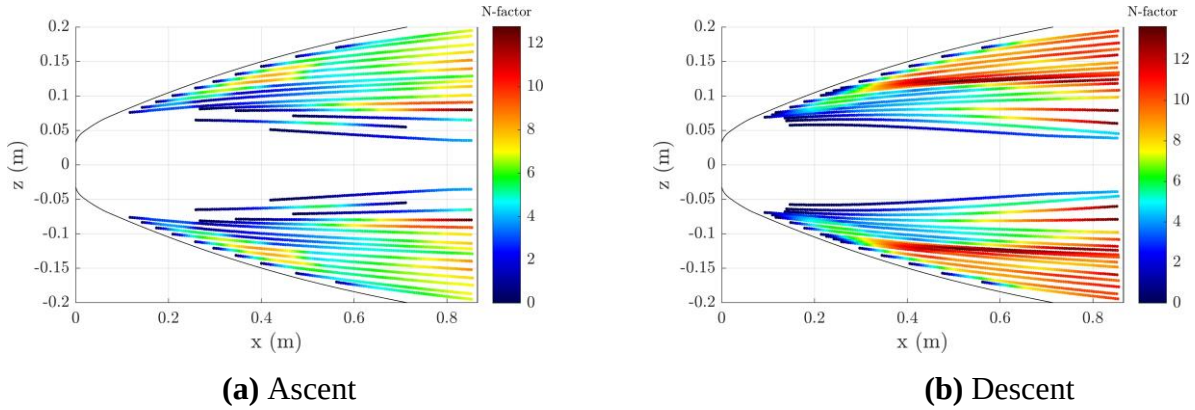


Figure 1.7. Second-mode LPSE N-factor-envelope top-view contours.

The ascent and descent cases have some very clear similarities. There appears to be strong second-mode growth near the leading edge and two highly amplified second-mode wedges at the back of the cone. The peak N-factors reach approximately 12-13. For the N-factor envelopes, the corresponding frequencies for the ascent case are between 325-1275 kHz and for the descent case are slightly larger between 425-1425 kHz. There are also some clear differences; the highly amplified second mode near the leading edge seems to remain extremely unstable for a much longer axial extent on the descent case. Also on the descent case, the overall large amplification appears more broadband.

The most unique feature to the flow field is the significantly amplified second mode near the leading edge. This has also been seen by Wheaton et al. [13] using LST. The frequencies, phase

speeds, and mode shapes of the instability definitely match the characteristics of a second mode. The freestream-to-wall temperature ratio at these flight conditions could reasonably destabilize and highly amplify the second mode, as seen by the various HIFiRE computations. Another insightful way to determine the strength of the second mode is to understand the marching-direction pressure gradient. According to Mack [9] and Zurigat et al. [15], an adverse pressure gradient is destabilizing to the second mode. In these computations, near the leading edge, the growth rates start out extremely large and unstable and then stabilize quickly. The trend of the pressure gradient matches; the pressure gradient starts extremely adverse and then quickly becomes favorable. In order to visualize the pressure gradient on BOLT, a top-view contour of the marching-direction adverse pressure gradient along each streamline is shown in Figure 1.8 for both cases. The seventh root of the gradient is only used in order to visualize a clear progression along each streamline. The favorable pressure gradient is not important for this analysis and has been neglected. Furthermore, the streamlines with the longer extent of an adverse pressure gradient directly align with the streamlines that produce the largest second-mode N-factors near the leading edge.

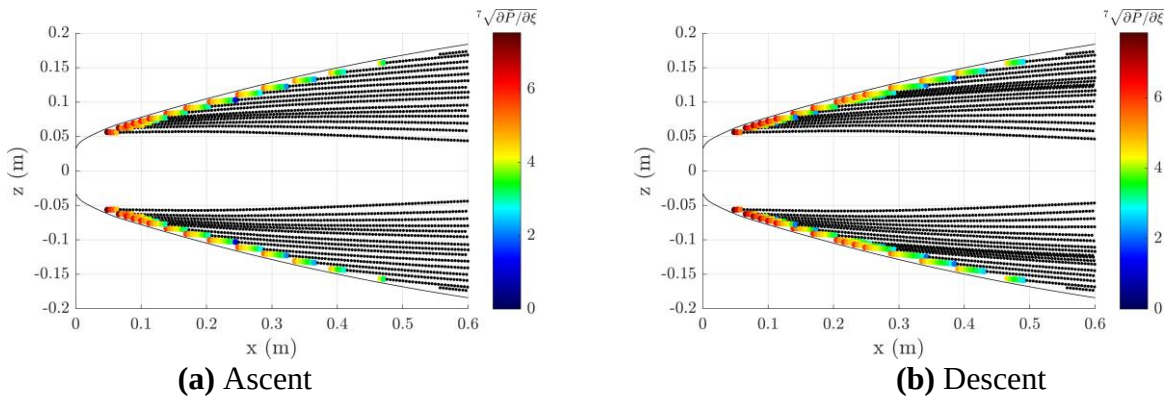


Figure 1.8. Marching-direction adverse pressure gradient top-view contours.

One streamline with a strong second-mode N-factor near the leading edge on the ascent case is studied in more depth. Figure 1.9 shows the marching-direction pressure gradient along the entire streamline with four dots at four different axial locations. It also shows the homogeneous acoustic impedance [16], the profile of the equation that produces the mean flow $\bar{\rho}\bar{u}$ generalized inflection point, and the individual wall normal $\bar{\rho}$ and $\bar{u}$ gradients at the four axial locations. From Figure 1.9, it is clear that a generalized inflection point exists for some of the axial stations. At the first two axial locations, a generalized inflection point and an unstable second-mode instability exist. As the third and fourth axial locations are approached, the second-mode growth rates stabilize. Looking at the wall-normal $\bar{\rho}$ and $\bar{u}$ gradients for the first two axial locations, it appears the existence of a generalized inflection point is mainly due to the positive wall-normal $\bar{\rho}$ gradient, which represents an acoustic impedance well [16]. The existence of an acoustic impedance well is another indicator that there is potential for a strong second mode near the leading edge.

Stationary crossflow is another mechanism present on the BOLT geometry at ascent and

descent predicted flow conditions. LPSE calculations for stationary crossflow were performed on the same streamlines as the second-mode analysis. Figure 1.10 shows top-view contours of the LPSE N-factor envelope for the ascent and descent cases. The envelope represents the largest LPSE N-factor at each location considering all initial spanwise wavelengths and their respective downstream variation. The peak LPSE N-factors on the top surface reach approximately 7-8 for both cases. For the N-factor envelopes, the corresponding spanwise wavelengths are between 0.5-7 millimeters and are distributed very similarly for each case.

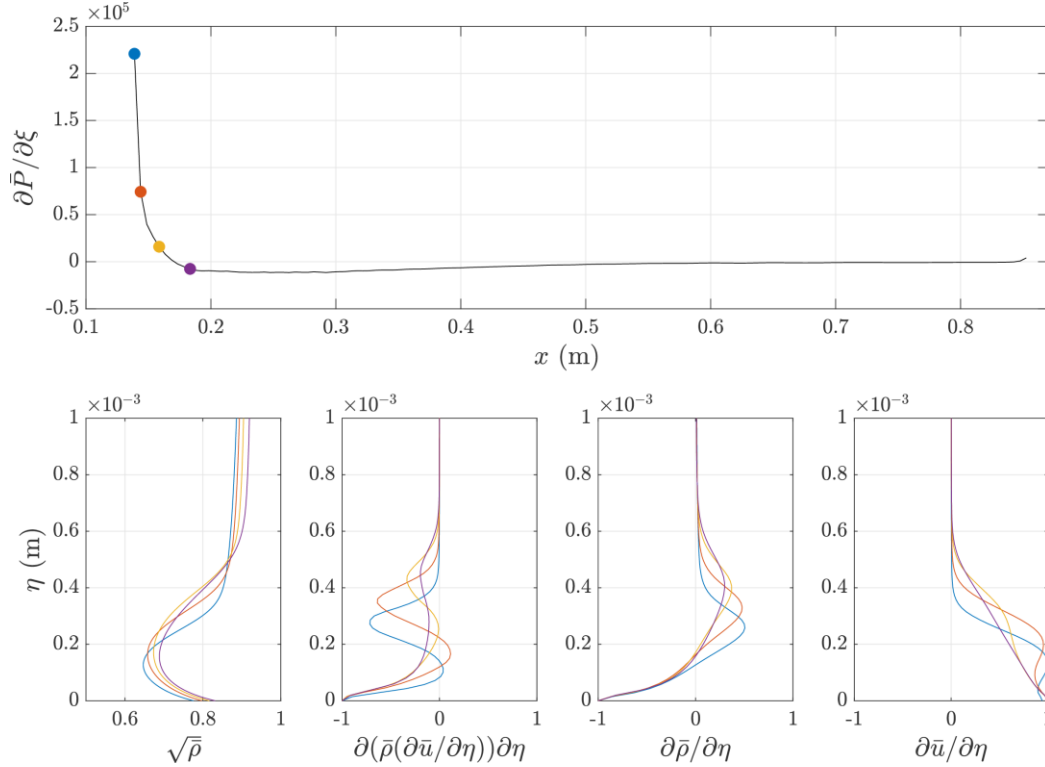


Figure 1.9. Ascent streamline marching-direction adverse pressure gradient, homogeneous acoustic impedance, profile of equation that produces the generalized inflection point, and wall-normal basic-state quantity gradients.

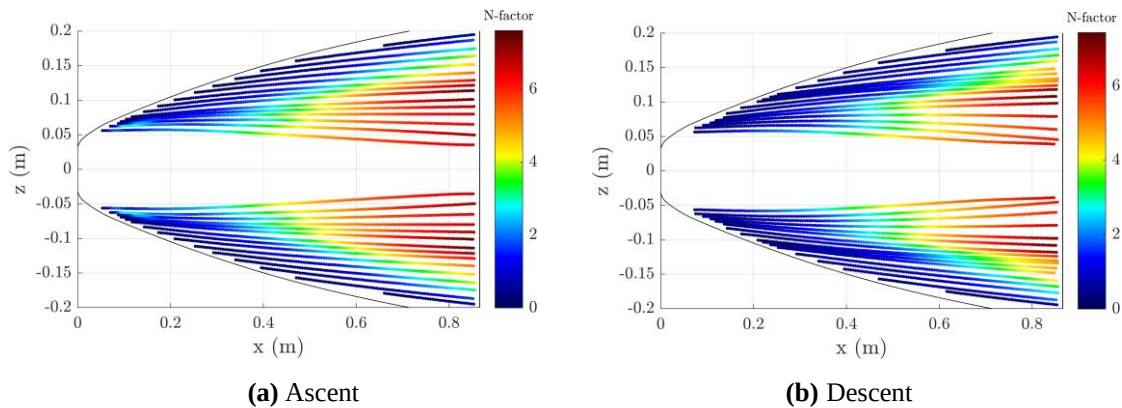


Figure 1.10. Stationary-crossflow LPSE N-factor-envelope top-view contours.

Furthermore, the contours produce a wedge-shaped amplification front similar to HIFiRE-5. This is significant for the flight experiment because if stationary crossflow is the dominant laminar-to-turbulent transition mechanism, even with a coarse distribution of temperature and heat-flux sensors, the relatively simple wedge structure should be clearly identifiable, as it was with HIFiRE-5b. Although the wedge-shaped amplification front is clearly defined, there are definitely some unique features when comparing the ascent and the descent cases. The flow field on the descent case becomes much more complex in the spanwise direction leading to strongly turning streamlines near the back of the cone and a flatter LPSE N-factor-wedge tip.

If stationary crossflow exists, traveling crossflow likely exists as well. In flight, it is likely not the transition mechanism, but it has been previously shown that traveling crossflow can play a significant part in the transition process in ground-experiment facilities [17]. Therefore, it is important to analyze and determine what boundary-layer stability content contributes to the growth of the instability. Figure 1.11 shows an LPSE N-factor envelope with a dense sweep of frequencies and spanwise wavelengths for the ascent and descent cases on the streamline that produced the most amplified stationary crossflow. This streamline does not capture the content of the entire 3-D flow field, but by looking at the strong stationary-crossflow region, it is a good estimate of the location for the largest traveling crossflow. As expected, the traveling crossflow is more amplified than the stationary crossflow.

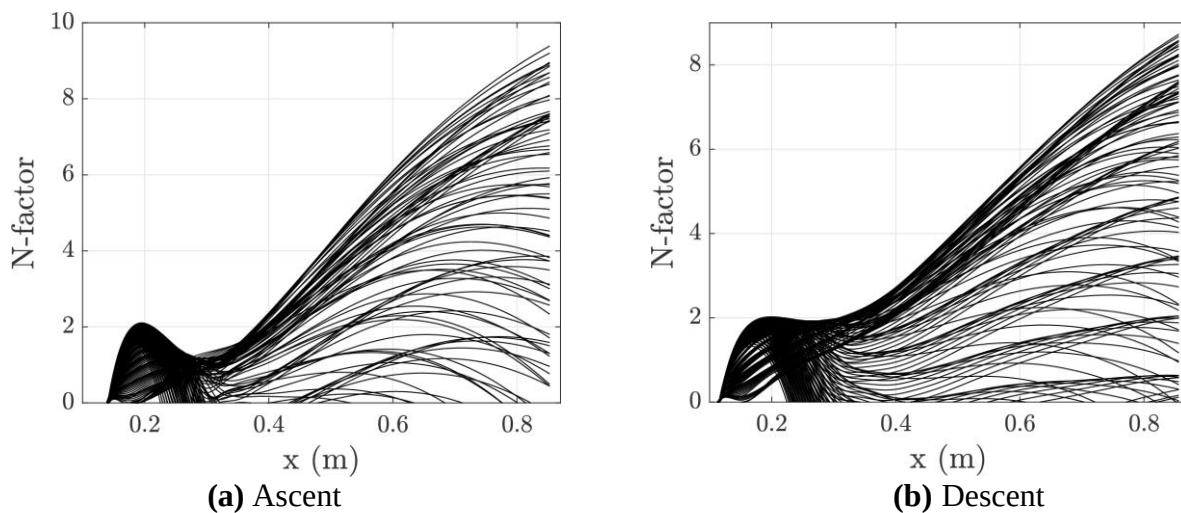


Figure 1.11. Traveling-crossflow LPSE N-factor envelope for streamline with most amplified stationary crossflow.

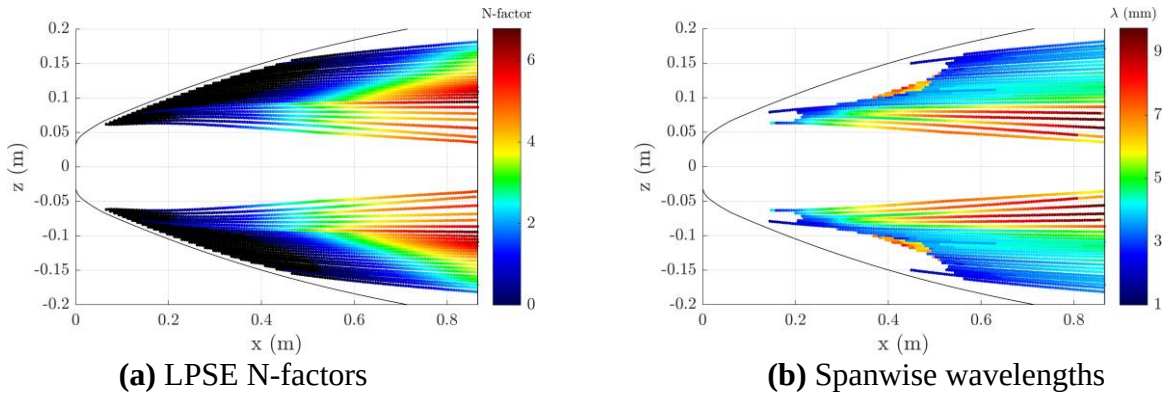
Two peaks appear in the LPSE N-factor distribution for each case, similar qualitatively to the second mode. Table 1.3 summarizes the boundary-layer stability results at those peaks. The frequency content is very interesting. Near the leading edge, higher frequencies are unstable and amplified (greater than or equal to 50 kHz), while downstream of the leading edge, the frequencies become quite small (less than or equal to 10 kHz).

Table 1.3. Traveling-crossflow LPSE N-factor-envelope peaks and features for streamline.

Case	Peak	x (m)	N-factor	F (kHz)	λ (mm)	k_c
Ascent	1	0.20	2.11	75	2.34	3381
Ascent	2	0.86	9.39	10	4.11	1860
Descent	1	0.20	2.02	50	3.34	2497
Descent	2	0.86	8.73	10	5.02	1518

1.4 3-D Primary Instability Analysis on Vortex Paths

The key mechanisms have been identified on streamlines and it appears stationary crossflow is going to be the dominant mechanism in flight away from the leading edge and the centerline. At this point, it makes sense to refine the stationary-crossflow calculations to get a more accurate understanding of the linear development of the stationary-crossflow instability. Utilizing vortex paths and the evolving spanwise wavelength technique [6]-[7] has been shown previously to approximate the development very well. LPSE analysis was performed on many vortex paths on the ascent and descent cases. The vortex paths were run for a large range of initial spanwise wavelengths. Figure 1.12a and Figure 1.13a show top-view contours of the LPSE N-factor envelope for the ascent and descent cases, respectively. The envelope represents the largest LPSE N-factor at each location considering all initial spanwise wavelengths and their respective downstream variation. The other contours shown in Figure 1.12b and Figure 1.13b are the corresponding spanwise wavelengths from the LPSE N-factor contour. These wavelengths approach one millimeter near the swept leading edge and grow to nine to twelve millimeters downstream toward the centerline.

**Figure 1.12.** Ascent various vortex paths and LPSE calculations.

The vortex path N-factors approach 6-7 at the back of the geometry for both cases. Comparing these N-factors with the streamline N-factors, the streamline N-factors are slightly larger. This trend was seen by Kocian [6] on other geometries. The N-factors develop less smoothly across all of the vortex paths, but overall the wedge remains regardless of the path chosen for analysis.

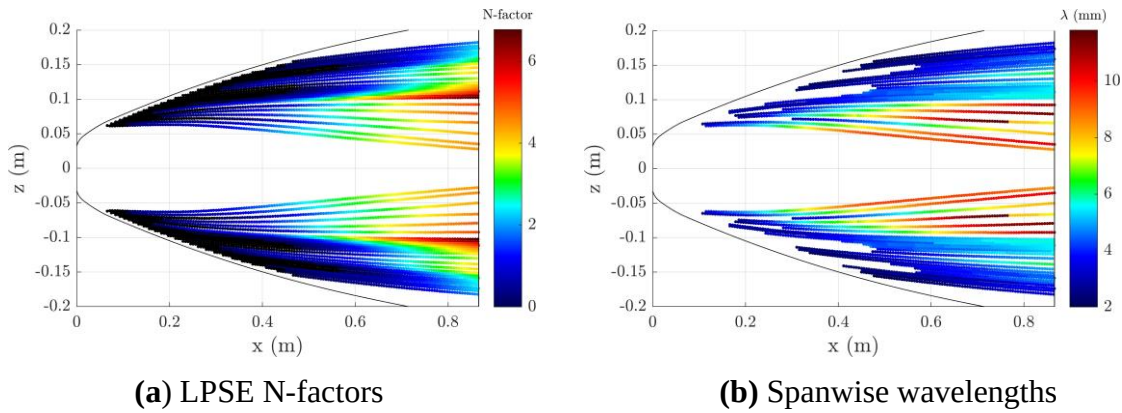
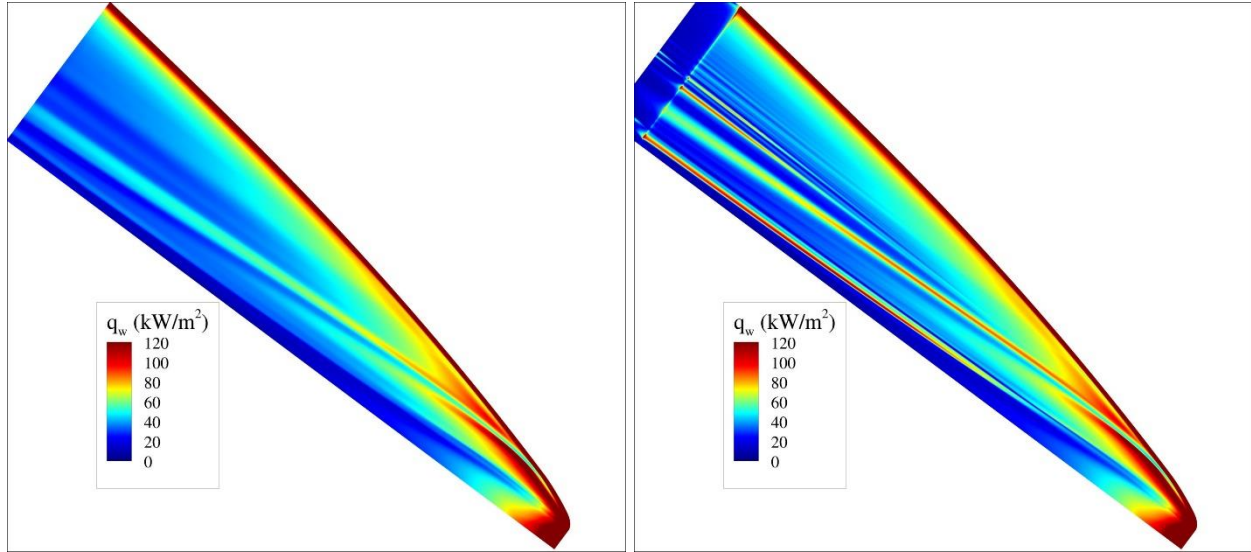


Figure 1.13. Descent various vortex paths and LPSE calculations.

1.5 3-D Nonlinear Primary Instability Analysis

While linear calculations were being performed on BOLT, a steady DNS was performed by Thome et al. [18] at flight conditions (this section will only focus on the ascent case). The steady DNS is defined for BOLT as a numerical scheme consisting of a fourth-order low dissipation method calculated in US3D. Many streaks appeared in the DNS; some of those streaks seemed to be captured in the basic-state calculations from the ascent case here using DPLR, but other streaks were not captured. Grid convergence was performed for the basic-state calculations here using DPLR and stationary crossflow was deemed to amplify strongly in the region of these streak differences. Thome et al. [18] began to analyze these streaks and determined the various streaks are due to the curved shock vorticity. A further investigation was undertaken here to determine if the observations in the DNS could be modeled using DPLR coupled with NPSE. That is, are the flow streaks attributable to a coupled receptivity (initial amplitude) and highly amplified stationary-crossflow (based on linear N-factors) problem? In order to study this, the basic-state calculations here will be discussed as “standard quality CFD”. In essence, the DPLR solutions consist of a standard second-order upwind method, which is consistent with many CFD codes in the community. Direct comparisons will be made to the DNS, as the DNS heat-flux data was provided by Mr. John Thome (private communication based on Thome et al. [18]).

As just mentioned, the streaking pattern does not match from the standard quality CFD and the DNS. Heat-flux contours are shown in Figure 1.14 for both. Figure 1.14a is the standard quality CFD solution and Figure 1.14b is the DNS. It is important to note that Figure 1.14b does not exactly appear in Thome et al. [18], but the figure is similar to their figure 11, both with the same data. The region of interest where the streak differences appear is directly between the centerline and the swept leading edge at the back of the geometry. Clear differences exist elsewhere in the flow field, but the focus of this section is in this region only. Note that the flow field near the nose looks very similar in the two calculations.



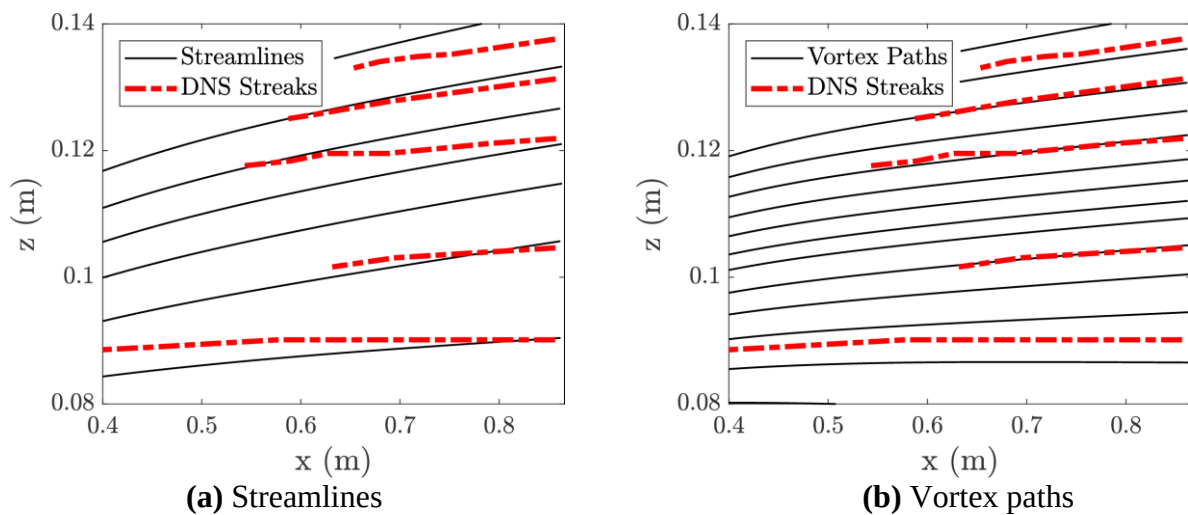
(a) Standard quality CFD basic state

(b) DNS

Figure 1.14. Wall heat flux for DPLR and US3D steady solutions, respectively.

In order to determine if the heating streaks from the DNS can be modeled as stationary crossflow, a series of predictive techniques using PSE are applied to the standard quality CFD basic-state solution in order to compare with the DNS.

The first technique is based on the path that these streaks follow. As already detailed, streamlines and vortex paths have been extracted from the basic state and analyzed using LPSE. They are shown in Figure 1.15. Furthermore, the streaks from the DNS have been extracted manually and are also shown in Figure 1.15, as dashed lines.



(a) Streamlines

(b) Vortex paths

Figure 1.15. Extracted standard quality CFD paths from DPLR compared to extracted DNS streaks from US3D.

It is clear that the streamlines (Figure 1.15a) do not seem to follow the heating streaks, but the vortex paths (Figure 1.15b) do seem to follow the heating streaks. This is critical because the first indicator that these streaks are due to the stationary-crossflow instability is satisfied. The vortex paths, although not exact, have been shown to model the direction of growth of the stationary-crossflow instability quite well, based on work from Kocian et al. [6]-[7].

For further analysis, the fourth streak from the top of Figure 1.15b, and the corresponding vortex path, will be studied. Figure 1.16 shows a contour of the DNS wall heat flux in the region near the fourth streak. There is clearly a strong localized maximum and minimum, as expected from a stationary-crossflow instability. The solid black line represents the vortex path that will be analyzed further. The dashed black line represents a neighboring vortex path that is used in order to understand the spanwise variation in wall heat flux for the standard quality CFD solution. The neighboring vortex path is far enough away to capture a spanwise wavelength of wall heat-flux data.

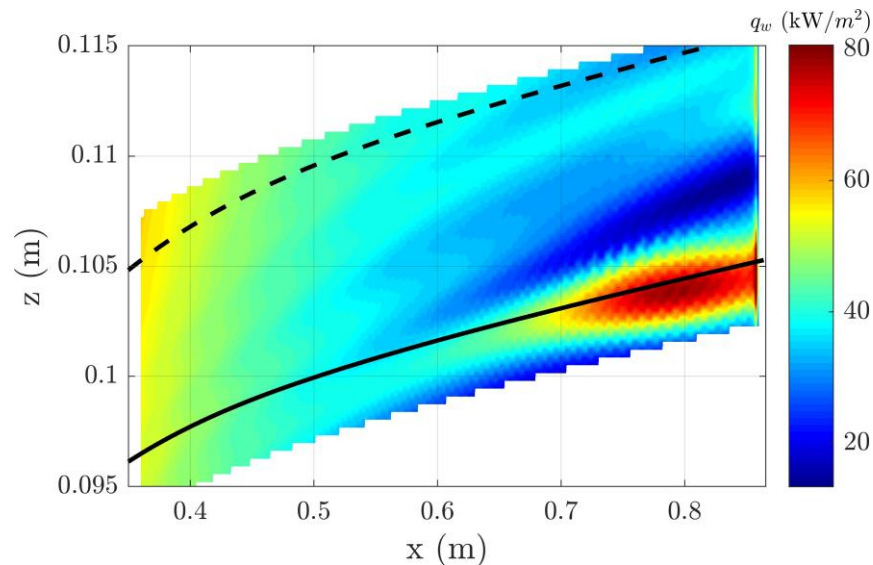


Figure 1.16. DNS wall heat-flux contour with standard quality CFD vortex path of interest and neighboring vortex path.

LPSE analysis was performed on the vortex path of interest. The N-factors are shown in Figure 1.17a. They show large amplification, approaching 6-7. The corresponding spanwise wavelength development that produced the largest N-factor at the back of the geometry is shown in Figure 1.17b with a black line. The spanwise wavelength evolution approaches 4.6 millimeters at the back of BOLT. Using the data from Figure 1.16, the DNS heat streak wavelengths were extracted. The left-side-skewed and right-side-skewed heat-streak wavelengths are shown (a linear fit was applied as the data had some noise, but it did not alter the main trends), with their average shown in red. The DNS average spanwise wavelength development matches the standard quality CFD basic state coupled with LPSE prediction. This is very significant because a second indicator that these streaks are due to the stationary-crossflow instability is satisfied.

Predicting the growth direction, the spanwise wavelength, and a linear amplification suggests that this streak and its neighbors are stationary crossflow. However, as CFD flow fields become more complicated, determining the boundary-layer stability mechanism can be less clear. NPSE provides a unique capability to be able to also calculate wall heating levels. If these heating levels match the DNS streak heating levels, this would be more evidence that these streaks are the result of the stationary-crossflow instability.

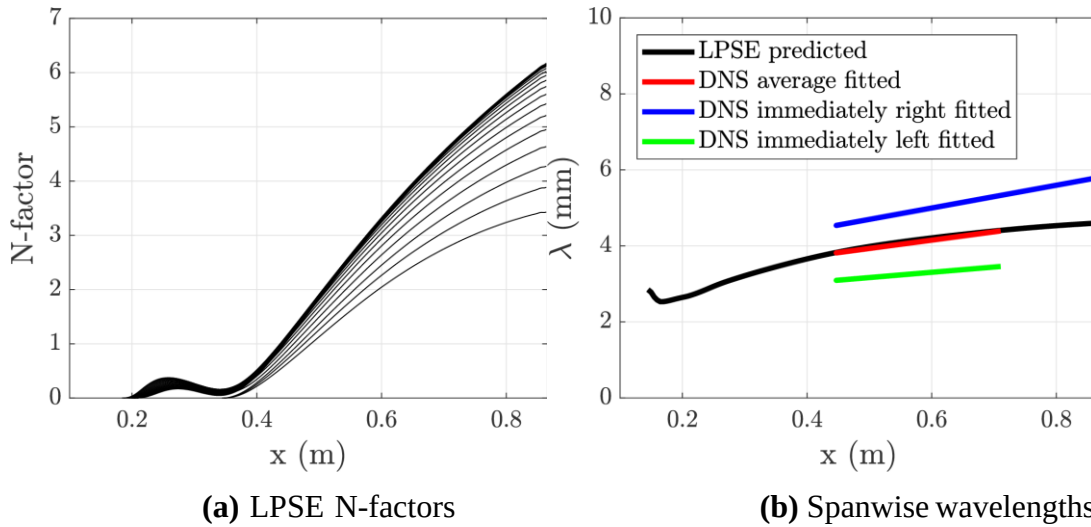


Figure 1.17. Vortex path of interest LPSE N-factors with predicted spanwise wavelengths compared to DNS streak.

In order to perform NPSE calculations, initial disturbance amplitudes need to be specified on the vortex path being analyzed. The NPSE calculations are initialized on the vortex path by specifying a u -amplitude, $u^i 0,1$, at a specified axial location, which is normalized by the edge u -velocity at that specified axial location (LST neutral point for these calculations). The edge is defined as the end of the wall-normal profile inside the shock. The “0,1” notation represents zero frequency and the fundamental spanwise wavelength in the spatial/temporal expansion in the NPSE formulation, based on the LPSE results. Anywhere from 5-10 superharmonics are enough to converge the NPSE results. The only missing component at this point is the initial disturbance amplitude.

Predicting the initial disturbance amplitude requires a computational receptivity transfer function. The receptivity problem is challenging and beyond the scope of this work. Instead, for the present analysis, the approximate initial disturbance amplitude will undergo an iterative process based on the first deviation of the standard quality CFD laminar basic state and the DNS heating rates. At this location, nonlinear effects from the NPSE should start to become noticeable (u -amplitude of the fundamental mode should approach 5-10%). It is important to note that for the present work the NPSE initial disturbance amplitudes are correlated so that this match occurs, but with a receptivity transfer function, the NPSE initial amplitudes would be predictive. After some basic analysis of the heating rates from the standard quality CFD and the DNS, it is clear that the

first deviation occurs just downstream of 0.40 meters. This can be seen in Figure 1.18, where the gray black lines (dashed and solid) represent the basic-state heat flux, the blue lines (dots) represent the DNS heat flux, and the red line represents the deviation location. The solid black line is the vortex path of interest, while the dashed black line is the neighboring vortex path (both are from Figure 1.16).

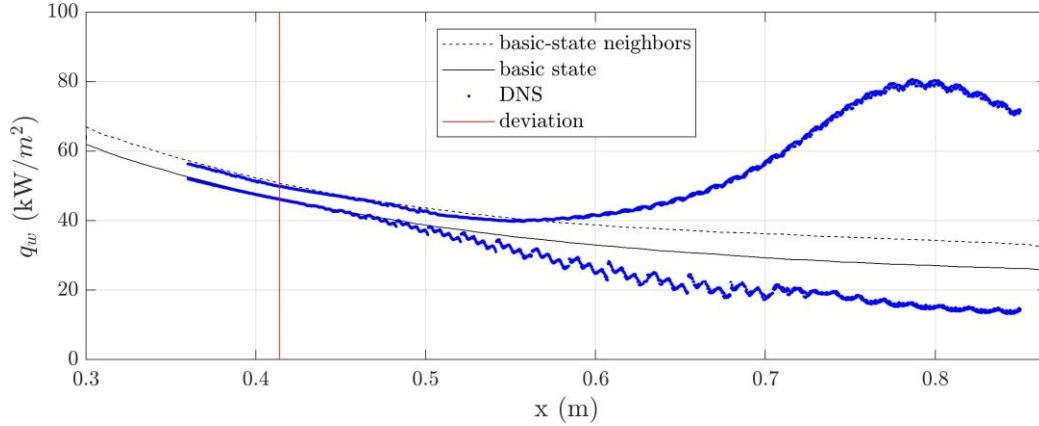


Figure 1.18. Vortex path of interest basic-state wall heat flux compared to DNS streak.

After a few NPSE calculations, it was determined that the required initial u -amplitude of the fundamental spanwise wavelength (based on Figure 1.17b) was 0.29% of U_e . For convenience, the LST neutral point location and reference parameters are all listed in Table 1.4. The characteristic length is defined by f_e .

Table 1.4. Reference parameters.

x_{np} (m)	U_e (m/s)	T_e (K)	ρ_e (kg/m ³)	f_e (m)
0.179	1477.35	302.33	0.19	1.09×10^{-4}

The corresponding NPSE u -amplitudes for the fundamental mode, the superharmonics, and the mean-flow distortion are shown in Figure 1.19. They portray linear growth of the stationary-crossflow instability early, followed by a highly nonlinear region with saturated amplitudes. The maximum u -amplitude of the fundamental mode is approximately 37%, with saturated u -amplitudes around 35-37%.

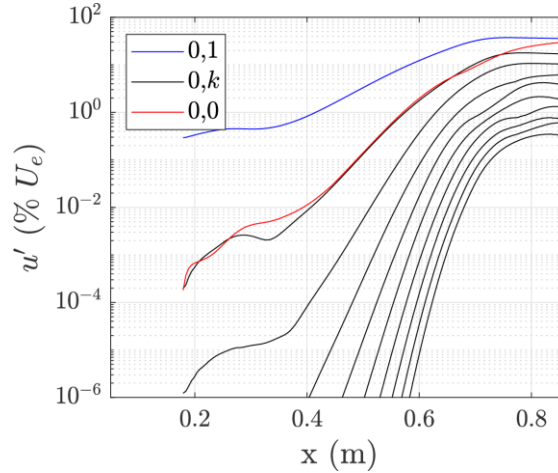


Figure 1.19. Vortex path of interest NPSE u-amplitudes.

As previously mentioned, the goal of the NPSE simulation using the standard quality CFD solution as the basic state is to match the DNS heating levels. Up until now, comparisons in the community have been made using a combination of the basic state and the mean-flow distortion to calculate the heat flux. This is a straightforward technique to calculate the heating, however it is more of a local average heating in the spanwise direction over the stationary-crossflow wavelength. It has been useful for computational analysis, but it is an incomplete understanding of the heating. A more complete physics-based technique is attempted. Under the crossflow vortex wave action, the wall heat flux varies in the spanwise direction over a wavelength reaching a minimum in the vicinity of the crest of the wave and a maximum near the trough. As the locations of the various spanwise phases of the NPSE stationary-crossflow wave on an actual geometry are currently unpredictable, a starting point is to understand the heat flux over the entire stationary-crossflow wavelength along the path, and determine the local spanwise *maximum* and *minimum* heating bounds as a function of axial position.

The physics-based method requires a completely rebuilt flow field (laminar basic state plus NPSE) over a spanwise crossflow-vortex wavelength (the phase is unimportant), as if secondary instability analysis were going to be performed. [Note that there will be no secondary instability analysis conducted here]. Using the rebuilt flow field at all axial stations and 51 equally spaced spanwise points per location (enough to converge the rebuilt flow fields), the local wall heat flux is calculated. This produces 51 spanwise lines of data to build the boundaries. The whole data set has been added to the data set represented in Figure 1.18 and all is shown in Figure 1.20. Here the basic-state heat flux (gray black dashed lines) has been supplemented with these 51 lines of spanwise varying data due to the inclusion of the stationary crossflow (gray black solid lines). As a reminder, the blue lines (dots) represent the DNS heat flux, and the red line represents the deviation location.

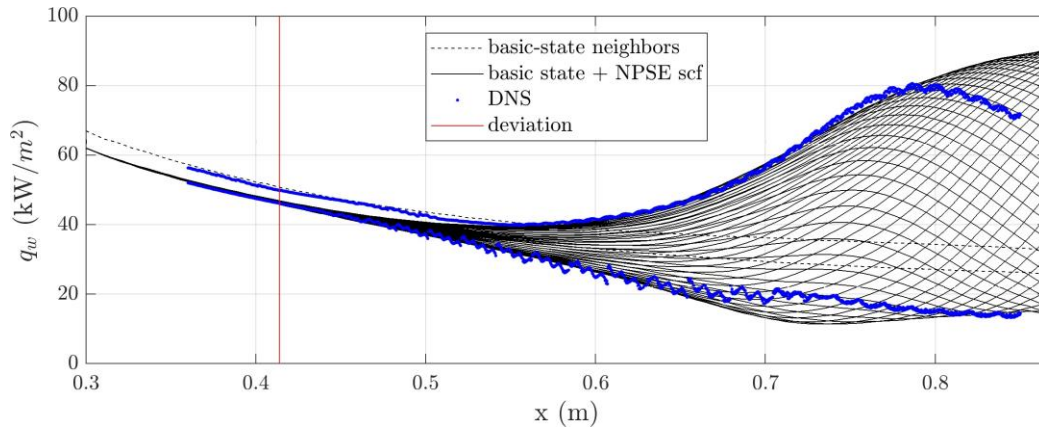


Figure 1.20. Vortex path of interest NPSE wall heat flux compared to DNS streak.

As mentioned, the local maximum and minimum wall heat flux bounds are the important aspect currently. The rebuilt-flow-field wall-heating bounds appear to match the DNS heating bounds except in a small region along the lower part and further into the nonlinear region toward the downstream boundary. At the time of the writing of this section, the DNS was being further investigated for convergence (private communication with Mr. John Thome). However, in the key region initially after transition onset (deviation), the behavior predicted compares very well between the two approaches up to approximately 0.8 m on the maximum-wall-heating envelope, and except in the vicinity of 0.7 m on the minimum-wall-heating envelope. From the NPSE perspective, the main difference appears downstream where the nonlinear effects become substantial, which means it is possible the NPSE technique might not be valid this far into the nonlinear region. Alternatively, from the DNS perspective, Mr. Thome mentioned (in private communication) that the calculations in Thome et al. [18] were a first attempt at a DNS on the BOLT geometry. Further refinement of the DNS solution could affect the stationary-crossflow rollovers, especially farther away from the wall, where there are fewer grid points. This could possibly explain the small discrepancy at the lower bound around 0.70 meters. [The lower bound is equivalent to the crest of the rollover.] The other discrepancy appears toward the end of the comparison domain at the upper bound, starting at about 0.80 meters. Based on Figure 1.14 and Figure 1.16, there seems to be some outflow difference between the standard quality CFD laminar basic state and the DNS solutions. This could possibly explain that discrepancy, however, it is not clear how far upstream the effect propagates, if at all. Regardless of the two differences, the overall development of the maximum and minimum wall heating from transition onset up until about 0.80 meters compares quite well between the standard quality CFD laminar basic state coupled with NPSE analysis and the DNS. Furthermore, because the wall heating bounds match and with the prior discussed match in direction and wavelength of the heating streaks, this is strong evidence that these streaks can be attributable to a receptivity problem, possibly from the curved shock vorticity, coupled with a highly amplified stationary-crossflow instability. Even more interestingly, NPSE has proven to be a technique that can help identify the character of the various streaks seen in the DNS and suggest they could be modeled as the stationary-crossflow instability.

This result has shown the benefits of closely coupling DNS studies with receptivity and NPSE studies for complex configurations.

1.6 Conclusions

BOLT is the newest hypersonic boundary-layer stability flight experiment that features a rather complex 3-D geometry. The geometry includes concave curvature in the spanwise direction and a highly swept leading edge. These geometric features lead to more complicated gridding techniques in order to generate quality flow fields for boundary-layer stability analysis. An in-depth boundary-layer stability convergence study of various grid architectures was completed and the key techniques to creating a quality grid for the BOLT geometry were discussed.

Two different predicted flight trajectory times were studied using the LPSE technique in EPIC and many mechanisms were identified. The second-mode instability obtains large LPSE N-factors, and the amplification near the back produces a multi-wedge front, and a unique large amplification region near the leading edge exists. The unique second-mode development near the leading edge was analyzed in depth and found to be reasonable due to the existence of an adverse pressure gradient and an acoustic impedance well. Stationary crossflow reaches reasonably large LPSE N-factors for both cases and the amplification front is wedge-shaped, similar to HIFiRE-5b. Traveling crossflow was also identified in the LPSE calculations and it produces larger N-factors than the stationary crossflow, as expected. If the LPSE analysis from HIFiRE-5b is any indicator of the occurrence of transition, stationary crossflow, at the conditions in this work, could reasonably be the laminar-to-turbulent transition mechanism for the BOLT geometry in the region between the leading edges and the centerline.

As the community looks to apply knowledge from computations and ground experiments to flight tests (or even actual flight vehicles), collecting wall data is the most realistic option. The researchers were able to verify EPIC's NPSE capability using wall quantities from direct numerical simulations. An initial amplitude was not provided for the NPSE calculations, and instead, a criterion was established using the provided data to estimate an initial amplitude. A physics-based technique to analyze stationary-crossflow NPSE spanwise maximum and minimum wall heating rates was assessed and determined to be quite useful. The findings suggest that stationary-crossflow NPSE analysis can match a steady direct numerical simulation. Not only can the NPSE analysis match a steady DNS, but for the BOLT geometry in flight (a complex 3-D flow field), in particular, NPSE has proven to be a technique that can help identify the character of the various streaks seen in the DNS between the leading edges and the centerline, and suggest they could be modeled as the stationary-crossflow instability coupled with a receptivity model.

2. GROUND-TEST CONDITIONS

After creating the mesh guided by the flight results above, NASA's DPLR CFD code [10]-[11] is used to create laminar basic states for the subscale BOLT model developed for ground testing. Paths along the geometry following inviscid streamlines are then extracted from these solutions using an in-house data extraction code. The surface heat flux is also extracted from the basic states. Finally, the stability research code EPIC [1]-[2] is used to perform linear PSE (LPSE) analysis on the surface of subscale BOLT.

The wind tunnel model of the BOLT geometry contains the same features as its full-scale counterpart, namely the highly swept leading edge, spanwise-concave surfaces, and the widely varying boundary-layer heights. The particular model under consideration is a 1/3 scale model of the flight geometry that measures 0.288 meters in length. The surface heat flux on subscale BOLT is compared to experiments run in the Mach 6 Quiet Tunnel (M6QT) at Texas A&M [19] and also to computations performed at the University of Minnesota [18]. Furthermore, three different types of instabilities are examined on the top surface of the geometry with a large number of inviscid streamline paths. The instabilities found on this geometry include the 2nd mode as well as stationary and traveling crossflow. The freestream conditions, isothermal wall temperature, and unit Reynolds number used to generate the basic state for BOLT are listed in Table 2.1.

Table 2.1. Tabulated input conditions for the subscale BOLT basic state.

M_∞	$\rho_\infty (kg/m^3)$	$T_\infty (K)$	$T_w (K)$	$Re^0 (1/m)$
6	0.040332	52.8	300	10.28×10^6

2.1 Laminar Basic State

The basic state for the BOLT wind tunnel model is visualized in Figure 2.1, and this figure shows a quarter of the vehicle geometry. This representation of the BOLT wind tunnel model is used in order to take advantage of the two planes of symmetry which exist along the center of the upper (main) surface and side (gutter) surface. Taking advantage of these symmetry planes by only computing a quarter geometry solution saves on computational cost of the basic state. The figure consists of the surface of subscale BOLT in black overlaid with axial slices contoured by Mach number. This visualization shows a number of unique flow features on the BOLT geometry. First, a large vortex lies along the symmetry plane on the main surface of the geometry. In addition, a weak secondary vortex can be seen outboard of the main vortex. This secondary vortex resides about halfway between the main vortex and the swept leading edge. Furthermore, one challenging aspect of this geometry can be seen in the wide range of boundary-layer heights, from relatively large heights near the main vortex, to very small heights on the swept leading edge. One additional difficult aspect of this geometry that can be seen is the highly curved shock that develops as a result of the swept leading edge.

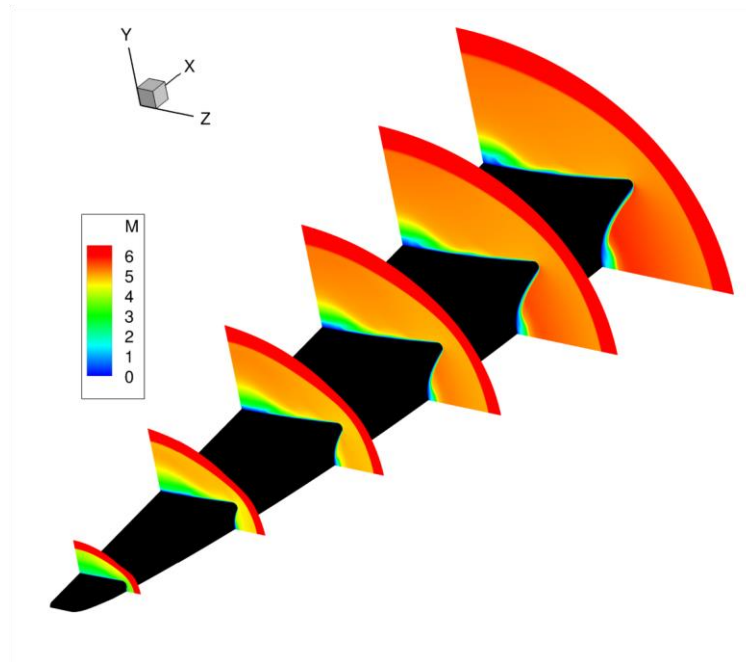


Figure 2.1. Basic state Mach contours on the subscale BOLT.

2.2 Surface Heat Flux Comparison

The heat flux at the surface of subscale BOLT is extracted from the basic state and plotted in Figure 2.2. In this top-down view, heat flux is shown over the top surface of BOLT with the quarter geometry solution reflected across the main surface symmetry plane. Several heating features are readily apparent on this geometry. Two main heating streaks are present over the majority of the top surface with the inboard streak being flanked by two dimmer streaks residing on its inboard side. In addition, a rather narrow heating streak is present immediately downstream of the corner between the nose and swept leading edge at the front of the geometry.

This heat flux plot compares favorably to results obtained in other computations as well as experiments. This image can be compared to the top image in figure 5 of Thome et al. [18] which shows the heat flux extracted from a DNS of subscale BOLT using US3D – this is shown later in this report. The inboard and outboard heating streaks are present in both images although there are differences in the features of the streaks. In particular, the DNS has two streaks in the location of the single outboard streak in Figure 2.2. Furthermore, the narrow heating streak originating downstream of the corner as well as the dimmer streaks flanking the inboard streak are also present in the DNS. Figure 16b in Kostak et al. [19] (shown later in this report) shows the heat flux extracted from a wind tunnel run in the M6QT.

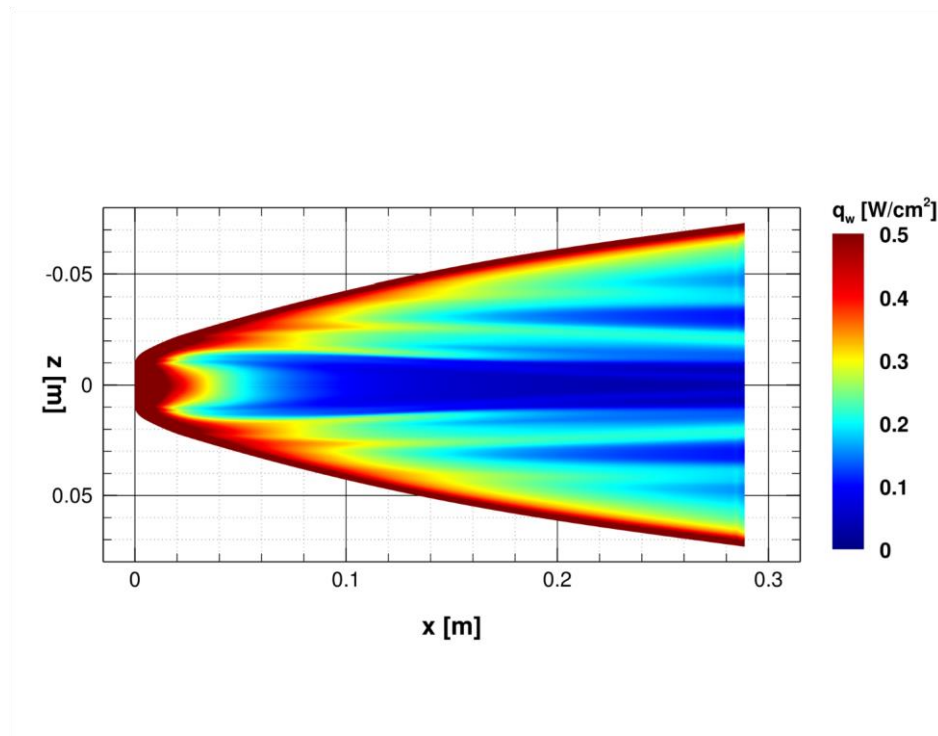


Figure 2.2. Surface heat flux contour plot on subscale BOLT geometry.

This image (in Kostak et al. [19] and shown later in this report), while plotted on a different scale, does reveal a similar inboard set of streaks to the present computation. The wind tunnel data also shows a dim inboard streak very near the centerline that the present analysis also captures. The heating results are further investigated in Section 3 below.

2.3 LPSE Analysis

After producing the basic state, an in-house code is used to extract inviscid streamlines. Paths are extracted across the majority of the top surface, ranging from the outboard edge of the top surface to very near the large top surface vortex described in the previous section. Each of these paths is examined for the presence of instabilities including 2nd mode, stationary crossflow, and traveling crossflow. It is noted that while vortex paths more accurately model the evolution of the crossflow disturbance [7], streamlines are used in the present study in order to identify the relevant mechanisms at play on this geometry.

2nd Mode. The first instability found on the geometry is the 2nd mode. This acoustic instability is found towards the outboard region of the top surface of the geometry. This is due to the streamlines in the region having less curvature than those closer towards the centerline, meaning that the flow is more 2D in nature. Figure 2.3a is a top-down view of a contour plot that shows the amplification of the 2nd mode, and results are reflected across the central axis of symmetry in this as well as all subsequent contour plots. This plot shows the locally most amplified 2nd modes along each path. As can be seen from the relatively low maximum N-factor of just over

1.5, the 2nd mode undergoes very little amplification over the length of this geometry. Despite this low amplification, a wedge of the largest N-factors can be seen towards the outer edge of the top surface.

In addition to N-factors, the frequency of the 2nd mode is also examined, and this is shown in a similar contour plot in Figure 2.3b. Like Figure 2.3a, this plot illustrates the frequencies of the most amplified 2nd mode at each point for each of the streamlines plotted. It can be seen that the most amplified 2nd modes have very high frequencies at the beginning of the streamlines. This is due to the very small boundary layers that lie just behind the swept leading edge. As the flow progresses along the geometry, the boundary layer gradually thickens, and this leads to the most amplified 2nd modes decreasing in frequency.

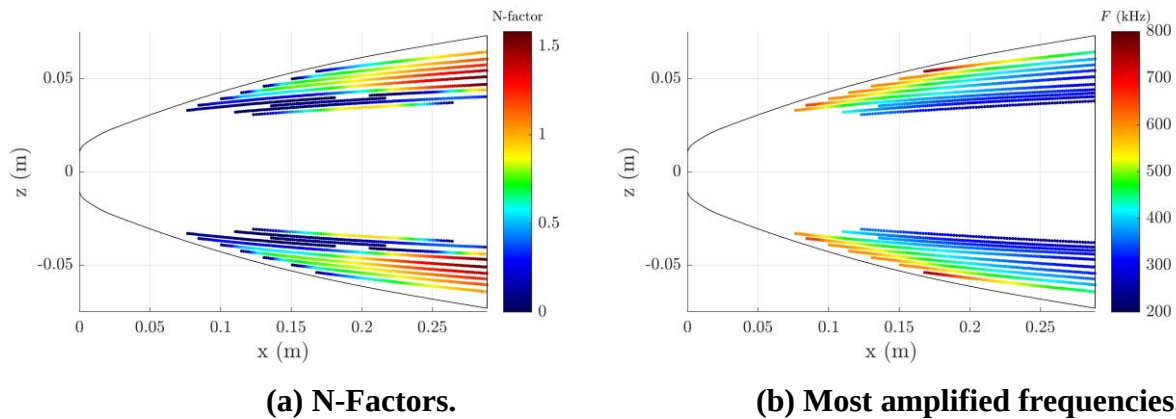


Figure 2.3. 2nd mode LPSE contour plots on subscale BOLT.

Crossflow. In addition to the 2nd mode, it is also found that both stationary and traveling crossflow are unstable on the top surface of subscale BOLT. This is due to the pressure gradient created by the azimuthal curvature of the top surface. This curvature causes a twisting of the streamlines, resulting in a velocity profile that is susceptible to both types of crossflow. In the present analysis, a constant wavelength is assumed for the paths due to the very large radius of curvature on the top surface.

Figure 2.4 shows a top-down N-factor contour plot of the most amplified stationary crossflow instabilities found on the top surface. The most amplified path is found to have an N-factor of just over 5. This path resides toward the middle of the top surface, and the wavelength corresponding to the most amplified disturbance is approximately 3.5 mm at the back of the vehicle. Similar to the results found for the 2nd mode, the largest stationary crossflow N-factors form a wedge in the middle of the top surface. This wedge is similar to what was seen on both the flight and wind tunnel models of HIFiRE-5 [20]. Over the entirety of the top surface, amplified wavelengths were found to be in the range of ~ 1.5 to 4.2 mm.

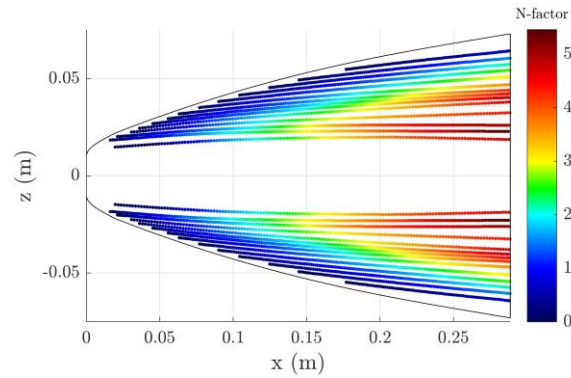


Figure 2.4. Stationary crossflow LPSE N-factor contour plot on subscale BOLT.

Moving on to traveling crossflow, Figure 2.5 shows a final top-down contour plot illustrating the N-factors for the most amplified traveling crossflow instabilities along the top surface of the subscale BOLT model. Traveling crossflow is found to be more unstable than its stationary counterpart, reaching a maximum N-factor of just over 8 along a path near the middle of the top surface. The frequency and wavelength combination leading to the most unstable traveling crossflow along this path is a frequency of around 25 kHz at a wavelength of 4.7 mm. Similar to stationary crossflow, a wedge of large N-factors can be seen in the traveling crossflow N-factor plot, however, the paths along which traveling crossflow is most amplified are slightly more outboard than those found for stationary crossflow.

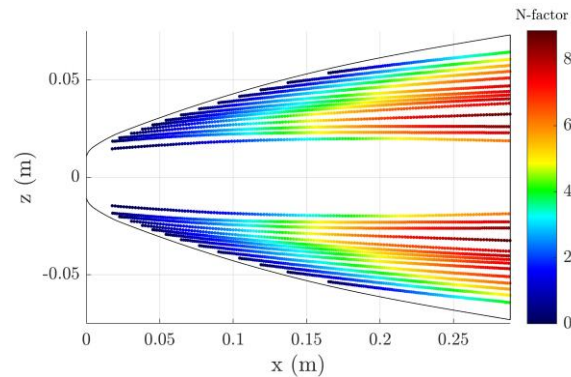


Figure 2.5. Traveling crossflow LPSE N-factor contour plot on subscale BOLT.

2.4 Conclusions

This work is focused on the analysis of the challenging hypersonic subscale model of the BOLT geometry. The heat flux on the top surface of the model is compared to DNS and experiments, and LPSE is used to determine the relevant instability mechanisms at play on the top surface of the geometry.

It is shown that the heat flux resulting from the basic state compares favorably to both DNS as well as experiments, with a number of heating features present in all three cases. However, there

are differences, and the implication of these differences among the heating results is under further investigation within our group.

LPSE is also used in order to uncover relevant instability mechanisms present on the top surface. Instabilities present include the 2nd mode, stationary crossflow, and traveling crossflow. The 2nd mode varies over a large range of frequencies due to varying boundary-layer height over the top surface of BOLT. In addition, this instability mechanism does not see large amplification, reaching a maximum N-factor of less than 2 at frequencies of around 300 kHz. By contrast, the crossflow instabilities see much higher amplification, with stationary crossflow being amplified up to an N-factor of more than 5 and traveling crossflow being amplified up to an N-factor of more than 8. The most amplified disturbance for stationary crossflow possesses a wavelength of around 3.5 mm at the end of the vehicle. For traveling crossflow, the most amplified disturbance has a frequency/wavelength combination of around 25 kHz/4.7 mm. The most amplified crossflow disturbances are also located farther inboard than those of the 2nd mode, likely due to the 2nd mode favoring flow that is more 2D in nature than that of the crossflow instability.

3. BASIC-STATE VERIFICATION STUDIES AND OFF-DESIGN FLIGHT-TEST CONDITIONS

3.1 Analysis Overview

The objectives of this part of the work are as follows:

- Boundary-layer stability analysis begins with an accurate laminar basic-state solution that is iteratively converged, grid independent, and disturbance free. Stability analysis is well known to be highly sensitive to the flowfield and boundary condition. Using the complex BOLT configuration, the first objective is to study the convergence behavior of a widely used computational tool to generate basic-state solution for both flight and sub-scale wind-tunnel conditions.
- The second objective is to study the BOLT flowfield and boundary-layer stability behavior under off- nominal flight conditions of pitch and yaw.

The geometries, meshes, and run conditions for this part of the work are presented first. Then, grid convergence of the BOLT geometry at 3 different operating conditions is discussed. Following this, a study of the BOLT flowfield at off-nominal conditions is performed. The work then analyzes the stability of these off-nominal flowfields using the linear parabolized stability equations (LPSE). Finally, remarks on the analysis and concluding thoughts are presented.

3.2 Geometries

To reiterate from above, the BOLT geometry has a wedge-like shape with a cylindrical nose tip, and this nose tip leads into four highly swept leading edges that span the length of the geometry. The broad top and bottom surfaces between the leading edges are the main experimental

surfaces, and these regions are where the majority of analysis is focused for work on the BOLT project. These surfaces also have slight concave curvature in the spanwise direction. Similar surfaces, known as gutters, span the side of the BOLT geometry, but these gutters are much more spanwise-concave. This design choice was made in order to use the gutters to isolate flow between the top and bottom experimental surfaces so that two different experiments could be performed on both sides. The BOLT geometry has two planes of symmetry which means that one quarter of the geometry can be simulated at nominal conditions and one half of the geometry can be simulated when pitching or yawing to save on computational cost. The flight geometry is 0.865 m long, and the height and width of the configuration are 0.256 and 0.438 m, respectively. See Wheaton et al. [10] for additional details on the BOLT project and geometry.

The front portion of the BOLT-II geometry is identical to BOLT, with additional acreage added aft to bring the final length to 1 meter. The goal of this geometry is to promote natural turbulence as a result of the additional length as compared to BOLT. The BOLT-II geometry is noted because the meshes and solutions for the convergence study portion of this work are computed on the BOLT-II geometry, but results are only considered up to the exit plane of BOLT. This choice to use the BOLT-II geometry was made such that lessons learned from this BOLT study can be directly extended to BOLT-II. The off-nominal work uses a BOLT, rather than BOLT-II, grid. Figure 3.1 shows a CAD image of the BOLT and BOLT-II geometries, separated by a green line.

Since all analysis in this work is carried out on the front “BOLT” section of BOLT-II, the term BOLT will henceforth be used to refer to the results. In addition to this flight configuration of BOLT, a 33% scale model of this geometry is also considered that has been tested in hypersonic wind tunnels [21]-[22].

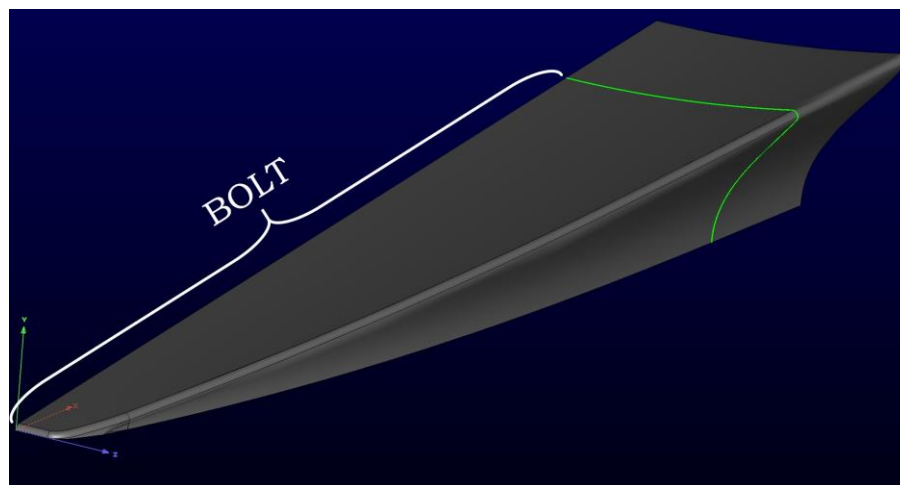


Figure 3.1. BOLT/BOLT-II geometry and coordinate axes.

3.3 Meshes

Using the BOLT geometry, a set of 5 meshes are created that range in grid density to study the flowfield at differing grid resolutions. These grids are constructed using identical grid topology (i.e. the only difference in the grids is refinement) and encompass one quarter of the geometry. To refine each grid, the cell counts are scaled up by $\sqrt[3]{2}$ (rounding where necessary to obtain integers) while the mesh spacing is simultaneously decreased by the same scaling factor. The first grid on BOLT is created using about 50 million cells. This mesh contains wall-normal clustering in order to resolve the boundary layer and shock and has a spacing at the wall of 1 μm . This grid is then refined 3 times to create meshes of 100, 200, and 400 million cells. Due to streamwise stretching, approximately 92% of the cells in each grid are contained within the original BOLT geometry with the remaining percentage residing in the extended length of BOLT-II. In addition to these 4 grids, one additional mesh is created from the 400 million cell mesh. This mesh roughly doubles the grid density in the azimuthal direction in the vicinity of the centerline structure by use of azimuthal grid clustering while also adding additional wall normal clustering. This mesh has roughly 150 wall-normal cells spanning the height of the centerline structure that is discussed in subsequent sections. As the outboard regions were seen to be well modeled by smaller-cell-count meshes, this alternative grid was created by focusing more cells from the 400 million cells mesh in the region of the centerline structure rather than uniformly increasing the mesh density in all regions. This clustered mesh will be henceforth referred to as 400HR (high resolution).

An additional 157 million cell mesh is used for the off-nominal condition simulations. This mesh was created prior to the present grid resolution study and was designed for LPSE analysis in the outboard region where amplified disturbances were observed in both DNS and in wind-tunnel studies [18],[22]. Moyes & Reed conducted a basic-state grid study for this analysis and determined that this resolution was adequate for their stability analysis [17]. This mesh spans to the aft end of BOLT and does not encompass the BOLT-II geometry.

Table 3.1 lists the sizes of the meshes used in the present study. The streamwise cells are measured along the centerline, and the azimuthal cells are measured at the end of the geometry. Due to the presence of a 3-cell singularity near the nose needed to wrap the hexahedral cells to the BOLT geometry, the mesh sizes are not equal to the product of these cell counts. Nevertheless, the point counts are given in order to provide a sense of the cell distribution.

Table 3.1. BOLT meshes

Mesh	Streamwise cell count	Spanwise cell count	Wall-normal cell count
50M	670	280	338
100M	845	352	426
200M	1066	446	537
400M	1348	567	677
400HR	1348	567	677
157M	900	700	300

This set of meshes has been manually aligned to the shock for both of the full-scale cases tested (both shocks are close in shape and location due to the Mach number). In addition, the grid resolution study meshes have been physically scaled down to 33% for the wind tunnel case, where the shock is also captured, but does reside slightly closer to the body due to the higher Mach number. That is, the grids themselves are scaled down, but the cell counts remain the same on the wind-tunnel-scale meshes.

3.4 Run Conditions

As previously mentioned, the BOLT grids are run at both flight conditions and wind-tunnel conditions. The flight conditions used are a set of nominal ascent conditions used in prior work by Moyes et al. [17] and by Thome et al. [18] at a length Reynolds number of about 10 million as well as an additional set of conditions used by Li et. al. [23] with a lower length Reynolds number of around 5.7 million. These will be referred to as the high and low Reynolds number ascent cases, respectively. Note that the low Reynolds number case in the present work refers to the set of conditions referred to as “Flight-Ascent-Higher Reynolds number” or FAH in [23]. The off-nominal simulations detailed in subsequent sections are run at the present high Reynolds number ascent conditions. In addition to flight conditions, a set of conditions from a wind tunnel test campaign at Texas A&M University is also used for the 33% scale BOLT model. Due to the model size, this set of conditions results in an even lower length Reynolds number of around 2.9 million. All of these conditions are listed in Table 3.2.

Table 3.2. Freestream conditions for BOLT simulations

Case	M_∞	P_∞ (Pa abs)	T_∞ (K)	T_w (K)	$Re' (1/m)$	Re_L	$L(m)$
High Re ascent case	5.20	6640.79	218.047	400	11.52×10^6	9.97×10^6	0.865
Low Re ascent case	5.37	3735.4	219.100	400	6.60×10^6	5.72×10^6	0.865
Wind-tunnel scale	6.00	564.3	51.600	300	9.88×10^6	2.85×10^6	0.288

3.5 Grid Resolution Studies

3.5.1 High Reynolds Number Ascent Case

Figure 3.2 shows the flowfield of the high Reynolds number ascent case using the 400HR grid. NASA’s DPLR CFD code [11] is used to compute these solutions using MUSCL Steger-Warming fluxes. This figure shows several slices of u -velocity contours on the underlying BOLT geometry. This figure follows the style of figure 7 in [18], and the contours are matched as best as possible using that figure. A number of features can be seen in the flowfield, including a large centerline mushroom structure and adjacent vortex. These structures are also seen in the

aforementioned figure from [18], and their shapes, locations, and extents both vertically and spanwise look very similar between the two figures, with only minor differences seen in the coloring of the centerline mushroom structure. Although a slight modulation of the boundary layer is observed in the outboard region, around 0.1 m. from the centerline, no additional vortices are observed in the solution on the top surface or in the gutter.

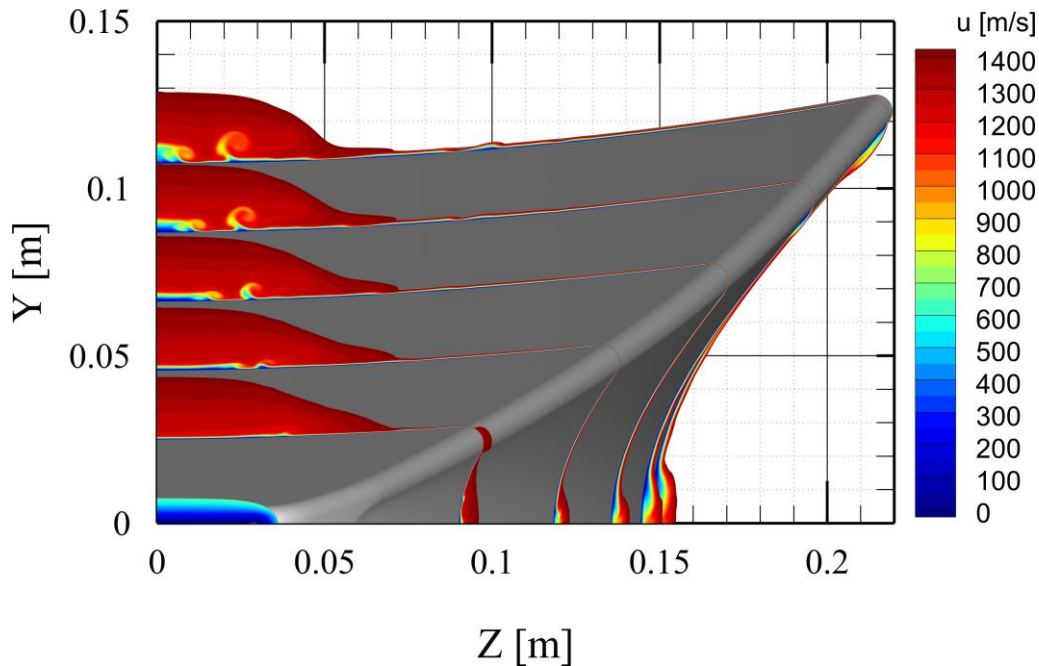


Figure 3.2. u -velocity contours on the 400HR grid.

Looking now at the other grid resolutions, Figure 3.3 shows a comparison between the 5 different convergence study grids. These and additional comparisons show lines of constant u -velocity at a slice located at the end of the BOLT geometry, 0.866 meters downstream of the nose. In this coordinate system, X (not shown) is aligned with the streamwise direction, pointing downstream, Y measures from the semimajor plane of symmetry on BOLT, pointing vertically up, and Z completes the right-handed coordinate system, pointing out the right side of the geometry when facing downstream. Each grid resolution is shown with different colored lines. The 50M, 100M, 200M, 400M, and 400HR solutions are shown using red, green, blue, magenta, and black lines, respectively. On these and subsequent figures, color scales and isoline levels are provided where possible in order to facilitate additional verification for future computations. By examining convergence at the aft end of BOLT, any differences between the solutions will be maximized and thus provide a strict test for grid convergence.

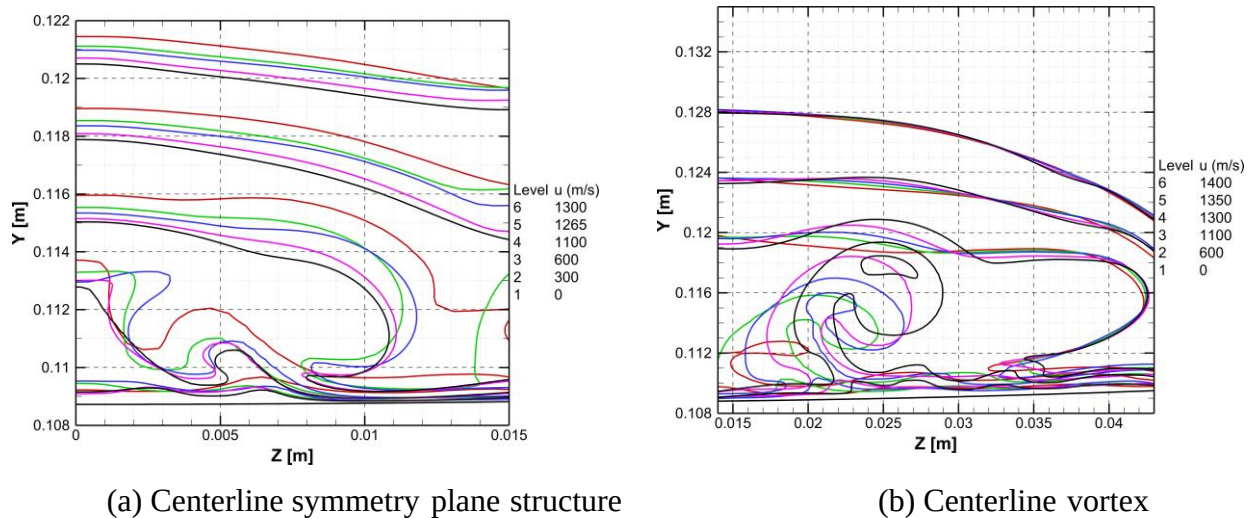


Figure 3.3. Inboard u -velocity contours at the end of the BOLT flight geometry. Red, green, blue, magenta, and black lines correspond to 50, 100, 200, 400, and 400HR million cell grids.

Figure 3.3 shows results of a convergence study of the centerline vortex structure seen in Figure 3.2 and in figure 7 of [18]. Figure 3.3a shows the back slice in the immediate vicinity of the top surface symmetry plane. The solutions appear to be converging for the different grids. However, analysis of this region is currently ongoing in order to further converge this innermost region of the top surface.

The next image, Figure 3.3b, shows the region located slightly farther outboard compared to the previous image (Figure 3.3a) and also located at the aft end of BOLT. Here, a distinct vortex can be seen forming near the wall. The higher grid resolutions seem to show the vortex becoming more developed such that it does not yet converge for the meshes used. However, this lack of convergence appears to be confined to the region between the symmetry plane out to about 0.04 m. azimuthally from the centerline of BOLT. As can be seen in the figure, all meshes converge very well as the contour lines move away from the vortex. This can be seen, for example, in the uppermost velocity contour line (1400 m/s), and in the outer portion of the 3rd highest contour line (1300 m/s). Similar to the last figure (Figure 3.3a), this comparison shows that the centerline region of BOLT is very challenging to resolve and requires further analysis under these operating conditions prior to the application of a stability analysis such as spatial biGlobal. Since convergence is observed in the outboard part of this geometry, stability analysis using LPSE is possible and is discussed in subsequent sections.

Figure 3.4 shows a similar comparison to the previous figure, but the focus here is on the outboard region. Away from the centerline region, wind-tunnel-scale tests have observed regions of possibly transitional behavior [21]-[22]. This is the region where LPSE has been used in past work [17],[24] and will be used in the present analysis.

Figure 3.4a shows the region of the flow slightly outboard of the previously mentioned centerline vortex. For this and the following figure, the plot lines have been drawn thinner in order to be able to show smaller differences between the solutions. Here, convergence of the flow is

achieved moving beyond about 0.04 m. from the centerline. The highest mesh resolution outboard of this azimuthal station can be seen to converge with sub-millimeter accuracy. In some cases, the 400M and 400HR grids (magenta and black lines) lie nearly on top of one another.

Moving yet further outboard, Figure 3.4b shows a final comparison for all of the computed solutions. Here, the center of the figure highlights the slight boundary layer modulation that has been seen in prior work with DPLR. This modulation was also shown to evolve into stationary crossflow waves in the Minnesota DNS [17]. In these basic states, however, the boundary layer in this region remains relatively constant, only seeing sub-millimeter differences as the resolution is increased, and at the highest resolution (400HR, black lines), the amplitude of this modulation begins to decrease which is indicative of convergent behavior.

3.5.2 Low Reynolds Number Ascent Case

Moving on now to the low Reynolds number ascent case, Figure 3.5 shows a set of Mach number contours at an axial station of 0.6495 m. on the 400HR grid. downstream. Here, both dimensions are nondimensionalized using the length of the BOLT flight geometry, 0.866 m. This location is chosen to coincide with figure 9 of Li et al. [23]. In the figure, two main structures can be observed. A thick structure forms immediately adjacent to the centerline, and an additional

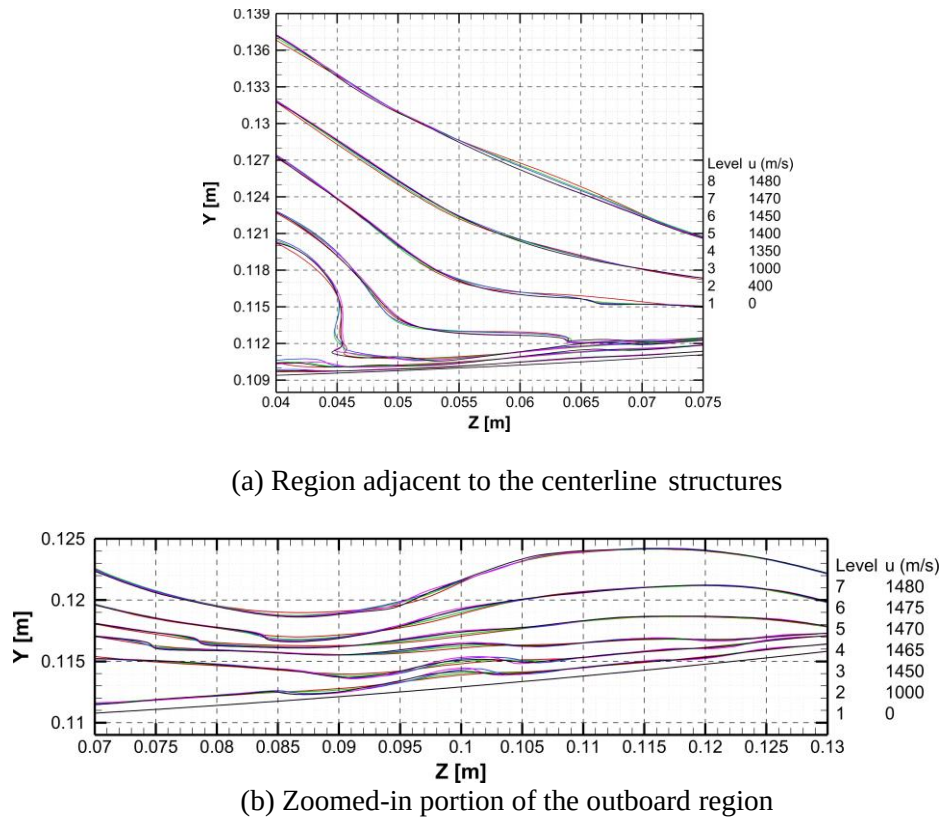


Figure 3.4. Outboard u -velocity contours at the end of the BOLT flight geometry. Red, green, blue, magenta, and black lines correspond to 50, 100, 200, 400, and 400HR million cell grids.

vortex forms slightly outboard of this centerline structure centered around $Z/L = 0.03$. Outboard of these structures, no additional vortices or boundary-layer modulations are observed. Matching the contour coloring and comparing this figure to figure 9 in [23] reveals good agreement between the structures in both figures, including shape, location, and vertical and spanwise extent. It is noted that some differences are observed in the shapes between the two sets of structures, and no additional wave-like structures are seen in the present solution whereas several wave-like structures are observed near the surface at azimuthal stations of $Y/L = 0.035$ and 0.045 in figure 9 of [23].

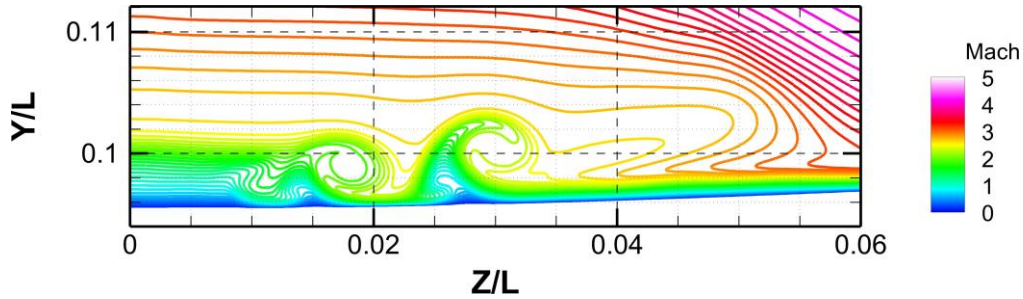
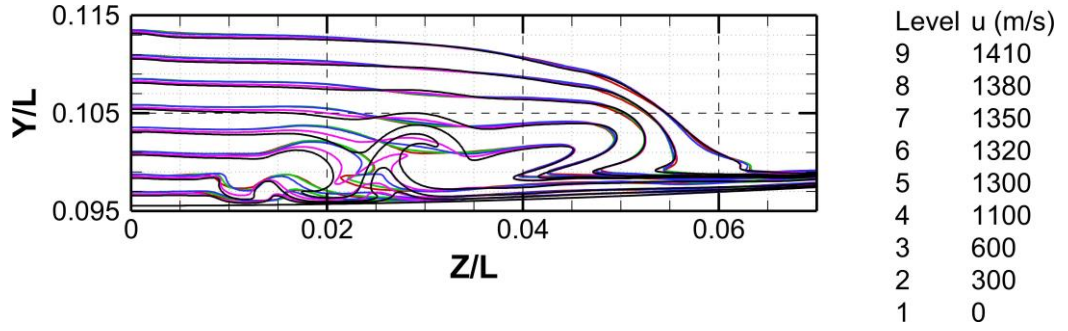


Figure 3.5. Centerline Mach number contours on the 400HR grid at 0.6495 m. ($X/L = 0.75$).

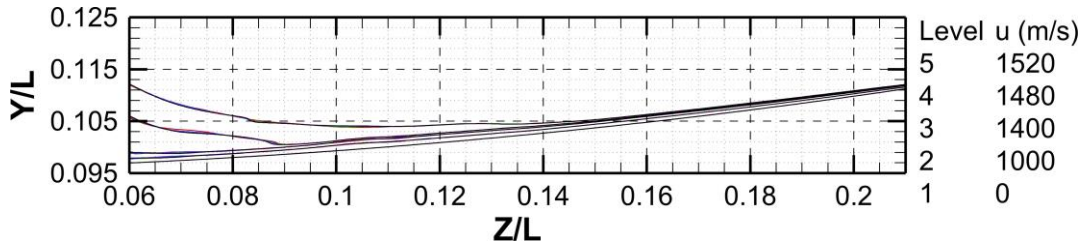
Maintaining the coloring from the previous section, Figure 3.6a now shows a similar region of the BOLT flowfield at the same axial station for all 5 of the convergence-study grids. Here, it can be seen that differences between grid resolutions are observed in the vicinity of the centerline structure and adjacent vortex. Similar to the high Reynolds number ascent case, this region is not yet converged, even at the highest grid resolution simulated, and analysis of this region is ongoing. Outboard of an azimuthal station of around $Z/L = 0.04$, all of the solutions begin to converge. Figure 3.6b shows all solutions across the top surface of BOLT. The figure is rendered in high resolution using very fine lines in order to show how close the solutions match one another. In this region, like the previous ascent case, sub-millimeter accuracy is achieved using all of the grid resolutions, and no additional vortices are found in this region.

3.5.3 Wind-Tunnel Case

An additional set of basic states are computed on 33% scale versions of the 50, 100, and 200 million grids mentioned in the prior sections, and these basic states are run using existing wind-tunnel conditions from Li et. al. [23] listed in Table 3.2. A similar grid resolution analysis is presented here to examine basic state convergence at the wind tunnel BOLT geometry conditions, but this work does not examine stability on the wind-tunnel geometry.



(a) Inboard region



(b) Outboard region

Figure 3.6. u -velocity contours at 0.6495m. for low Reynolds number ascent case. Red, green, blue, magenta, and black lines correspond to 50, 100, 200, 400, and 400HR million cell grids.

Figure 3.7 shows Stanton number distributions near the back of the geometry. The axes on these figures follow those of figures 5a and 5b in [23], and the horizontal axis is nondimensionalized using the length of the 33% BOLT wind-tunnel model (0.28867 m.). In this figure, the red, green, and blue lines correspond to the 50, 100, and 200 million cell basic states. Outboard of $Z/L = 0.1$, all 3 solutions lie nearly on top of one another, demonstrating the low resolution needed to resolve the flow outboard of this azimuthal station. Inboard, some differences among the 3 solutions are observed especially around the three peaks in Stanton number around azimuthal locations of $Z/L = 0.025$, 0.045 , and 0.08 . Comparing these figures to figures 5a and 5b in [23], good agreement is found between the present basic states and both DNSs shown in that work. In addition, these 3 basic states also fall within the scatter between the two sets of experimental results indicating good agreement with experiments.

Figure 3.8 shows basic state u -velocity contours near the back of the geometry ($X/L = 0.84$). In particular, these contours show the region of flow adjacent to the centerline of the geometry, and the red, green, and blue lines correspond to the 50, 100, and 200 million cell geometries, respectively. Note that both of the axes are nondimensionalized by the model length. Figure 3.8a shows the basic states for all three different grid resolutions simulated with contours of u -velocity from 0 to 800 m/s in 100 m/s steps. For the entirety of the top surface except the region of flow that comprises the vortex structure, good agreement is obtained across all resolutions. Only inside this vortex structure near $Z/L = 0.02$ and $Z/L = 0.036$ are notable differences found between the different basic states. Then, Figure 3.8b shows a very similar set of u -velocity contours, this time

only for the 200 million cell case. Here, the velocity contours are chosen to match the results from prior results seen in figure 7 of [25] and figure 5c of [23]. Some slight differences can be seen in this basic state compared to the DNSs noted in the previous figures, but in general, very good agreement is found between the DNS results and the present basic states.

In addition to velocity contours, Figure 3.9 shows a Stanton number contour of the surface of the BOLT geometry. As before, both of the horizontal and vertical axes have been nondimensionalized by the length of the model geometry. Across the surface, a number of heat streaks can be seen, both in the region near the centerline as well as farther outboard. Comparing the present results to figure 2a from [23] reveals a number of similarities in the general streak pattern. The two dim streaks very close to the centerline are remarkably similar in both computations as are the streaks located at an azimuthal station of $Z/L = 0.125$. In addition, the bright streak located at $Z/L = 0.05$ (at the end of the model) is present in both computations at very similar azimuthal stations, but the streak found in the current computation is hotter than that found by Li et al. [23].

Further analysis of the BOLT flowfield is ongoing, especially in the centerline region, but the results shown here indicate that the use of DPLR to generate basic states and the use of LPSE in regions away from the centerline of BOLT appears to be viable. The current results presented in the outboard region further verify the prior convergence analysis using DPLR on the BOLT geometry [17] and demonstrate that basic states using DPLR converge in the outboard region using relatively low grid resolutions of 100 to 200 million cell (or even less in some regions) for both the wind-tunnel and flight geometries on BOLT.

3.6 Off-Nominal Simulations

As mentioned above, a 157 million cell mesh on a quarter of the BOLT geometry is used to compute the off-nominal basic states for $\pm 1^\circ$ and $\pm 2^\circ$ pitch and yaw at the high Reynolds number ascent operating conditions. This mesh has been used in prior work on this geometry [17],[24]. The mesh is reflected about the semimajor axis of symmetry for the pitching cases and about the semi-minor axis of symmetry for the yawing cases. These reflections double the resolution of the mesh up to about 314 million cells for each case.

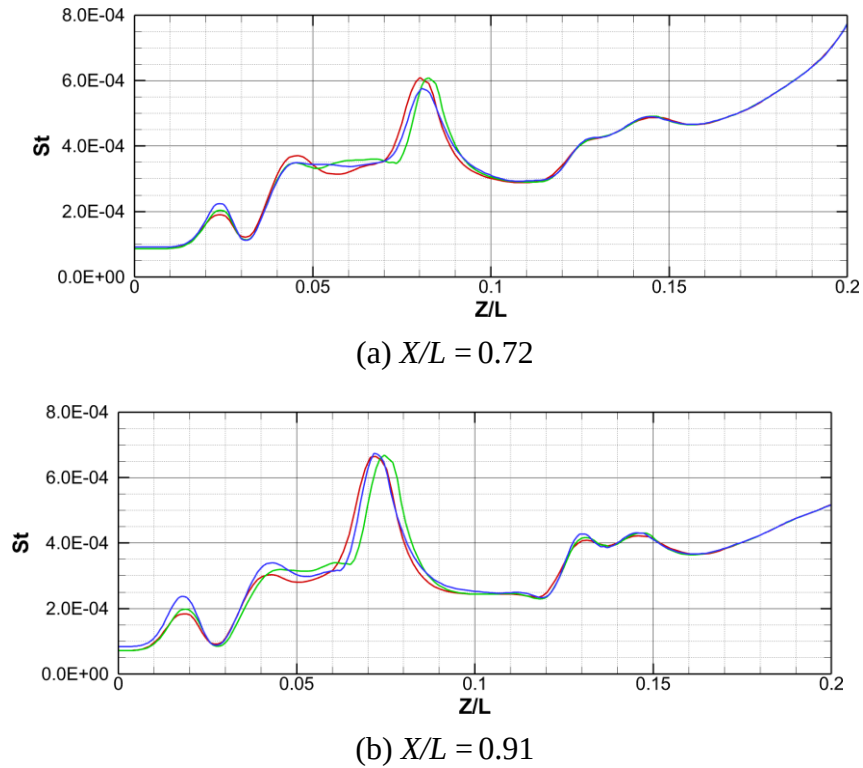
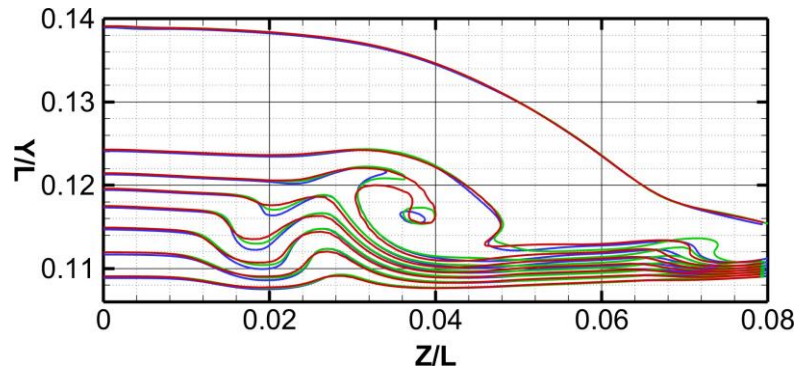
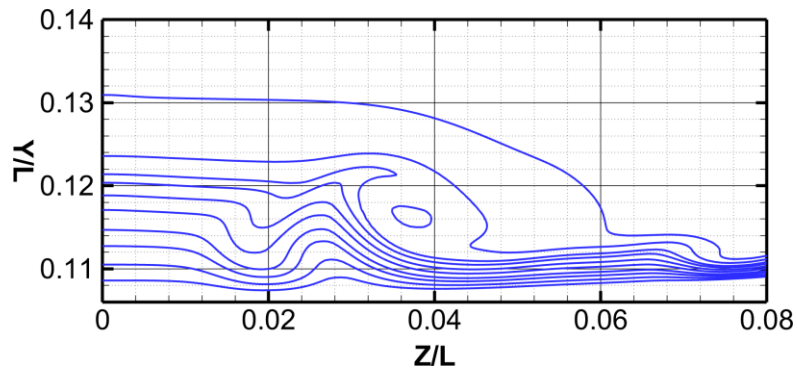


Figure 3.7. Stanton number variation on the 33% scale BOLT geometry. Red, green, and blue lines correspond to 50, 100, and 200 million cell grids.

Note that, because of the symmetry of the geometry, the windward side of a positive pitch (yaw) is the same as the windward side of a negative pitch (yaw), and the leeward side of a negative pitch (yaw) is the same as the leeward side of a positive pitch (yaw). Therefore, DPLR is used to compute the four basic states for these 8 cases ($\pm 1^\circ$ and $\pm 2^\circ$ pitch and yaw). Referring to Figures 3.10 and 3.11 for the pitching cases, the leeward side of positive pitch is located within the $(+X, +Y, +Z)$ quadrant, and the windward side of positive pitch resides in the $(+X, -Y, +Z)$ quadrant. Referring to Figures 3.12 and 3.13 for the yawing cases, the leeward side of positive yaw lies within the $(+X, +Y, +Z)$ quadrant, and the windward side of positive yaw is located within the $(+X, +Y, -Z)$ quadrant. In each of these cases, the freestream velocity vector is inclined towards the windward side of each case. This corresponds to the freestream velocity vector being rotated into the $(+X, +Y)$ direction for the pitching cases and rotated into the $(+X, +Z)$ direction for the yawing cases.



(a) Inboard region for all solutions



(b) 200M solutions with isolines tuned to match figure 5c of [23]

Figure 3.8. u -velocity contours at $X/L = 0.84$ on the 33% scale BOLT geometry. Red, green, and blue lines correspond to 50, 100, and 200 million cell grids.

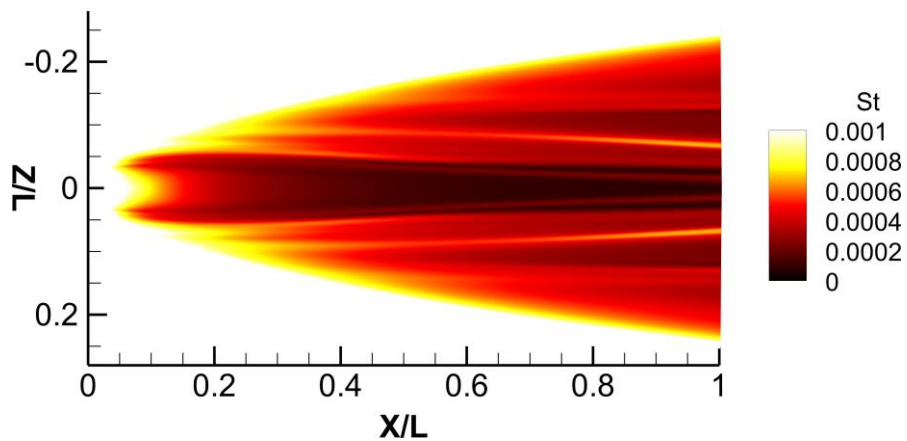


Figure 3.9. Stanton number contours for the 33% scale BOLT geometry on the 200M cell solution.

3.6.1 Pitch

Using the aforementioned visualization, Figure 3.10 shows the $\pm 1^\circ$ pitching cases, and Figure 3.11 shows the $\pm 2^\circ$ pitching cases. Looking at these figures, a number of trends can be seen looking at the centerline vortex structure, the heat flux on the main surfaces, and the flow in the gutter regions. First, the central vortex expands on the leeward side and compresses on the windward side during positive pitching. This effect has been previously noted during pitched HIFiRE-5b simulations [20]. Due to the small angles, the effect is very slight between the solutions, and larger pitch angles cause larger differences between the pitched cases and the nominal case. It is also noted that the pitching of the test article does not appear to cause any additional vortices in the outboard regions of the top surface.

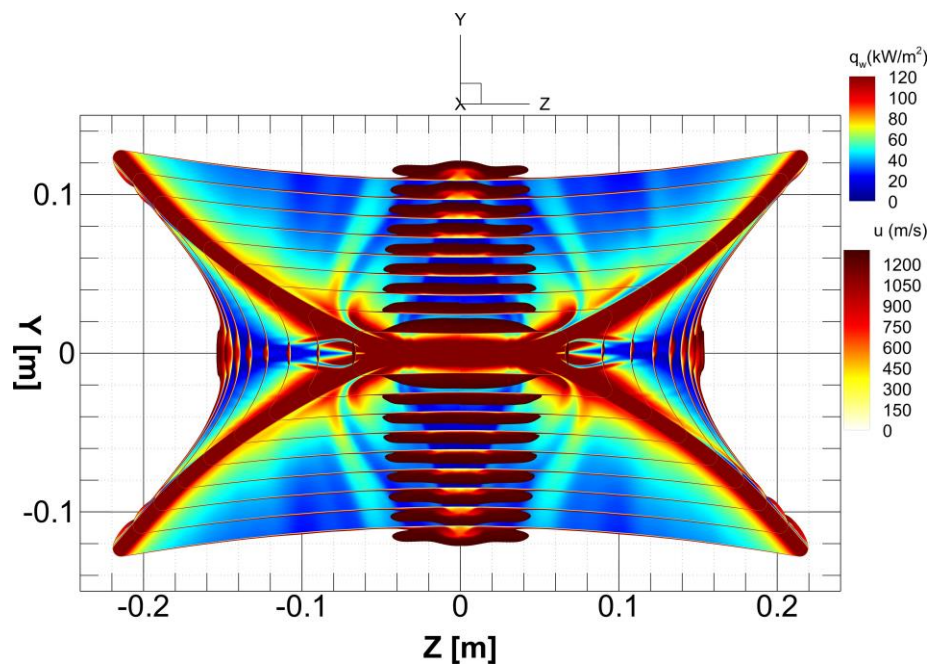


Figure 3.10. Surface contour of heat flux and axial slices of u -velocity. Solutions moving clockwise from the top right quadrant are leeward side of $+1^\circ$ pitch, windward side of $+1^\circ$ pitch, nominal solution, and nominal solution.

Next, looking at the surface heat flux, the heat streak patterns appear to become more washed out on the windward side of the positive pitch cases with the $+2^\circ$ case seeing a more washed out pattern but overall slightly higher heating compared to the $+1^\circ$ case. Interestingly, the leeward sides of both pitched solutions do not show a noticeable decrease of heating across the acreage. However, both cases do show a region of lower heat flux forming just downstream of the leading edge on the leeward side. This cooler region appears at $+1^\circ$ and grows in extent at $+2^\circ$ while also becoming cooler. At $+2^\circ$, the lower heat flux region extends to the end of BOLT creating a cooler region from 0.14 to 0.16 m. outboard of the centerline. For both pitch cases, additional heat streaks

begin to form in the outboard region of the leeward side with the $+2^\circ$ case showing more streaks compared to the $+1^\circ$ case. In addition, the leeward sides of the pitch case develop additional localized heating areas near the swept leading edge that are not present in the nominal case. For the leeward side of the $+1^\circ$ case, a localized heating area forms just downstream of the swept leading edge at about 0.08 to 0.12 m. outboard of the centerline. Moving to $+2^\circ$ of pitch causes this region to shrink, spanning from 0.08 to 0.1 m. from the centerline. However, an additional area of local heating formed from 0.1 to 0.12 m. away from the centerline. At $+2^\circ$ of pitch, both of these localized heating areas see higher heat flux compared to the $+1^\circ$ case.

Finally, several changes can be seen in the gutters as a result of the pitching. The structure residing in the center of the gutters becomes shifted towards the leeward side in each case with the $+2^\circ$ pitch case resulting in a larger shift compared to the $+1^\circ$ case. In addition, a differential fluid buildup can be seen on the gutter side of the swept leading edge. For both pitching cases, fluid can be seen accumulating on the windward side with more fluid building up at $+2^\circ$ than at $+1^\circ$ of pitch. In both instances more fluid accumulates compared to the gutter in the nominal case. A lack of fluid buildup is also observed on the leeward side of both cases with the $+1^\circ$ case showing a small accumulation of fluid and the $+2^\circ$ case showing nearly no fluid buildup.

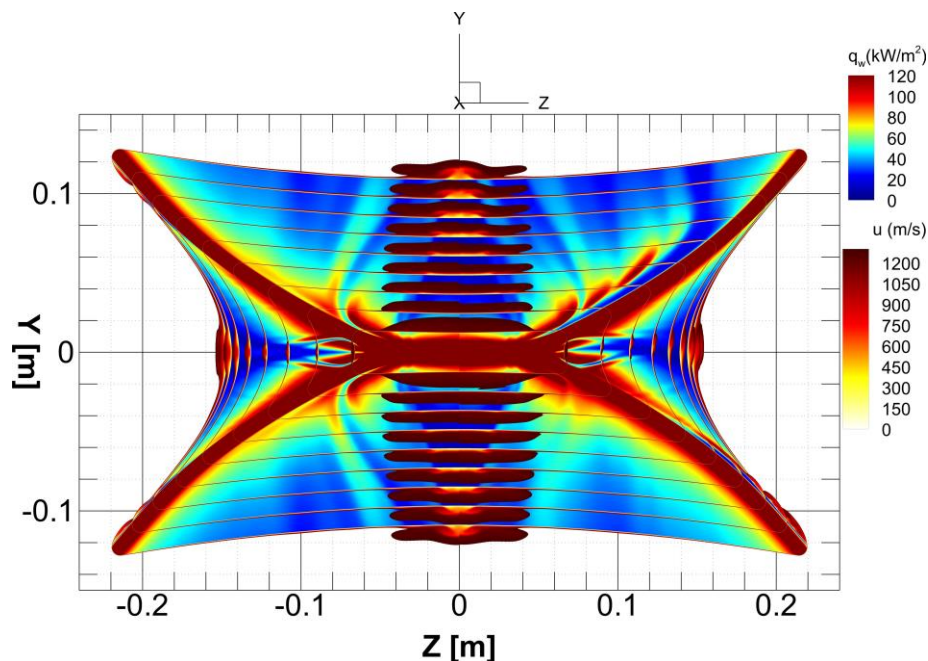


Figure 3.11. Surface contour of heat flux and axial slices of u -velocity. Solutions moving clockwise from the top right quadrant are leeward side of $+2^\circ$ pitch, windward side of $+2^\circ$ pitch, nominal solution, and nominal solution.

3.6.2 Yaw

Similar to the angle of attack cases, Figure 3.12 shows the $\pm 1^\circ$ yawing cases, and Figure 3.13 shows the $\pm 2^\circ$ yawing cases. In this set of figures, the nominal cases are located on the bottom halves of the figures, rather than on the left halves. Similar to the pitching figures, a number of trends result from the yawing of the flowfield that affect the centerline vortex, the main surface heat streaks, and the flow in the gutters. First, instead of changing in height, the centerline vortex becomes skewed as a result of the yaw. By the end of the geometry, the centerline vortex is shifted laterally by around 1.5 cm. in the $\pm 1^\circ$ yaw case and about 3 cm. in the $\pm 2^\circ$ yaw case. As in the pitching cases, no additional vortices are observed on the top surface outboard of the centerline.

Next, several differences also appear in the surface heat flux. Across all 4 yawed surfaces, only the leeward side of the $+2^\circ$ case sees a slight increase in heating over the acreage of the main surface. Yawing of the geometry has a more noticeable effect on local heating features such as the main heat streak, which spans from 0.04 to 0.06 m. (and -0.04 to -0.06 m.) azimuthally in the nominal case. The windward sides of both positive cases tend to create a more washed out main streak, but the leeward side still maintains the shape and intensity of this main streak. In addition, several additional heat streaks can be seen faintly on the windward side of the $+1^\circ$ case and appear slightly brighter on the windward side of the $+2^\circ$ case. Looking towards the leading edge, localized heating and cooling occurs similar to what was seen in the pitching cases. For the windward side of the $+1^\circ$ case, a region of high heat flux forms just downstream of the swept leading edge at about 0.08 to 0.1 m. outboard of the centerline, and this region is detached from the leading edge unlike in the nominal case. For the windward side of the $+2^\circ$ case, this localized heating region is much stronger and occupies roughly the same azimuthal extent. In addition, a region of lower heat flux occurs between these regions of high heat flux and the swept leading edge. Larger yaw angles result in lower heat flux which is similar to the effect seen during the pitch cases.

Finally, the gutter regions also show several differences as a result of geometry yaw. First, slight differences are seen in the structure at the center of the gutter for the $\pm 1^\circ$ case, but this effect is not observed at $\pm 2^\circ$ of yaw. In addition, differential fluid buildup is seen near the swept leading edge similar to the pitch cases. On gutter surfaces facing the freestream, less fluid buildup occurs compared to the nominal case, and on surfaces facing away from the flow, additional fluid accumulation is observed compared to the nominal case. As before, larger yawing angles cause larger differences in the amount of fluid accumulation.

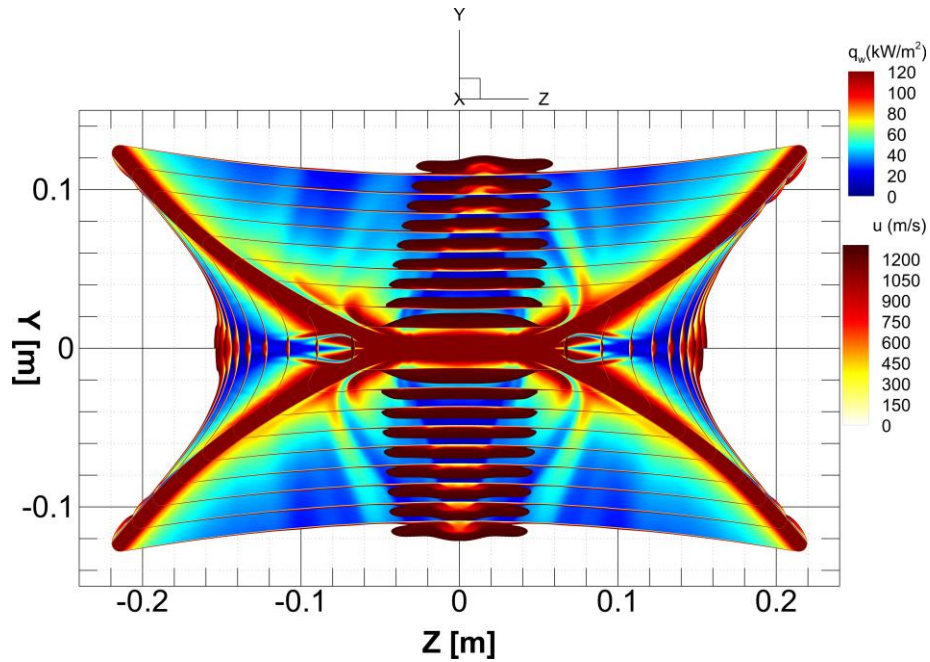


Figure 3.12. Surface contour of heat flux and axial slices of u -velocity. Solutions moving clockwise from the top right quadrant are leeward side of +1° yaw, nominal solution, nominal solution, and windward side of +1° yaw.

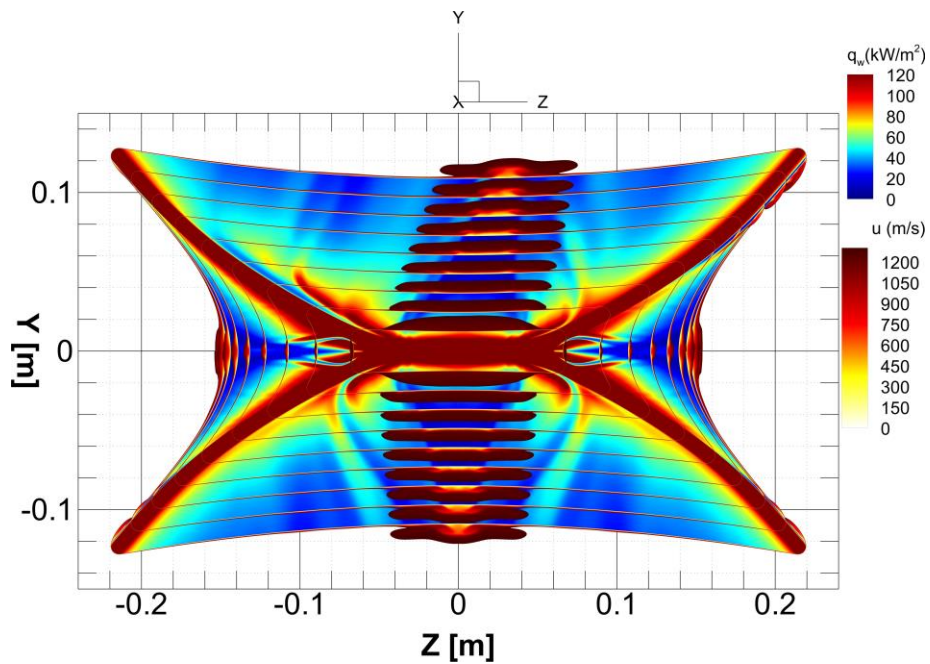


Figure 3.13. Surface contour of heat flux and axial slices of u -velocity. Solutions moving clockwise from the top right quadrant are leeward side of +2° yaw, nominal solution, nominal solution, and windward side of +2° yaw.

3.6.3 Basic-State Summary

To summarize, several trends are found across these 8 solutions. First, the central vortex is directly affected by the inclination of the flow, expanding and compressing as a result of pitch and skewing laterally as a result of yaw. Additional localized areas of high heat flux and thin cooler regions are found on sides of the geometry oriented away from the freestream. Several differences are also apparent in the heat streaks on the acreage of the geometry depending on freestream orientation. Differential fluid buildup also occurs in the gutter with additional fluid accumulation occurring on gutter surfaces facing away from the freestream and less accumulation occurring on gutter surfaces facing the freestream. For both the centerline vortex and the fluid buildup in the gutter, larger pitch or yaw angles result in greater differences compared to the nominal case, but the effect on heating is more nuanced.

BOLT is a complex geometry, and these flowfield differences seen at different levels of pitch and yaw in this work and in other work underscores this idea [23].

3.7 Parabolized Stability Analysis

3.7.1 Analysis Overview

For each of the 8 off-nominal cases discussed in the previous section, a stability analysis is carried out in the subsequent sections. In particular, the 2nd mode and stationary crossflow are examined for these solutions. Results are discussed per freestream angle and per instability in the following order: leeward side for +2°, leeward side for +1°, nominal 0° case, windward side for +1°, and windward side for +2°. The order is chosen such that trends can be examined as the pitch or yaw is smoothly varied. Results from the nominal case [17] are presented from prior work in order to better analyze trends in the data. Each figure is shown on the same set of axes and plots the locally most amplified N-factors and frequencies for 2nd modes or locally most amplified N-factors and wavelengths for stationary crossflow, as appropriate. N-factors are shown on the same scale per figure, but the wavelengths or frequencies (where appropriate) are shown on individual scales. This decision is made in order to analyze trends in N-factors across the different solutions.

Two sets of data paths are extracted for each of the five distinct cases analyzed (2°, 1°, and 0° pitch and yaw). For analysis of the 2nd mode, inviscid streamlines are extracted across the top surface of the geometry, and these streamlines are defined by 99.5% total enthalpy. To study the stationary crossflow instability, vortex paths are used, as they model the evolution of crossflow vortices more accurately than inviscid streamlines [6]. Traveling crossflow is not considered for this analysis. In each case, 25 to 30 paths are extracted in order to cover the majority of the top surface except the region in the immediate vicinity of the centerline structure due to this region breaking the azimuthal invariance assumption of the LPSE method. Away from the centerline region, wind-tunnel-scale tests have observed regions of possible transitional behavior [21]-[22]. After extracting the streamline and vortex paths from the different basic states, the EPIC

[1],[2],[4],[6] stability code is used for the stability calculations presented in the following sections.

The results are organized as follows. Figure 3.14 shows the cumulative results for 2nd mode at pitch with the subfigure sets [3.14a, 3.14c, 3.14e, 3.14g, 3.14i] and [3.14b, 3.14d, 3.14f, 3.14h, 3.14j] showing the trends in N-factors and frequencies, respectively, for the leeward side with decreasing pitch angle and then the windward side for increasing pitch angle. Figure 3.15 shows a similar type of figure for stationary crossflow at pitch with the subfigure sets [3.15a, 3.15c, 3.15e, 3.15g, 3.15i] and [3.15b, 3.15d, 3.15f, 3.15h, 3.15j] showing the trends in N-factors and wavelengths, respectively. Figures 3.16 and 3.17 show a similar set of results for 2nd mode and stationary crossflow, respectively, for the yaw cases.

Note that the N-factor color scales in each set of subfigures within a figure have been made the same so that in some instances the N-factors exceed the top value. The top value is selected to be an indicator that transitional flow may be occurring based on prior results and knowledge, that is, nominally $N = 16$ for 2nd mode and $N = 8$ for stationary crossflow.

3.7.2 Pitch Cases – 2nd Mode

Looking at Figure 3.14 and the variation in 2nd mode for pitching cases reveals a number of common results and trends due to pitching. In all cases, the locally most amplified 2nd mode instabilities have relatively high frequencies near the leading edge where the boundary layer is thinnest, with frequencies lowering towards the aft end of the geometry. Each of these also cases reveals some instances of moderate 2nd mode growth at relatively low frequencies of 400 to 500 kHz. As the geometry is pitched up beyond nominal, the windward side tends to show more organized wedges of moderate 2nd mode amplification towards the aft end of the geometry. In addition, more positive angles result in higher frequencies for the 2nd modes on the windward side, while more positive angles result in lower frequencies on the leeward side.

Conversely, as the geometry is pitched up beyond nominal, the 2nd mode is still amplified in two wedges as before, but now the leeward side tends to also see localized hotspots of 2nd mode growth near the swept leading edge. These localized regions experience large 2nd mode growth, and these regions of high growth can occur at frequencies of 400 to 700 kHz. Larger pitching angles result in lower 2nd mode frequencies in general on the geometry. In some cases, this very large growth might suggest the flowfield could locally transition. In particular, from Figures 3.14c and 3.14d and looking at the region of high amplification near the swept leading edge, it can be seen that the most amplified 2nd mode frequency remains relatively constant, allowing a few 2nd modes to tune in and grow to very large amplitudes. This tuning in of the 2nd mode is a result of a relatively constant boundary layer height in this region caused by the pitching of the geometry.

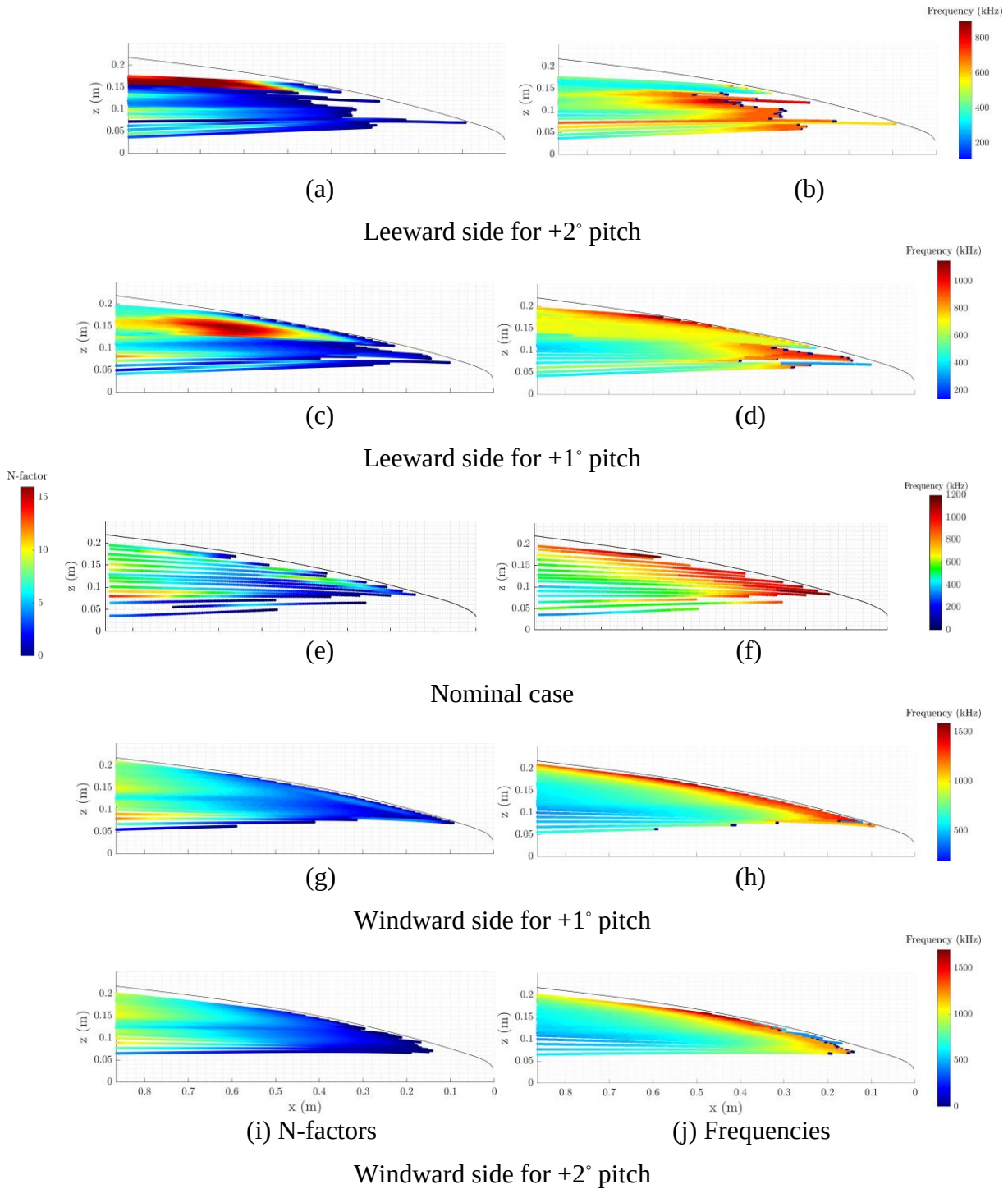


Figure 3.14. 2nd mode results for the pitch cases.

3.7.3 Pitch Cases – Stationary Crossflow

Looking at Figure 3.15, several trends and results are found for the set of stationary crossflow pitch results. Stationary crossflow amplifies in a wedge that intensifies as the geometry is pitched up beyond nominal on the windward side. The level of growth found might suggest the flowfield will transition. HIFiRE-5b stationary crossflow N-factors also formed a wedge and N-factors of 5-8 were associated with transitional flow [20]. The locally most amplified stationary crossflow wavelengths are also found to be very small next to the swept leading edge and grow larger towards the aft end. Here, the wavelengths tend to be larger towards the centerline and smaller farther outboard. Finally, greater pitch angles also tend to result in little change in wavelengths for the locally most amplified stationary crossflow instabilities on the windward side as seen in Figure 3.15.

3.7.4 Yaw Cases – 2nd Mode

Referring to Figure 3.16 and comparing now the 2nd mode results from the different yaw cases reveals several common results. It is noted that a number of paths are deflected towards and away from the centerline region, and the use of LPSE is still valid along these paths because the centerline structure itself skews toward and away from the plane of symmetry due to the yaw. Like the pitch cases, very high frequencies are still found adjacent to the swept leading edge for the locally most amplified 2nd mode instabilities, with lower frequencies found towards the aft end of the geometry as the boundary layer thickens. Looking now at the leeward side, more yaw can cause the 2nd mode to be more amplified in local areas that extend to the aft end of the geometry, and in general, the locally most amplified 2nd mode frequencies are found to increase with greater yaw. However, this amplification is rather moderate. The opposite is true on the windward side where more yaw results in lower 2nd mode growth at lower frequencies.

Further analysis of the windward side sees the appearance of localized hotspots of 2nd mode growth near the swept leading edge with increasing yaw. These localized regions experience large 2nd mode growth, and these regions of high growth can occur at frequencies of 400 to 500 kHz. The associated N-factors are rather large, and these 2nd modes might be large enough to cause the flow to transition locally.

3.7.5 Yaw Cases – Stationary Crossflow

Referring to Figure 3.17 and looking at this final set of results reveals several trends for stationary crossflow across the yaw cases. In general, moderate to large stationary crossflow growth is seen at all angles of yaw with associated N-factors possibly being sufficient to cause transition. A large wedge of amplification is observed similar to the pitch results, increasing in extent on the leeward side with increasing yaw. As before, very small wavelengths are observed next to the swept leading edge for the locally most amplified stationary crossflow instability with

larger wavelengths observed on paths closer towards the centerline. For the leeward side, higher yaw angles result in greater path deflection away from the centerline of the geometry, and the converse is true of the windward side. That is, paths on the windward side deflect further towards the centerline at higher yaw angles.

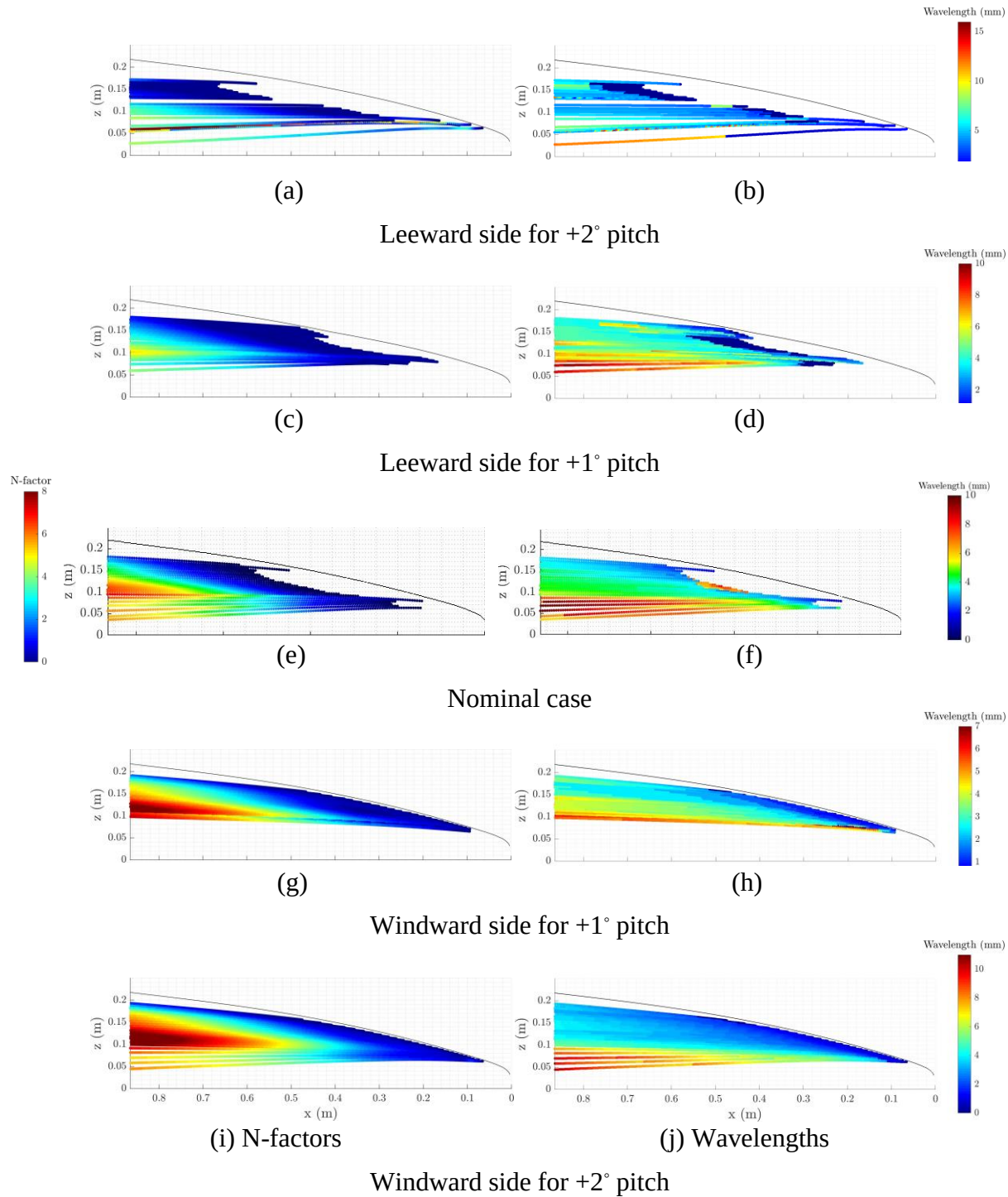


Figure 3.15. Stationary crossflow results for the pitch cases.

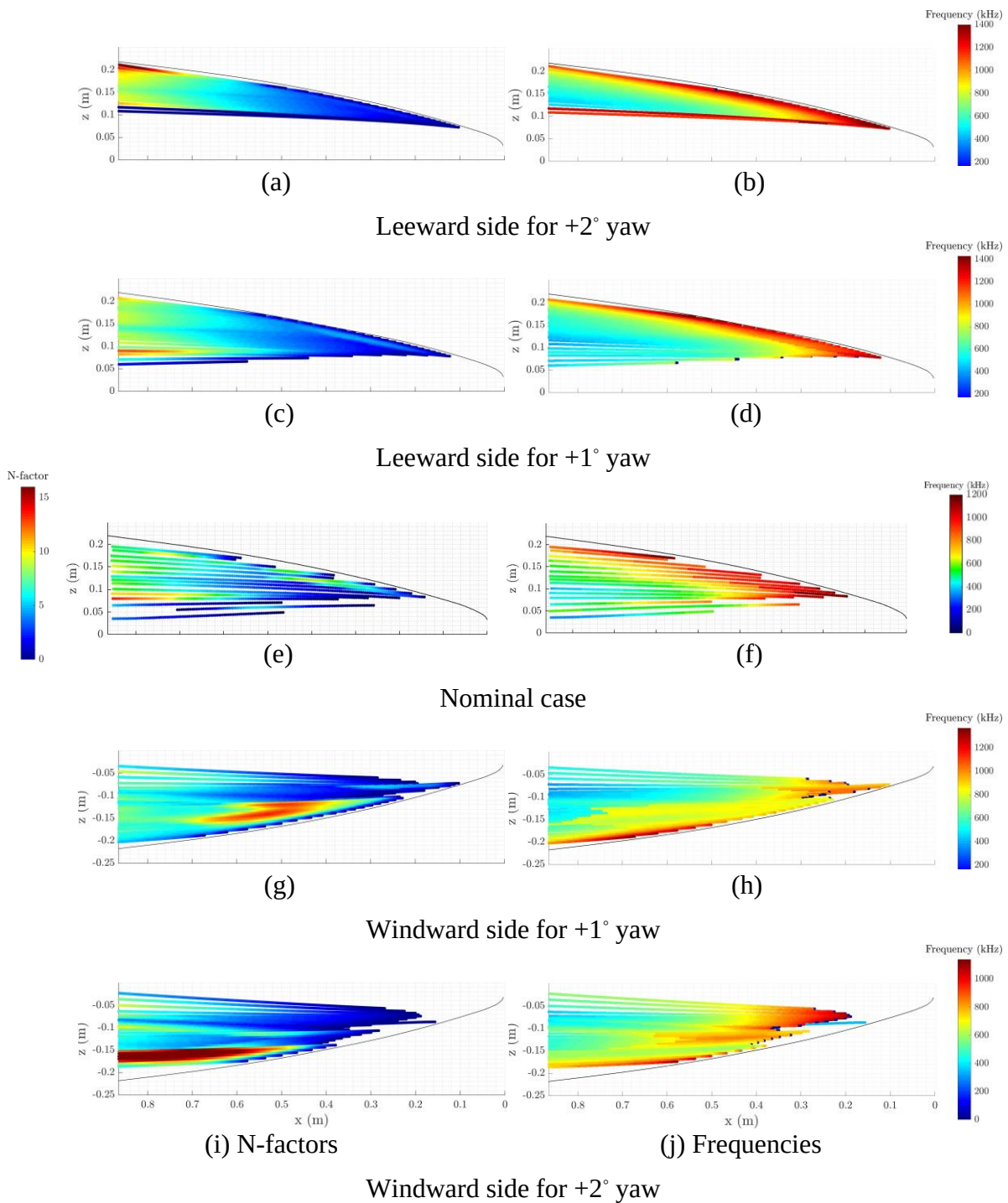
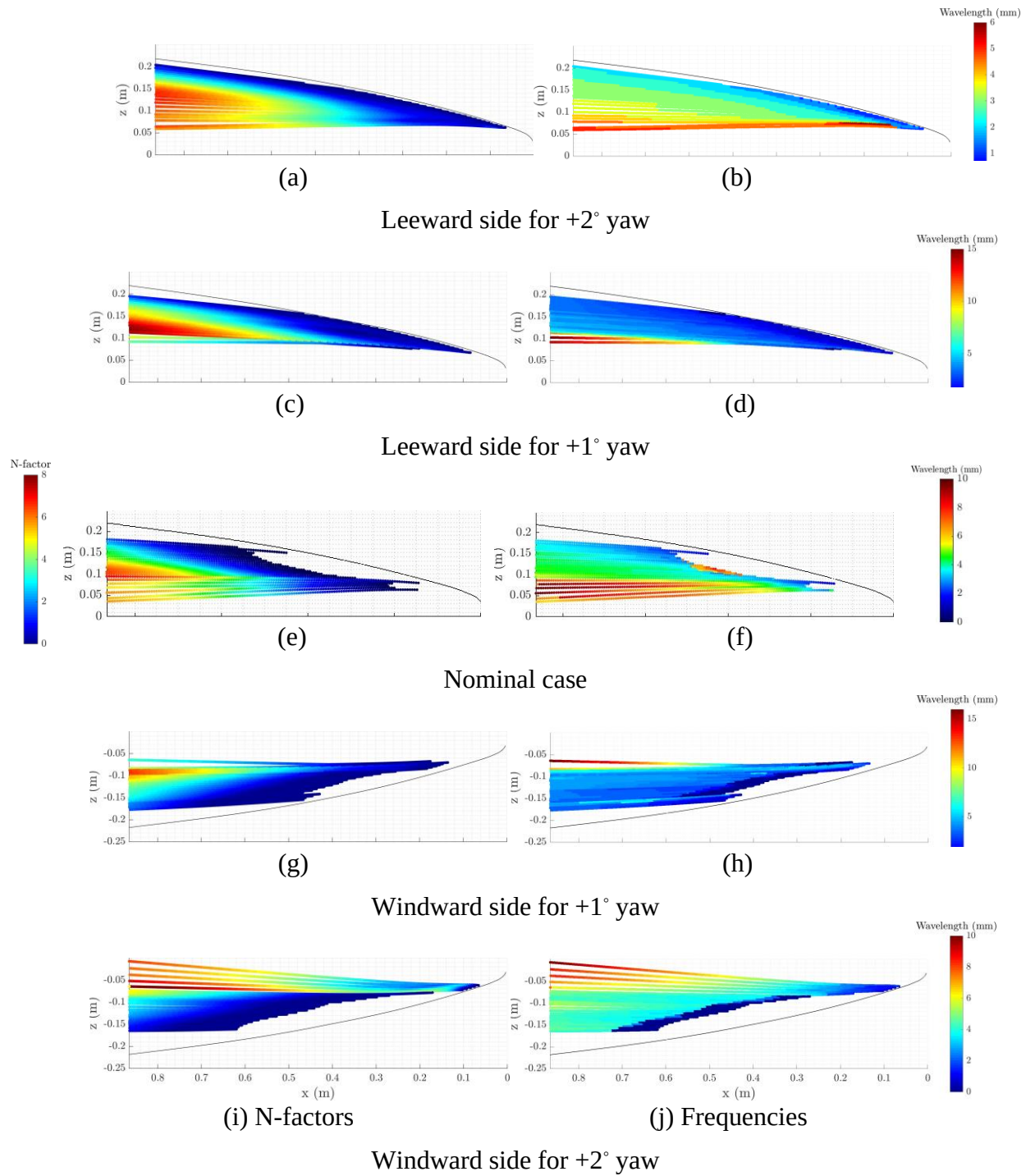


Figure 3.16. 2nd mode results for the yaw cases.



3.8 Analysis Remarks and Conclusions

3.8.1 Basic States

This part of the work has considered several different aspects of BOLT including the grid convergence of the flowfield at different freestream conditions. Two different full-scale flight conditions are considered along with a set of wind tunnel conditions for a 33% scale wind-tunnel of the BOLT geometry. In addition, the BOLT geometry is pitched and yawed systematically, and the effect of these off-nominal conditions on the different instabilities present in the flowfield is analyzed.

For the high Reynolds number ascent condition at a length Reynolds number of about 10 million, grid convergence on BOLT was examined using 5 different meshes that have identical grid topology. Using the DPLR hypersonic flow solver with MUSCL Steger-Warming fluxes, it is found that the centerline structure and adjacent rollover are similar in location, shape, and vertical and spanwise extent to prior DNS results [18], but a quantitative examination of velocity contours shows that the results are not yet fully converged at the highest grid resolutions used. However, these two structures only span 3 cm. from the centerline at the exit plane of the BOLT geometry, and the lack of quantitative convergence is found to extend only about 4 cm. from the centerline. Outside of this azimuthal location, the flowfield converges with sub-millimeter accuracy. This is the region where LPSE has been used in prior work [17] and is used in the present analysis [24]. Furthermore, no additional vortices are found in the outboard region in the present solutions, even with the highest resolution meshes.

A similar analysis was carried out for the low Reynolds number ascent condition at a lower length Reynolds number of around 5.7 million. Here, a grid convergence study using the same methodology showed similar results to the previous case. The centerline structure and adjacent rollover are once again similar in location, shape, and vertical and spanwise extent to prior results [23], but a quantitative examination of velocity contours again shows that the results are not yet fully resolved at the highest grid resolutions used. The region where the solution does not converge only extends out to around $Z/L = 0.03$ or about 2.6 cm. from the plane of symmetry. Outboard of this region, convergence is achieved with all five grids converging with high accuracy. In addition, no additional vortices or boundary-layer modulations are observed in the outboard region, as was found in the high Reynolds number ascent case.

Considering now the results from the wind-tunnel case at a unit Reynolds number of 2.9 million, all three grid resolutions simulated (50, 100, and 200 million cells) showed good agreement, albeit with slight variance found near the centerline. Comparisons of an azimuthal distribution of Stanton number with that from experiments [23] and from DNS results [23] are good, and all solutions lie within the scatter of experimental results. In addition, velocity contours at the highest resolution showed very good agreement when compared with DNS. A comparison of Stanton numbers on the surface of the BOLT scale model also revealed good agreement with DNS [23]. Features were found to be very similar in structure between the DPLR and DNS

solutions, with only minor differences in the Stanton numbers found in some locations.

Overall, BOLT flowfields computed with DPLR were found to be sufficiently converged in regions outboard of the centerline region, and it is here that wind-tunnel-scale tests have observed regions of possible transitional behavior [21]-[22]. The use of a stability analysis using LPSE has been shown to be appropriate in this region when compared against DNS [17], and this technique is applied in the present work. Note that work is ongoing to further converge the centerline region of BOLT in order to study the stability of this region using more advanced stability techniques such as the spatial biGlobal method.

3.8.2 Off-Nominal Stability Analysis at Flight Conditions

Summarizing now the off-nominal stability analysis under flight operating conditions, several interesting trends were noted for the various cases. The BOLT geometry is symmetric so that positive pitch and yaw results relate directly to negative results. Also, the present work focuses on an LPSE analysis of the flowfield outboard of the centerline

As the geometry is pitched up beyond nominal, on the windward side, wedges of moderate 2nd mode amplification tended to become more organized and extended toward the aft end of the geometry. In addition, larger pitch angles resulted in generally higher frequencies across the geometry on the windward side. On the other (leeward) side of the positive pitch cases, the 2nd mode growth was far more localized, where small regions of the geometry experienced very large growth. Furthermore, these regions were not found near the back of the geometry but instead closer to the swept leading edge. In some cases, the growth was large enough such that it might cause the flowfield to transition locally.

Considering now the stationary crossflow results for the positive pitch cases, a number of similarities were found between all of the cases. In general, on the windward side, stationary crossflow was found to be amplified in a large wedge extending towards the aft end of the geometry and growth intensifies as the geometry is pitched up. The level of growth as indicated by N-factor results might be large enough to cause the flow to transition. In addition, the locally most amplified stationary crossflow instabilities were found to have small wavelengths closer towards the leading edge with larger wavelength disturbances being predicted more locally amplified in moving toward the back of the geometry. At the aft end of the geometry, these wavelengths tend to be larger towards the center of the geometry and smaller in the outboard region.

A number of trends were also found for the 2nd mode at different yaw angles. Larger yaw resulted in larger growth (although moderate) on the leeward side in local areas that extend to the aft end of the geometry with the opposite on the windward side. Similarly, larger yaw angles resulted in higher frequencies in general for the locally most amplified 2nd modes on the leeward side but lower frequencies on the windward side. In all cases analyzed, relatively high frequencies were found next to the swept leading edge with lower frequencies found downstream. Furthermore, localized regions of very large 2nd mode growth were found near the swept leading edge on the

windward side with increasing yaw, and this growth might be large enough to locally cause transition.

Looking at the final set of results for stationary crossflow at different yaw angles revealed that geometry yaw has a large influence on the trajectory of vortex paths. For the leeward side, higher yaw results in greater path deflection away from the central structure and the opposite is true on the windward side, with paths being deflected further towards the center structures. In addition, a wedge of amplification is observed similar to the pitch results, increasing in extent on the leeward side with increasing yaw. Growth is found to possibly be large enough to cause transition. Like the pitch cases, larger wavelengths for the locally most amplified stationary crossflow instability were found towards the centerline of the geometry, with smaller wavelengths found closer towards the leading edge (farther outboard) at the aft end of the geometry.

ACKNOWLEDGEMENTS

The computational work was supported by AFRL/AFOSR under grant number FA9550-18-1-0010 with program officers Drs. Ivett Leyva and Sarah Popkin. The researchers are grateful to Pointwise and NASA for providing the Pointwise meshing program and the DPLR CFD code, respectively. The researchers also acknowledge the Texas Advanced Computing Center (TACC) at The University of Texas at Austin for providing high performance computing (HPC) and storage resources that have contributed to the research results reported within this paper. In addition, portions of this research were conducted with the advanced computing resources provided by Texas A&M High Performance Research Computing. The researchers also acknowledge a research fellowship from the Texas Space Grant Consortium, and would especially like to thank Mr. John Thome and Dr. Graham Candler from the University of Minnesota for providing valuable DNS data and insight for this work.

REFERENCES

1. Oliviero, N., Kocian, T., Moyes, A., and Reed, H., 2015. "EPIC: NPSE analysis of hypersonic crossflow instability on yawed straight circular cone," AIAA 2015-2772.
2. Moyes, A., Beyak, E., Kocian, T., and Reed, H., 2019. "Accurate and efficient modeling of boundary-layer instabilities," AIAA 2019-1907.
3. Malik, M., 1990. "Numerical methods for hypersonic boundary layer stability," Journal of Computational Physics, vol. 86, pp. 376–413.
4. Oliviero, N., 2015. "EPIC: A new and advanced nonlinear parabolized stability equation solver," Masters Thesis, Texas A&M University, Advisor: H. Reed.
5. Chang, C., 2004. "LASTRAC.3d: Transition prediction in 3D boundary layers," AIAA 2004-2542.
6. Kocian, T., 2018. "Computational hypersonic boundary-layer stability and the validation and verification of EPIC," Ph.D. Dissertation, Texas A&M University, Advisor: H. Reed.
7. Kocian, T., Moyes, A., Reed, H., Craig, S., Saric, W., Schneider, S., and Edelman, J., 2019. "Hypersonic crossflow instability," Journal of Spacecraft and Rockets, vol. 56, no. 2.
8. Whitham, G., 1960. "A note on group velocity," Journal of Fluid Mechanics, vol. 9, pp. 347–352.

9. Mack, L., 1984. "Boundary-layer linear stability theory," AGARD Report 709: Special Course on Stability and Transition of Laminar Flow, pp. 3.1–3.81.
10. Wheaton, B., Berridge, D., Wolf, T., Araya, D., Stevens, R., McGrath, B., Kemp, B., and Adamczak, D., 2021. "Final design of the boundary layer transition (BOLT) flight experiment," *Journal of Spacecraft and Rockets*, vol. 58, no. 1, pp. 6-17.
11. Wright M., Candler G., and Bose D., 1998. "Data-parallel line relaxation method for the Navier-Stokes equations," *AIAA Journal*, vol. 36, no. 9, pp. 1603-1609.
12. Reed, H., Perez, E., Kuehl, J., Kocian, T., and Oliviero, N., 2015. "Verification and validation issues in hypersonic stability and transition prediction," *Journal of Spacecraft and Rockets*, vol. 52, no. 1.
13. Wheaton, B., Berridge, D., Wolf, T., Stevens, R., and McGrath, B., 2018. "Boundary layer transition (BOLT) flight experiment overview," AIAA 2018-2892.
14. Dinzl, D., and Candler, G., 2017. "Direct simulation of hypersonic crossflow instability on an elliptic cone," *AIAA Journal*, vol. 55, no. 6.
15. Zurigat, Y., Nayfeh, A., and Masad, J., 1992. "Effect of pressure gradient on the stability of compressible boundary layers," *AIAA Journal*, vol. 30, no. 9.
16. Kuehl, J., 2018. "Thermoacoustic interpretation of second-mode instability," *AIAA Journal*, vol. 56, no. 9.
17. Moyes, A., and Reed, H., 2021. "Pre-flight boundary-layer stability analysis of BOLT geometry," Special Section on Boundary Layer Transition Flight Experiment (BOLT) - Pre-Flight Studies. Editors: Leyva, Cummings, *Journal of Spacecraft and Rockets*, vol. 58, no. 1, pp. 78–89.
18. Thome, J., Dwivedi, A., Nichols, J., and Candler, G., 2018. "Direct numerical simulation of BOLT hypersonic flight vehicle," AIAA 2018-2894.
19. Kostak, H., Bowersox, R., McKiernan, G., Thome, J., Candler, G., and King, R., 2019. "Freestream disturbance effects on boundary layer instability and transition on the AFOSR BOLT geometry," AIAA-2019-0088.
20. Moyes, A., Kocian, T., Mullen, D., and Reed, H., 2018. "Boundary layer stability analysis of HIFiRE-5b flight geometry," Special Section on HIFiRE-5b. Editor: Kimmel, *Journal of Spacecraft and Rockets*, vol. 55, no. 6, pp. 1341-55.
21. McKiernan, G., Chynoweth, B., Schneider, S., Berridge, D., and Wheaton, B., 2021. "Boundary layer transition preflight experiments in a Mach-6 quiet tunnel," *Journal of Spacecraft and Rockets*, vol. 58, no. 1, pp. 54-66.
22. Kostak, H., and Bowersox, R., 2021. "Preflight ground test analyses of the boundary layer transition (BOLT) flight geometry. *Journal of Spacecraft and Rockets*, vol. 58, no. 1, pp. 67-77.
23. Li, F., Choudhari, M.M., and Paredes, P., 2020. "Streak instability analysis on BOLT configuration," AIAA-2020-3028.
24. Mullen, D., and Reed, H., 2021. "Computational modeling and stability analysis of BOLT hypersonic geometry including off-nominal conditions," *Theoretical and Computational Fluid Dynamics*, August.
25. Knutson, A., Thome, J., and Candler, G., 2021. "Numerical simulation of instabilities in the boundary-layer transition experiment flowfield," *Journal of Spacecraft and Rockets*, vol. 58, no. 1, pp. 90-99.

E. TECHNICAL UPDATES: EXPERIMENTAL RESULTS (BOWERSOX)

1. INTRODUCTION

1.1 Research Objectives

The TAMU BOLT project objective is to perform a scientific examination of transition mechanisms on a geometry that includes concave curvature and highly swept leading edges.

1.2 Research Approach

The approach taken was to identify transition instabilities and mechanisms both on and off the surface of the BOLT geometry at hypersonic speeds through ground test and simulation. On-surface measurements of the geometry are compared between Texas A&M University, Purdue University, NASA Langley Research Center, and CUBRC hypersonic wind tunnel facilities. The freestream environment within each facility ranges from conventional to quiet (flight-like) to high-enthalpy flow. Measurements on the surface include surface pressure fluctuations measured by PCBs and Kulites as well as the surface heat flux viewed by infrared thermography and TSP and measured directly with Schmidt Boelter gauges. The heat flux results from the quiet wind tunnel facilities were compared with computations performed by Texas A&M University and the University of Minnesota.

1.3 FY2020 Focus

The focus this year has been threefold. First, the team continued to analyze the ground test data from the BOLT experimental campaign [1,2,3]. Second, the team completed the off-body results and subsequent analyses as described in Section 3.4; also see [4]. Third, the team focused on the development of the BOLT II (companion project, FA9550-19-1-0154) flight test article design with CUBRC, AFRL, AFOSR, NASA and NSROC.

1.4 FY2021 Focus

The focus this year has shifted to the ground tests in support of BOLT II (companion project, FA9550-19-1-0154). The flight test article design with CUBRC, AFRL, AFOSR, NASA and NSROC was completed under the companion project support. However, ground tests in the TAMU Mach 6 Quiet Tunnel on two subscale models (based on a 1.50-m and 1.35-m flight geometry) were supported in part by this project. Those results are provided in the reviews for the companion project.

2. EXPERIMENTAL METHODS

2.1 Wind Tunnel Models

The Boundary Layer Transition (BOLT) flight test geometry was designed by the Johns Hopkins University Applied Physics Laboratory (JHUAPL) [5]. Experimental wind tunnel testing of two 33% scale models, a 31% scale model, and a full-scale model will be discussed. JHUAPL designed and fabricated a 33% scale model that was tested at Texas A&M University, Purdue University, and NASA Langley Research Center. This 33% scale model will be referenced as the *JHUAPL* model. The machined model contains a single PEEK plastic surface for IR thermography viewing with the remaining assembly made of 6061 aluminum. The assembly is shown in Figure 2-1 with the dimensions included. The geometry features concave surfaces with highly swept leading edges.

Thermal conductivity, specific heat capacity, and density of the PEEK material are stated in Table 2-1. The values are based on the manufacturer's quote of the material and were provided to TAMU from JHUAPL. The properties are necessary for heat flux calculations from IR thermography for all facilities that are discussed in Chapter 3.

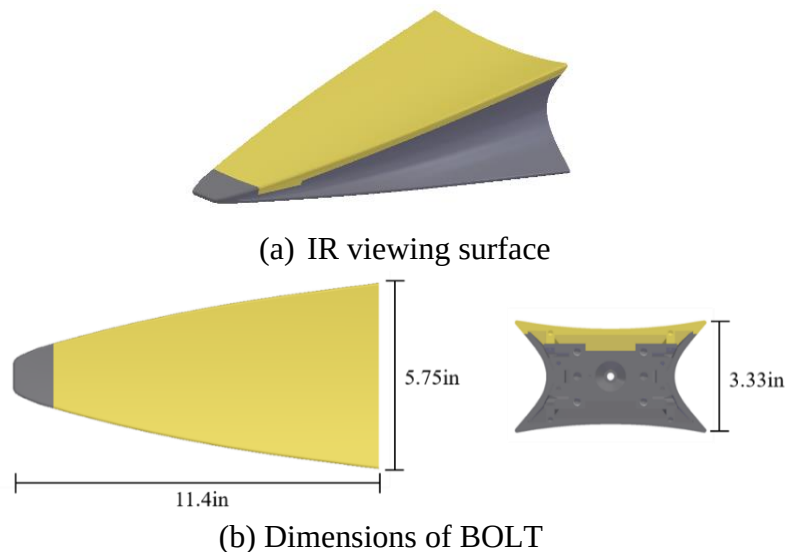


Figure 2-1. 33% scale BOLT geometry and assembly in SolidWorks (yellow - PEEK plastic, gray - 6061 aluminum); (a) is the IR viewing surface and (b) are the dimensions.

Table 2-1. PEEK property materials used for analysis for all IR results in this paper

Thermal Conductivity $\kappa \left(\frac{W}{m^2 \cdot K} \right)$	Specific Heat Capacity $c_p \left(\frac{J}{kg \cdot K} \right)$	Density $\rho \left(\frac{kg}{m^3} \right)$
0.290	1026	1300

An exploded view of the model, with the PEEK surface replaced with an experimental aluminum surface for roughness experiments [6], is shown in Figure 2-2. The assembly allows for easy access and placement of surface pressure transducers and heat flux gauges from below each surface.

JHUAPL provided TAMU with spare machined parts and CAD drawings to machine a duplicate of the 33% scale geometry. This geometry will be referenced as the *TAMU* model with the data. The manufacturing and design are similar to Figure 2-2, but the roughness cut-out located on the top surface is not present on the TAMU model. Off-body measurements were made on the TAMU model only due to a pure aluminum body with no surface pressure transducer ports. A 31% scale model was tested in the TAMU facilities at various Mach numbers that the vehicle will see in flight. The model was 3-D printed out of polycarbonate material using a Stratasys FDM 400mc printer with a layer thickness of 0.010 in. Images of all three subscale wind tunnel models are shown in Figure 2-3.

A flight-scale model of BOLT was tested at the Calspan-University of Buffalo Research Center (CUBRC). The length of the solid model is 0.865 m from the leading edge to the back of the geometry. The majority of the testing campaign at CUBRC consisted of roughness experiments, but those data will not be featured here. The top surface and gutter of the aluminum body was painted with temperature sensitive paint to view the surface heating on the model. An image of the full-scale model with Mike Holden as a reference for the size is shown in Figure 2-4.

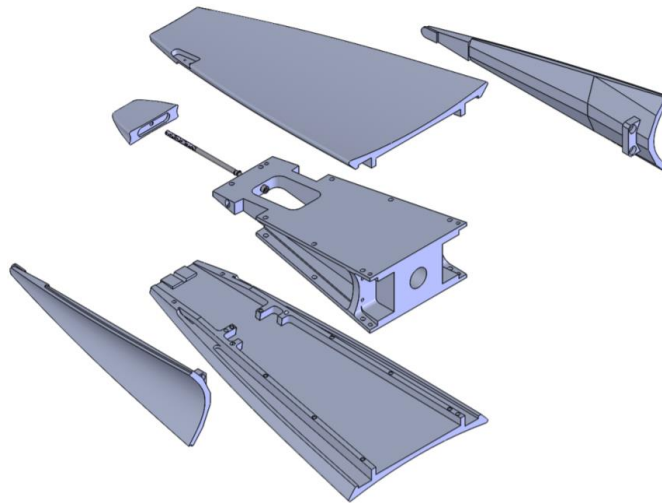


Figure 2-2. Exploded view of 33% scale BOLT model [6]

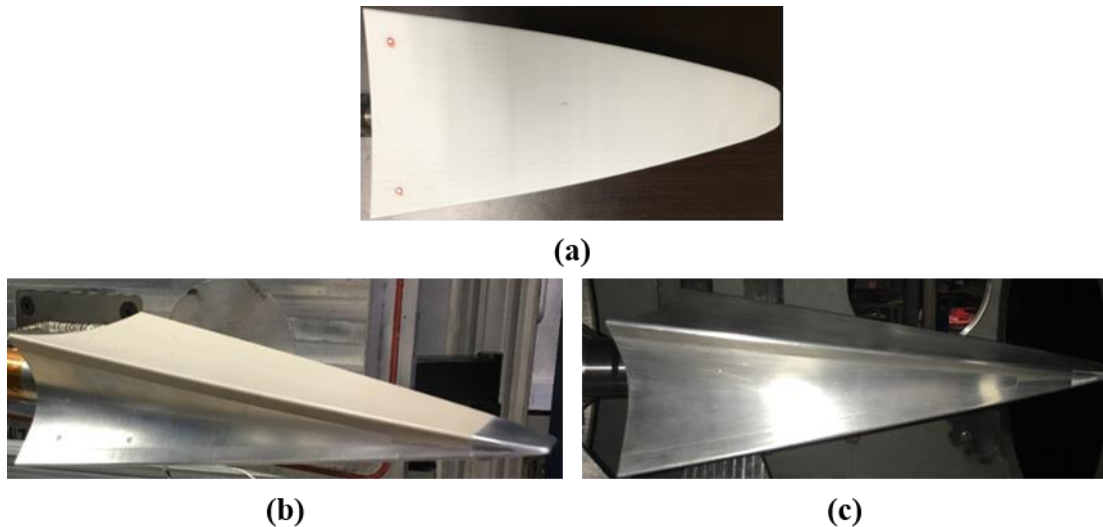


Figure 2-3. 31% and 33% scale BOLT wind tunnel models: (a) 31% scale, (b) JHUAP, (c) TAMU

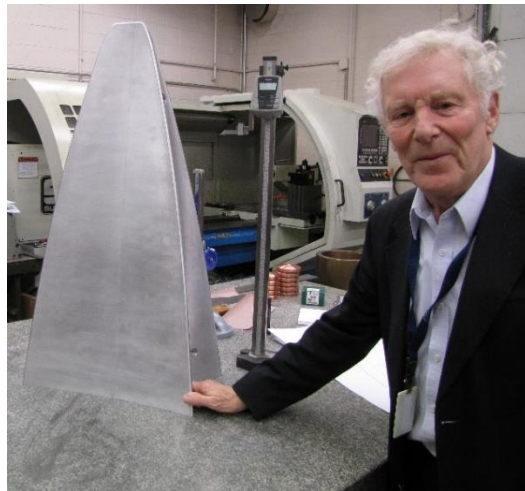


Figure 2-4. Flight-scale model of BOLT tested at CUBRC

2.2 Facilities

2.2.1 Texas A&M University

Located at Texas A&M University is the National Aerothermochemistry and Hypersonics Laboratory. Two hypersonic blow-down wind tunnel facilities located within the lab and used extensively for this research are the NASA Langley Mach 6 Quiet Tunnel (M6QT) and the Actively Controlled Expansion (ACE) tunnel. Both wind tunnel facilities are supplied by the same infrastructure, so no two tunnels can run simultaneously. Two CompAir Reavell 5442 compressors provide high-pressure air stored in a 2500 pisa capacity tank. The compressed air is filtered of sub-micron contaminants and oil from the compressors and dried by desiccant driers to 233 K. A 2-inch pipe supplies air from the compressed tank to the laboratory with a 4-inch pipe supplying air to the ejector system. Once the air from the 2-inch line reaches the laboratory, it is heated by a 0.5 MW Chromalox heater, sent through a one-micron particle filter, and blown into the tunnel. The

tunnel is preheated to a stagnation temperature of 430 K with low-speed (around 35 psi – 55 psi depending on the facility) air to prevent liquefaction within the tunnel. The JHUAPL and 31% 3-D printed model were inserted into the tunnel after preheat, whereas the TAMU model was present. A two-stage Venturi air-ejector system, located outside the building, pulls the vacuum for both facilities which uses the majority of the air supplied by the compressed tank. The Fox brand ejectors can supply a vacuum of 530 kPa with 25 kg/s of compressed air at 1 MPa. A typical run for the present experiments lasts approximately 40 seconds. More information on tunnel infrastructure is well documented in dissertations from previous colleagues [7].

2.2.2 The NASA Langley Mach 6 Quiet Tunnel (M6QT)

The M6QT, originally located at NASA Langley Research Center, is currently located at the National Aerothermochemistry and Hypersonics Laboratory (NAL) at Texas A&M University. The previous design of the tunnel was to maximize the existence of a laminar boundary layer off the nozzle walls, thereby producing stability and transition measurements in an environment closely resembling that of flight. Upon reestablishing the M6QT at Texas A&M in 2005, the performance within the tunnel was brought back to its quiet conditions [8] consistent with its past environment at NASA [9]. Located within the first half of the settling chamber are a series of meshes and screens to help dampen and reduce the noise of the incoming flow into the nozzle. At the nozzle throat is a bleed valve system that when engaged, produces a new laminar boundary layer on the nozzle wall, therefore minimizing disturbances within the testing environment. The slow-expanding, axisymmetric nozzle is highly polished with a length of 1 m, a throat diameter of 1 in, and an exit diameter of 7.5 in. The nozzle was formed by electroforming nickel onto a stainless-steel mandrel that was later plated with a nickel-phosphorus alloy. The contour was designed to minimize nozzle wall curvature and delay the growth of Göertler vortices off the wall [8, 10]. The infrastructure of the facility is shown in Figure 2-5 [11]. For the present experiments, the tunnel operated in the range of $Re/m = 7 \times 10^6 - 11 \times 10^6$.

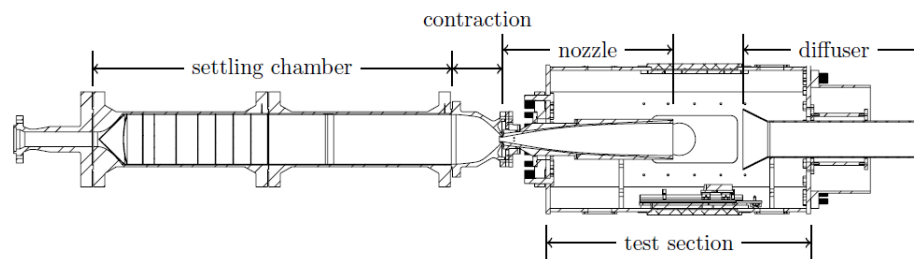


Figure 2-5. Texas A&M University Mach 6 Quiet Tunnel schematic [11]

Creating a ‘quiet’ environment is critical and challenging for wind tunnel design. A “double cone” diagram of the quiet test core environment is given in Figure 2-6, where the upstream cone is bounded by the uniform flow at the exit Mach number and the downstream cone is bounded by the pressure disturbances radiating off the turbulent nozzle wall at the Mach angle [8]. Having as much of the model in the quiet test core creates a low-disturbance flow over the

model. With a change in Reynolds number, the quiet core shifts upstream or downstream within the nozzle. A representation of this effect from Pitot pressure fluctuations measured with a Kulite dynamic pressure sensor in the facility on a flared cone [12] is shown in Figure 2-7. The color bar represents the freestream noise measured within the facility, where the dark blue region is the field of least disturbances denoted as the ‘quiet core’. A more detailed discussion of the existence and concept of the quiet core is given by Hofferth [8, 12, 13].

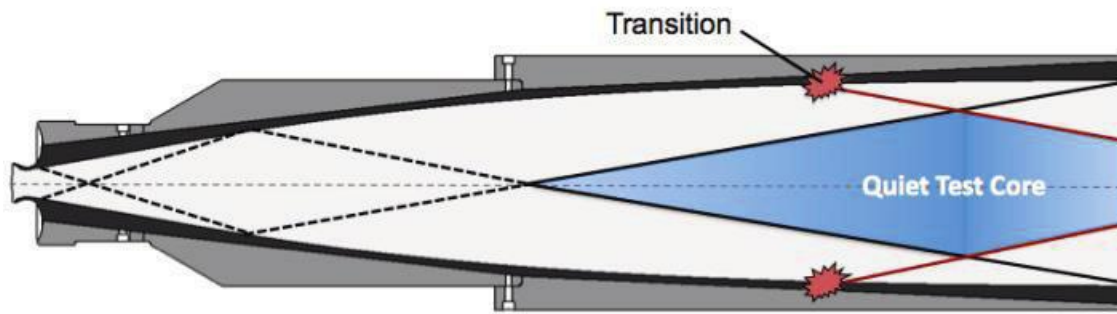


Figure 2-6. Mach 6 Quiet Tunnel quiet test core located within the nozzle of the facility [8]

The BOLT model was pushed as far upstream into the quiet core as capable in Figure 2-8. The minimum amount able to protrude past the nozzle exit while avoiding the shock off the model leading edge was 2.54 cm. This placement was kept for each run of both 33% scale models for direct comparison. Optical access was accessible by viewing the PEEK surface with the IR camera positioned at the top of the M6QT. Upon investigating the IR imagery results, elevated heating is seen on the outer portion of the JHUAPL model from noise interference. This effect will be seen in the results in Chapter 3.

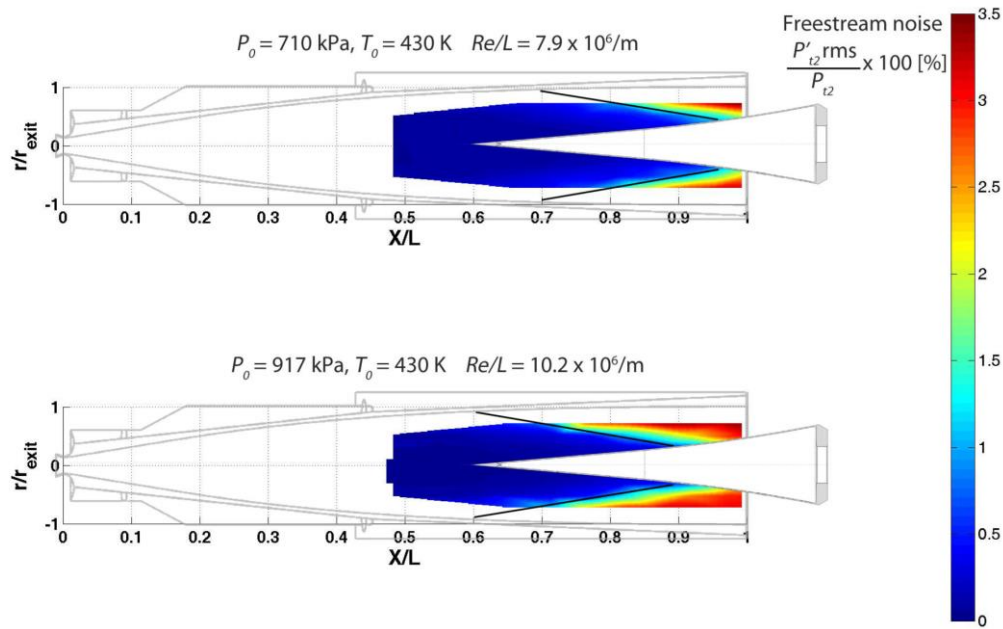


Figure 2-7. Contours of normalized freestream RMS Pitot pressure around a flared cone. Freestream conditions are $Re/m = 7.9 \times 10^6$ (top) and $Re/m = 10.2 \times 10^6$ (bottom) [12]

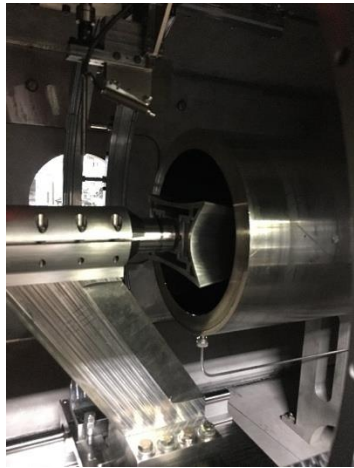


Figure 2-8. Placement of the TAMU BOLT model within the M6QT test section

2.2.3 The Actively Controlled Expansion (ACE) Tunnel

ACE is a conventional, variable Mach number wind tunnel facility that contains a higher percentage of freestream disturbances within the flow. The Mach number within the facility is varied by manually adjusting the height of the nozzle throat. For the current study, three Mach numbers were tested: 5.4, 6, and 7.4. The settling chamber contains two aerogrids and three mesh screens to help make the incoming flow more uniform. The test section is 0.23 m x 0.36 m. A diagram of the ACE facility is shown in Figure 2-9. More detailed schematics of the ACE facility as well as the new diffuser design and installation are available in various sources [7, 14, 15, 16].

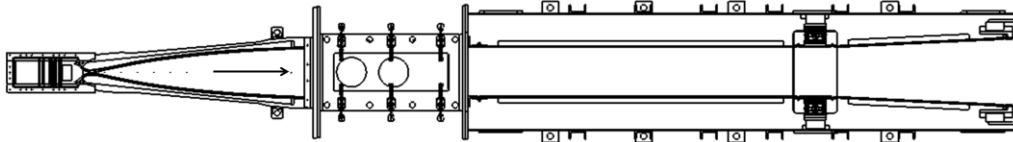


Figure 2-9. Diagram of the ACE wind tunnel facility at TAMU

The freestream environment within ACE has been well-documented over the years [16, 17, 18]. At $Re/m = 3 \times 10^6$, the nozzle walls of ACE transition and a larger amount of freestream disturbances are present in the flow due to noise radiating off the nozzle sidewalls. An excellent graph that portrays the difference in the freestream disturbance environment relative to the Mach number in ACE, the M6QT, and the Purdue BAM6QT is shown in Figure 2-10 [18].

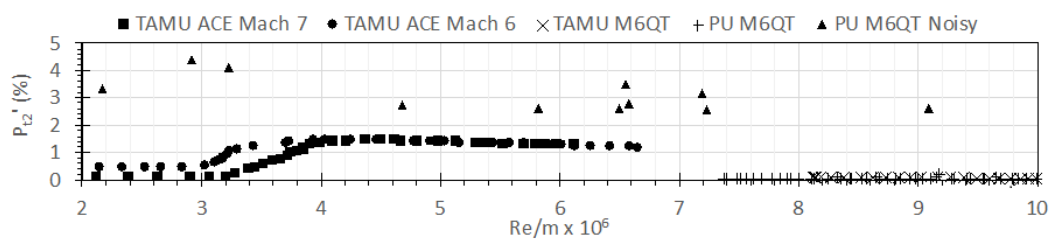


Figure 2-10. Freestream disturbance environment comparison between M6QT, BAM6QT, and ACE [18]

The y-axis represents the freestream Pitot pressure fluctuations within the designated facilities and the x-axis is in Reynolds number per meter (Re/m). As the Mach number increases within ACE, the acoustic freestream disturbance levels are less effective than at the lower Mach numbers. However, at $Re/m = 3 \times 10^6$, the freestream disturbances begin to dominate within the flow, showing a sudden increase in the curves. Comparing these results to the M6QT and Boeing/AFOSR Mach 6 Quiet Tunnel (BAM6QT), very low disturbances are seen in quiet flow. When the BAM6QT is run with bleed valves closed (noisy), higher freestream fluctuations are present than those in ACE. A side view of the JHUAPL BOLT model within ACE is shown in Figure 2-11 where flow is from left to right. The IR camera is also seen mounted above the tunnel on a cage constructed of XT-95 optical railing. The position of the camera is ideal for looking straight through a Zinc selenide ($ZnSe$) window onto the PEEK surface of the model.



Figure 2-11. BOLT located in the Actively Controlled Expansion tunnel

2.2.4 Purdue University

Located at Purdue University, the BAM6QT is a low-disturbance facility featuring a Ludwig tube design producing hypersonic flow. The BAM6QT contains a similar bleed valve system as Texas A&M to produce quiet or conventional flow within the facility. The highly-polished nozzle is elongated to minimize the Göertler instability. The wind tunnel design is capable of producing freestream noise levels as low as 0.01% when ran as quiet but can produce noise levels near 3%, previously seen in Figure 2-10, when operated as a conventional hypersonic facility [19]. A schematic of the facility is given in Figure 2-12 where the design incorporates a long driver tube followed by a converging-diverging nozzle ending with a large dump tank. The facility utilizes two aluminum diaphragms with pressure located between them, which is half that of the pressurized region within the tunnel [20]. When the diaphragms burst, an expansion wave travels upstream through the nozzle and a shock wave propagates downstream. The expansion wave continues to reflect back and forth within the tunnel, causing a quasi-drop in the total pressure [20] during a wind tunnel run. The run lasts approximately 5 seconds. Wind tunnel models are placed at the end of the diverging section of the nozzle where the change in the tunnel diameter is gradual. Models within the facility can be mounted at various angles of attack (AoA) and yaw, but the present study will focus on zero AoA and yaw experiments with the JHUAPL BOLT model. An image of the JHUAPL model right before closing of the BAM6QT is shown in Figure 2-13. The model is mounted such that the PEEK surface is towards the window for IR thermography. The other experimental surface (not visible) is painted with TSP.

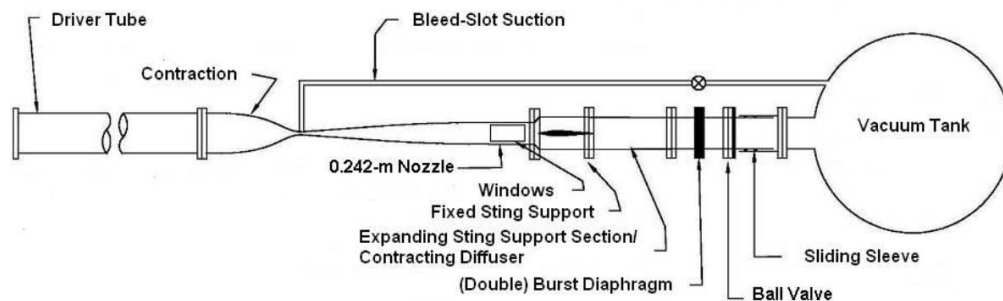


Figure 2-12. Purdue University's Boeing/AFOSR Mach 6 Quiet Tunnel schematic [20]

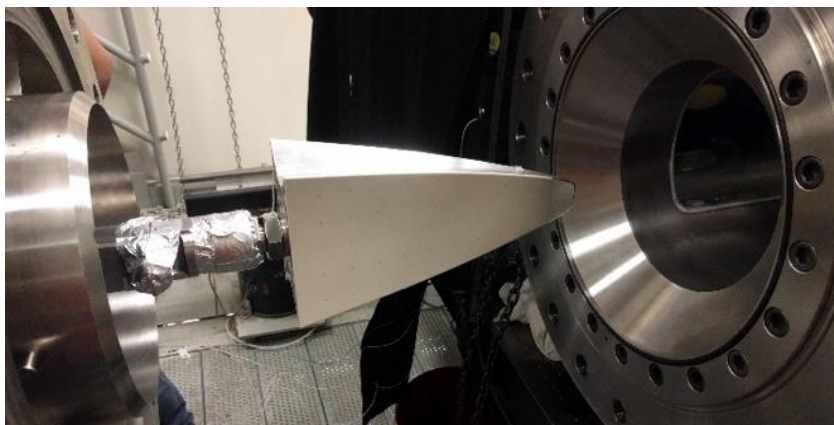


Figure 2-13. BOLT outside of the BAM6QT before model placement in the facility; PEEK side shown

2.2.5 NASA Langley Research Center

The 20-inch Mach 6 Air Tunnel is a conventional hypersonic blow-down facility located at NASA Langley Research Center. The tunnel first became operational in the year 1958 with a basis for aerodynamic and aerothermodynamic tests. Air is supplied by a 4250 psi bottle field and a 600 psi bottle field that is preheated [21] and filtered. A layout of the facility is shown in Figure 2-14 with a more concise and detailed breakdown of the infrastructure given in [21].

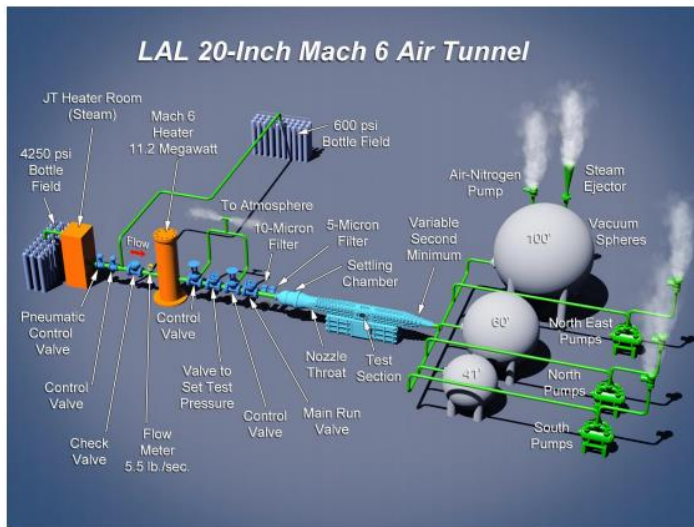


Figure 2-14. NASA Langley Research Center 20-Inch Mach 6 Air Tunnel schematic [21]

The settling chamber is equipped with screens and a perforated conical baffle at the entrance. The 7.45 ft long nozzle is two-dimensional, where the top and bottom walls are contoured and the sides are parallel. The test section is 20.5 in x 20 in. The facility incorporates a bottom-mounted injection system that transfers the wind tunnel model from the sheltered model box to the tunnel test section centerline in less than half a second when the tunnel is on condition. For the BOLT experiments, a run time of a few seconds was only needed, but the facility can run well over 15 minutes. The Mach number is 5.8-6.1, with a unit Reynolds number (Re/m) range of 1.6×10^6 to 27×10^6 . Zero AoA and yaw are discussed for the current analysis. A photograph from a roughness experiment of the JHUAPL wind tunnel model before placement into the tunnel is shown in Figure 2-15. The tunnel has the capability of a full-field view for IR thermography through a ZnSe antireflective-coated window located at the top of the test section.

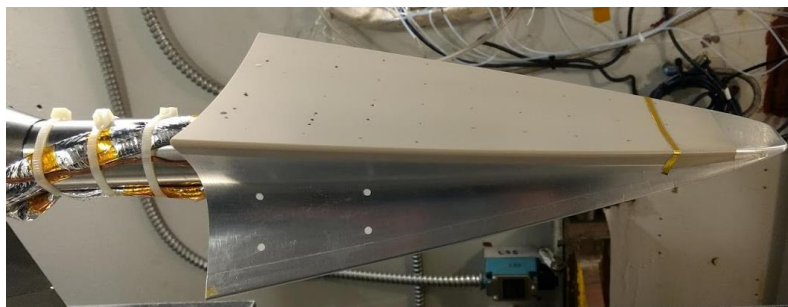


Figure 2-15. BOLT located in shelter box in the 20-Inch Mach 6 Air Tunnel for a roughness experiment.

2.2.6 Calspan-University of Buffalo Research Center (CUBRC)

The CUBRC LENS II facility is a hypervelocity reflected shock tunnel. The facility can run as either a shock tunnel or Ludwieg tube with fully duplicated flight conditions from Mach 3 to 9 and is shown on the right in Figure 2-16. The facility features long driver and driven tubes, around 60- and 100-ft in length, respectively, [22] with three different contoured nozzles to achieve the Mach number range previously mentioned. Test times range from 100-200 ms. Within the double-diaphragm unit, the test gas is pressurized which causes the diaphragms to burst. A shock wave then travels towards the driven tube, is reflected, and travels back upstream creating stagnant high-pressure and temperature gas. The test gas travels through the nozzle, into the test section, and a valve quickly closes at the nozzle throat terminating the flow [22].

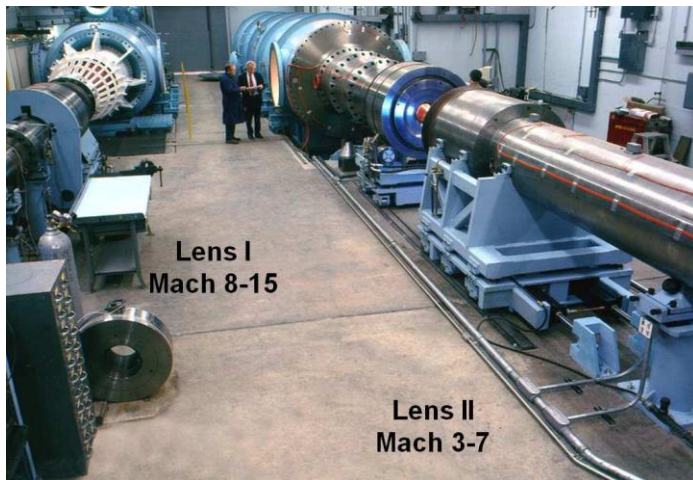


Figure 2-16. LENS I and LENS II facilities located at CUBRC [22]

For the current BOLT experiments, the facility ran as a shock tube within the Woomera trajectory Mach numbers and Reynolds numbers with a run time of 100 ms. The full-scale aluminum blockage model in the LENS II facility is shown in Figure 2-17. Due to the high-enthalpy environment within the tunnel and model material, TSP was used to view the heating on the surface. The TSP is represented as the yellow color painted on the aluminum surface.

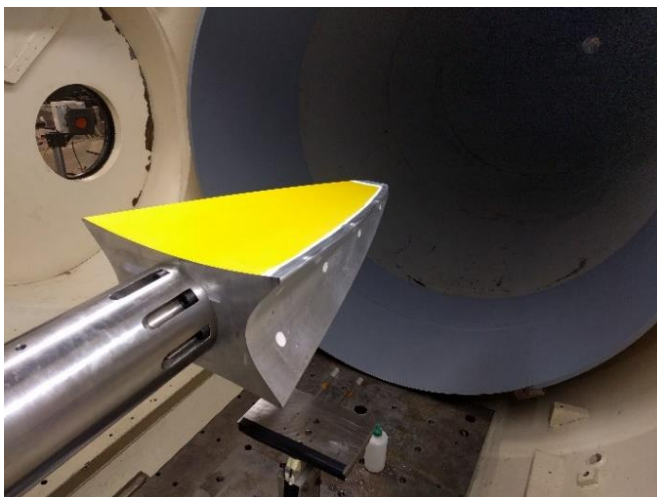


Figure 2-17. Full-scale model of BOLT in the CUBRC LENS II facility. Yellow surface represents temperature sensitive paint.

2.3 Instrumentation

2.3.1 IR Thermography

Infrared thermography was used by Texas A&M University, Purdue University, and NASA Langley to view the global surface temperature on the PEEK surface of BOLT. The sensitivity of IR thermography allowed for the identification of boundary layer flow structures (such as stationary crossflow instability vortices) and transition. Each facility used different mounting techniques, software, and cameras when viewing the surface. Table 2-2 lists each IR camera and corresponding characteristics used by each facility to produce temperature maps of the surface of BOLT.

Table 2-2. IR thermography camera characteristics for each facility

Facility	Camera	Resolution (pixels)	Wavelength sensitivity (μm)	Raw output
Texas A&M	FLIR SC8100	1,024 × 1,024	3-5	14-bit
Purdue	Xenics Onca MWIR-640	640 × 512	3.7-4.8	14-bit
Langley	FLIR SC6701	640 × 512	3-5	14-bit

Calibration of each camera was achieved separately by each facility. At Texas A&M, a thermocouple was embedded in a black aluminum calibration plate that was gradually heated and observed by the FLIR SC8100. The process and method is heavily documented in [7]. NASA Langley incorporated a bench-top calibration method using a Mikron M310 black-body source. The raw temperature IR images produced from each facility were calibrated and processed at Texas A&M for the current data analysis.

The Texas A&M NAL 1-D heat transfer data reduction code for converting the temperature maps of the IR images into heat flux was developed at Texas A&M [23] but tweaked for BOLT configurations [1]. The MATLAB code was initially inspired by the FORTRAN code QCALC [24] and configured for a more modern approach for HIFiRE studies [25, 26]. All experimental IR data were processed using this code. The code involves a forward time central difference scheme shown in Equation 2-1 for solving the heat conduction equation.

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2} \sim \frac{T_i^{n+1} - T_i^n}{\Delta t} = \alpha \frac{(T_{i+1}^n - 2T_i^n + T_{i-1}^n)}{\Delta x^2} \quad (2-1)$$

The thermal diffusivity, α in Equation 2-1, is defined in Equation 2-2:

$$\alpha = \frac{\kappa}{\rho c_p} \quad (2-2)$$

The material properties in Equation 2-2 were previously mentioned in Table 2-1 for PEEK provided by JHUAPL. The heat flux is then calculated using a second order approximation of the first spatial derivative using Equation 2-3:

$$q(n) = \frac{-k(-\frac{3}{2}T_0^n + 2T_1^n - \frac{1}{2}T_2^n)}{\Delta x} \quad (2-3)$$

Where n is the time coordinate and i is spatial. Two boundary conditions are implemented: a surface condition which is the time dependent surface temperature from the IR camera and a depth condition which is assumed constant within the defined depth of the model material. Due to the low thermal conductivity of PEEK, the 1-D analysis performed is deemed appropriate. The comparisons with CFD provides some support of this assumption. A more detailed explanation of the code and process during a tunnel run is provided by Neel [18].

Since each facility has different run conditions, i.e. settling chamber temperature, converting the heat flux data into Stanton number using Equation 2-4 were more appropriate. The subscript ∞ denotes freestream conditions, 0 is the stagnation properties, and w is the wall properties. These results will be shown in Chapter 3.

$$St = \frac{q}{\rho_{\infty} U_{\infty} c_p (T_0 - T_w)} \quad (2-4)$$

2.3.2 Temperature Sensitive Paint (TSP)

Temperature sensitive paint (TSP) is a luminescent paint mixture that enables surface temperature measurements through thermal quenching. TSPs are coatings that involve a carrier, binder, and photo luminescent probes [27]. Incoming light collides with the luminescent molecules within the paint and forces an electron into an excited state. The electron then returns to its ground state and releases a photon with decreased intensity. The mechanism is dependent on temperature, thus changes in temperature on the surface can be determined from a charged-couple device (CCD). Both Purdue and CUBRC acquired TSP images. The process of TSP is heavily documented for high speed flows in the LENS I facility at CUBRC [27]. The process is assumed the same for the LENS II facility.

At Purdue University, TSP data were acquired simultaneously with IR thermography. Shown in Figure 2-18 is a photo of the aluminum surface on the JHUAPL BOLT wind tunnel model painted with TSP. To maximize the excitation of the molecules, two arrays of light-emitting-diodes (LED) were used. An Innovative Scientific Solutions Inc. (ISSI) LMA LM4 array and an LM2xLZ-465 array were used to illuminate the model. Images of the surface were recorded on an IMPERX Bobcat IGV-B1620 14-bit CCD camera. When converting the intensity values from the CCD to temperature maps, three sets of images are recorded per run: a ‘dark’ image, an ‘off’ image’, and an ‘on’ image. The change in temperature is a function of the ratio of intensity values from these images. Liu states the empirical process for correlating these images [28]. Fourier’s law of heat conduction is simplified by assuming one-dimensional heat transfer. These data are then calibrated to a Schmidt-Boelter Gauge located on the model surface. All TSP images were imaged and processed at Purdue.

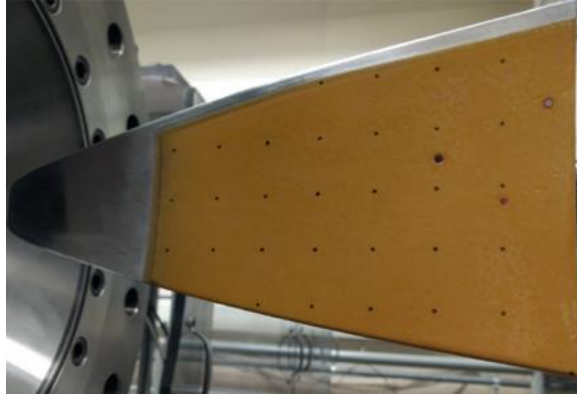


Figure 2-18. TSP painted on the aluminum surface of the JHUAPL model at Purdue

2.3.3 Surface Mounted Pressure Transducers

Each facility instrumented BOLT with surface mounted pressure transducers to help identify transition mechanisms. High frequency pressure measurements were achieved using both Kulite and PCB pressure sensors. Kulite XCE-062-15A were used in each facility and have been characterized under a supersonic turbulent boundary layer [29]. The transducer is expected to have a linear frequency response up to 30% to 40% of their resonant frequency [29], which corresponds to about 0 – 100 kHz. The Piezotronics PCB 132B38 model were used extensively in this campaign. The PCBs were powered by manufacturer in-line signal conditioners. Measurements in the hundreds of kilohertz is necessary due to the known frequency range of the second mode and potential secondary instabilities. PCBs have proven reliable for frequencies between 50 kHz and 1 MHz. A summary of the pressure fluctuation methods for each facility with the corresponding filtering of sensors is given in Table 2-3.

At Texas A&M, data were acquired by an in-house LabVIEW VI with two National Instruments USB-6366 (16-bit) DAQ systems. The PCBs are high-passed filtered by built-in electronics with a 2 dB cutoff at 11 kHz. The Kulite signal conditioning box was built in-house by Dr. Jerrod Hofferth, with a circuit design modified by the one previously developed by S.P. Schneider research group at Purdue University [13]. The box provides a DC and AC signal output for each sensor, where the DC output is gained by 100 and low-passed filtered with a first-order RC circuit at 482.5 kHz. The AC-coupled content was high-pass filtered at 842 Hz with a gain of 28.9. The AC-coupled PCB data signals were low-passed at either 500 kHz or 1 MHz. The authors found no difference in the data with either of these two filters. Kulite data signals were low-pass filtered at 200 kHz. Both PCBs and Kulites used 8-pole Bessel filters of unity gain for anti-aliasing. The filtering units were located within a 2-channel Khron-Hite FMB3002 chassis.

At Purdue University, data were acquired through a Tektronix TDS7104 oscilloscope using High-Res mode. Data were sampled at 2-5 MHz to satisfy the Nyquist criteria for resolving pressure fluctuations up to 1 MHz. NASA Langley acquired data using PXIe-6358/6124 National Instruments DAQ systems with precision filters.

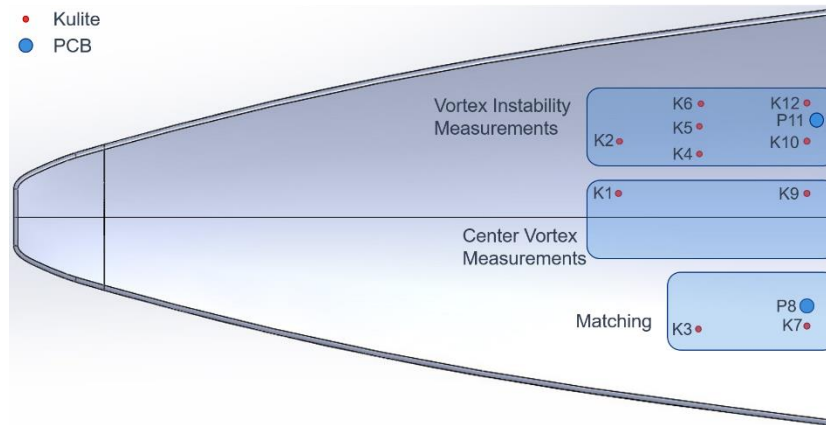
Table 2-3. Type of sensor and filtering characteristics for each wind tunnel facility

Type	TAMU	Purdue	Langley
PCB Sampling	2 MHz	2 – 5 MHz	2.5 MHz
PCB Low-pass	500 kHz or 1 MHz	None	1 MHz
PCB High-pass	11 kHz	11 kHz	11 kHz
Kulite	2 MHz	1.25 MHz	200 kHz
Sampling			
Kulite Low-pass	200 kHz	None	80 kHz
Kulite High-pass	842 Hz	840 Hz	100 Hz

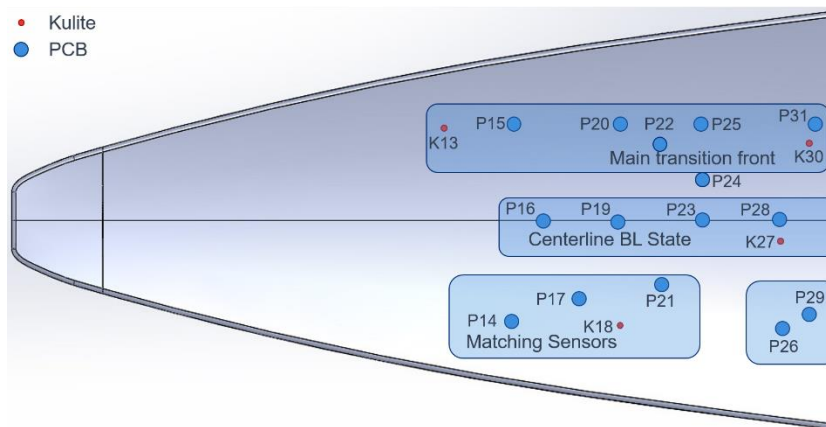
The PCB and Kulite sensors were adhered into the wind tunnel models using the PCB manufacturers preferred method of nail polish which has a higher pressure sensitivity than RTV and Neoprene [30]. NASA Langley added RTV to the back of the sensor to limit movement. The sensor top surface was mounted flush with the PEEK surface. With a limited quantity of PCB and Kulite sensors available at Texas A&M, various sensor configurations were tested. The unused ports were plugged using Teflon rod stock.

Power spectral density plots of the spectra measurements at Texas A&M and Purdue were computed using Welch's method, where a Hanning window of size 2^{10} was used with a 50% overlap. NASA Langley also used Welch's method, where the Hamming window size for the PCBs and Kulites were 2^{10} and 2^8 , respectively, both with a 50% overlap. The authors found the different window size techniques made little to no difference in the final results.

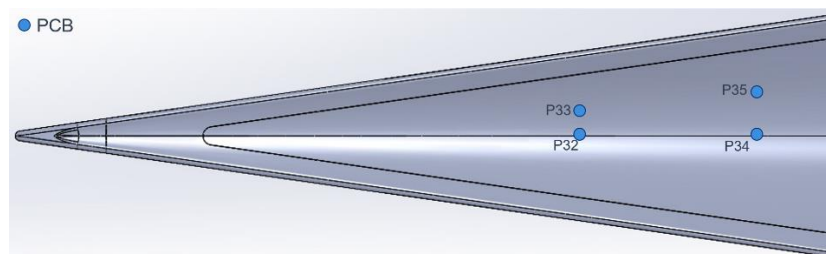
The sensor locations were strategically placed based on the structures observed in the IR and TSP data [6] and is shown in Figure 2-19. Table 2-4 lists the location of each sensor located on BOLT. A limited number of sensors were available at Texas A&M. Therefore, Appendix A lists the configuration of sensors per each wind tunnel run. NASA Langley and Purdue ran all sensors accordingly.



(a)



(b)



(c)

Figure 2-19. Sensor layout on BOLT surfaces; (a) is the PCB and Kulite sensor layout on the PEEK surface, (b) is the layout on the aluminum surface, and (c) is the layout on one of the gutter surfaces. The rectangular-shaped boxes represent measurement regions of interest.

Table 2-4. Location of sensors on 33% scale model of BOLT

Sensor	x-location from back of BOLT (inches)	z- location (inches)
K1	3.000	0.250
K2	3.000	1.000
K3	2.000	-1.500
K4	2.000	0.900
K5	2.000	1.250
K6	2.000	1.500
K7	0.500	-1.500
P8	0.375	-1.273
K9	0.500	0.250
K10	0.500	1.000
P11	0.375	1.273
K12	0.500	1.500
K13	5.500	1.200
P14	4.500	1.250
P15	4.500	1.250
P16	4.000	0.000
P17	3.500	-1.000
K18	3.000	-1.500
P19	3.000	0.000
P20	3.000	1.250
P21	2.500	-1.000
P22	2.500	1.000
P23	2.000	0.000
P24	2.000	0.500
P25	2.000	1.250
P26	1.000	-1.500
K27	1.000	-0.250
P28	1.000	0.000
P29	0.375	-1.273
K30	0.500	1.000
P31	0.375	1.250
P32	3.000	0.000
P33	3.000	0.680
P34	1.000	0.000
P35	1.000	0.880

2.3.4 Surface Mounted Heat Flux Gauges

Schmidt Boelter gauges were mounted in specific locations at NASA Langley to provide a direct measurement of heat flux for comparison to the IR results. The sensors used were Medtherm Corporation 8-2-0.25-48-20835TBS model lent to Langley from Purdue. The sensor contains an array of Type-T thermocouples (thermopile), which forms a differential thermoelectric circuit that can be used for both body and surface measurements. The diameter of the sensor is 0.125 inches, the same diameter as a PCB, and was mounted flush with the surface through the same procedure as a PCB. The output voltage was acquired directly into Langley's DAQ system. Table 2-5 lists the signal processing of the gauges. The gauges were amplified by 100 and low-passed by 30 Hz as per the instructions from Purdue. The location of the sensor used in this study is circled in yellow in Figure 2-20.

Table 2-5. Medtherm heat flux gauge processing

Medtherm 8-2-0.25-48-20835TBS Gauge		
<i>Amplified Signal</i>	<i>Low-pass (Hz)</i>	<i>High-pass (Hz)</i>
100x	30	0

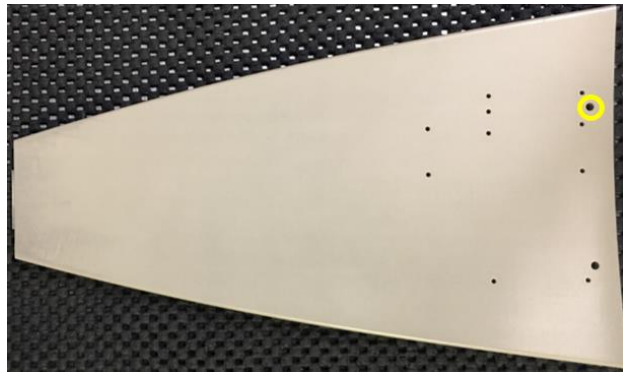


Figure 2-20. Location of the Schmidt Boelter gauge in the PEEK surface for analysis

2.3.5 Hot-wire Anemometry

TSI high temperature probes (1220-PI2.5 and 1220-PI5) with slack were used to quantify off-body instability modes for frequencies less than 100 kHz. Hot-wires respond to changes in mass flux and total temperature [7]. The sensor temperature loading factor was set to $\tau = 0.8$ to minimize total temperature sensitivity. An A.A. Lab Systems AN-1003 constant temperature anemometer was employed to power the sensor. The wire was tuned during a wind tunnel run by adjusting the damping and ferric screws located on the AN-1003 system. The response was

monitored in NI SignalExpress Project simultaneously using a PCI-5122 digitizer/oscilloscope card. The AN-1003 unit balances the bridge that contains a feedback loop for maintaining the wire's set resistance. The schematics of the sensor are shown in Figure 2-21, along with a photograph of the anemometer system. The sensors were not calibrated for the present study as the interest was frequency content. All hot-wire traverse runs were at a set Reynolds number of $9.9 \times 10^6/m$ for comparison with DNS results from the University of Minnesota. The hot-wire signal processing parameters are listed in Table 2-6. The designation of A or B in the table for probe 1220-PI2.5 is a replacement of probe A to probe B due to wire breakage. The frequency response after tuning each probe with respect to the 3-dB roll-off is also listed in Table 2-6.

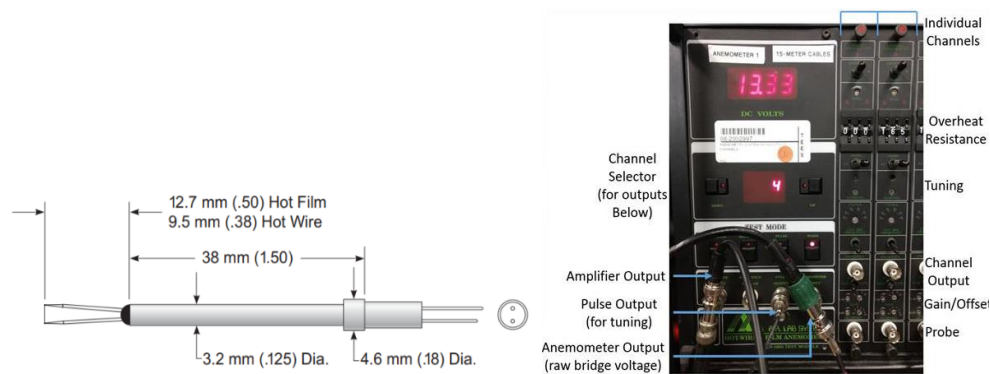


Figure 2-21. Model 1220 high-temperature straight probe (left) and A.A. Lab Systems interface (right) [7]

Table 2-6. Hot-wire parameters for processing

Name	Diameter of Sensing Area (micrometer)	Frequency Response (kHz)	Sample rate (kHz)	High-pass (kHz)	Low-pass (kHz)
1220-PI2.5 (A & B)	6.3	130	500	1	200
1220-PI5	12.7	100	500	1	200

Within the test section of the Mach 6 Quiet Tunnel, a three-axis traverse was previously designed and built by Stuart Craig for capability of studying three-dimensional phenomena [11]. The traverse moves in cylindrical coordinates: z (axial), r (radial), and θ (azimuthal). The complete design characteristics of it are described in [11]. The traverse aids in quantifying effects over the surface of models through off-body measurements. Figure 2-22 is the traverse mechanism [11]

with BOLT installed. Two-dimensional planar data (maps) were acquired at a single streamwise location at $x = 0.24$ m. Results will be discussed in Chapter 3.

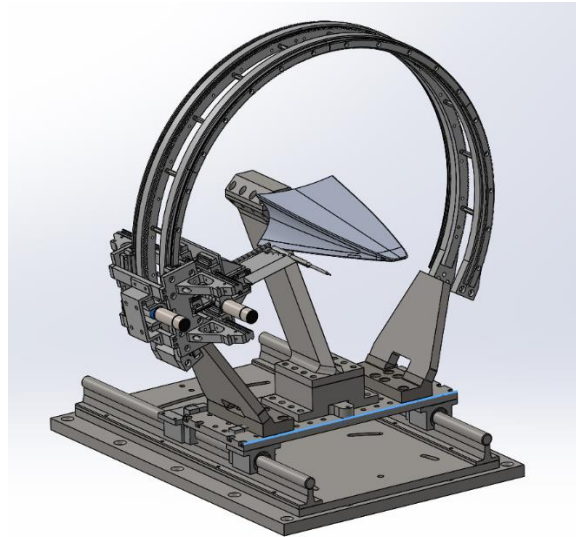


Figure 2-22. Three-dimensional traversing mechanism in the M6QT with BOLT mounted (the location of the hot-wire in the image is for visualization only).

To protect the sensor from abnormal flow in the beginning and the shock from tunnel unstart at the end, the hot-wire was hidden behind the nozzle exit during preheat, before tunnel startup, and before tunnel shutdown. Once the tunnel is on condition, the wire moves to the first measured point as per hard-coded in LabVIEW and continues with its path before stepping back behind the nozzle. The number of steps/traverses depended on the time available for a complete wind tunnel run.

2.4 Uncertainty Estimates

2.4.1 TAMU Freestream environment

Tunnel freestream conditions for both TAMU facilities are computed using the settling chamber total pressure and temperature and the nozzle wall static pressure. The total pressure in ACE is measured using an Endevco 8540-200 pressure transducer with a factory uncertainty stated of 2.4%. The uncertainty is based on temperature, hysteresis, repeatability, and non-linearity. The total pressure in the M6QT is measured with an MKS Baratron 615A capacitance manometer with an uncertainty of $\pm 0.12\%$ of its measured value. The static pressure of ACE is measured by a Baratron 631C-10 with a manufacturer uncertainty listed of 0.5%. The M6QT static pressure is measured by an MKS 902 vacuum transducer with an uncertainty of $\pm 1\%$ of the reading. The Mach number and Reynolds number are calculated from these measurement quantities. At a $Re/m = 9.9 \times 10^6$ in the quiet tunnel, the total pressure, total temperature, and Mach number standard deviation is $\pm 0.8\%$, $\pm 0.5\%$, and $\pm 0.6\%$, respectively. At a $Re/m = 5 \times 10^6$ in ACE, the total pressure,

total temperature, and Mach number standard deviation is $\pm 1.35\%$, ± 0.57 , and $\pm 0.13\%$, respectively.

2.4.2 Instrumentation

The uncertainty in the IR thermography involves the properties of the PEEK material, emissivity, camera angle, temperature calibration, and processing assumptions. PEEK has a broad range of emissivity values causing large sources of uncertainty in the analysis. The properties of PEEK used for the heat-transfer analysis were provided by JHUAPL. The initial temperature of BOLT within the M6QT is assumed to be the temperature of the model as it is assembled within the test section after preheating the tunnel to 430 K. A source of uncertainty is present for the long wind tunnel run times (up to 40 sec) in ACE and the M6QT, where the thinner regions of the model may heat up to higher temperatures. To better quantify the uncertainty of the IR code, experiments were performed at NASA Langley with a Schmidt Boelter heat flux gauge. Comparing the heat transfer results from IR at NASA Langley to the Schmidt Boelter gauges at similar locations agreed to within 6%. These results will be shown in Chapter 3.

Kulite XCE-062-15A transducers have a factory stated uncertainty of $\pm 0.1\%$ of its 103.5 kPa range. The Kulites were calibrated at room temperature under vacuum and experience a ΔT of 50 K during a wind tunnel run. This results in a $\pm 1\%$ thermal zero and sensitivity shift. PCB 132B38 transducers were provided with a factory rated calibration and sensitivity uncertainty of $\pm 30\%$ up to its 345 kPa range. These time of arrival sensors utilized a single 50 psia test point for calibration. Various influences such as the sensing area, amplitude linearity, and structural strain cause uncertainty in the PCB measurements. Work is underway for quantifying these uncertainties in the PCB measurements by Piezotronics [30]. The PSD plots for both sensors were generated from 390 averages, which corresponds to an uncertainty of 5.1% for a PSD value.

Data were acquired at 500 kHz for 100 ms. The PSD plots were generated by 97 averages resulting in an uncertainty of 10.2% for each value. The location of the hot-wire within the M6QT test section is dependent on the three-axis traverse hardware explained more in-depth in [31]. The r -axis utilizes a Faulhaber 2232S024BX4 AES-4096 brushless DC motor with an absolute encoder. The z - and θ -axes contain a Faulhaber 2250S024BX4 AES-4096 motor with absolute encoders. The traverse was designed such that the r - and z - axes move with a resolution of 244 nm per encoder line with an accuracy of ± 1220 nm. [31]. The θ -axis moves with an accuracy of $3^\circ \times 10^{-5}$ per encoder line with an accuracy of $\pm 3^\circ \times 10^{-4}$ [31].

The model support apparatus used for BOLT in the M6QT was previously aligned to zero degree incidence in pitch and yaw using a 7° cone described by Hofferth et al [12]. The angle was adjusted such that the second mode frequency was consistent around the azimuth of the cone. The uncertainty in the angle of the model with respect to the support structure in the M6QT is $\pm 0.1^\circ$. In ACE, the model is mounted on the door parallel to the flow. The uncertainty in the angle of the sting mount located on the door with respect to the bottom of the test section is $\pm 0.2^\circ$.

Uncertainty in the computations of heat flux from IR thermography involve the properties of the material, camera angle, and calibration of the camera. Of these parameters, the material

properties are the main sources of uncertainty. The properties used in this study were provided by JHUAPL. The uncertainties are based on nominal manufacturer documentation. Thermal conductivity and density were seen to vary between $0.25\text{--}0.29 \frac{W}{m \cdot K}$ and $1300\text{--}1310 \frac{kg}{m^3}$, respectively, at room temperature. Specific heat variances of $1026\text{--}1160 \frac{J}{kg \cdot K}$ were observed within the temperature range the PEEK surface experienced during testing. Comparisons have been made between the IR thermography and Schmidt Boelter heat flux gauges tested at NASA Langley and results were found to agree within 6%.

The hot-wire results were acquired at 500 kHz for 100 ms. The PSD plots were generated by 97 averages resulting in an uncertainty of 10.1% for each value. The spectral resolution for each point plotted in the PSD was 0.5 kHz. The location of the wire itself within the M6QT test section is dependent on the three-axis traverse hardware. The r-axis utilizes a Faulhaber 2232S024BX4 AES-4096 brushless DC motor with absolute encoder. The z- and θ -axes contain a Faulhaber 2250S024BX4 AES-4096 motor with absolute encoders. The traverse was designed such that the r- and z-axes move with a resolution of 244 nm per encoder line with an accuracy of ± 1220 nm [7]. The θ -axis moves with an accuracy of 3×10^{-5} degrees per encoder line with an accuracy of $\pm 3 \times 10^{-4}$ degrees [7]. The relative position of the hot-wire with respect to the model surface was measured before each tunnel run using a ruler with an uncertainty of ± 0.4 mm.

3. RESULTS

3.1 Flow Visualization and Heat Transfer

Features on the surface of BOLT through wind tunnel testing were visualized through infrared thermography and temperature sensitive paint. A quiet DNS result of BOLT was computed by the University of Minnesota for Texas A&M M6QT conditions. Comparisons between the quiet DNS results and the quiet wind tunnel results will be made in this section as well as comparisons with the conventional wind tunnel results.

3.1.1 Quiet Tunnel Results with Comparison to DNS

IR thermography results from the Purdue BAM6QT for the 33% scale wind tunnel model are shown in Figure 3-1 at $Re/m = 12.4 \times 10^6$, where flow is from left to right. To break down the configuration, Figure 3-1.a is the measured temperature difference from the baseline temperature, meaning a background image before the tunnel starts is subtracted from the actual tunnel run image. The computed heat flux using Texas A&M's 1-D heat transfer code is presented in Figure 3-1.b with Stanton number (St) following in Figure 3-1.c. The color bar adds to clarify the certain process: rainbow for change in temperature, fire for heat flux, and grayscale for Stanton number. IR results in the M6QT are shown in Figure 3-2 at $Re/m = 10.2 \times 10^6$. Additional heating can be seen on the outer 'winged' region of the model, but the inner 60% of the model verifies the same streak-like structure on the model at Purdue. Extra streaks can be seen on the surface of BOLT at

Purdue, but this may be a result from the higher Reynolds number and/or slight model yaw. The outboard heating on the model in the TAMU facility remains a discussion point, where transition effects off the nozzle walls previously seen in Figure 2-6 may have been a contributor. The high heat flux values are a result of the outboard heating on the model. This effect is to not be confused as transition to turbulence.

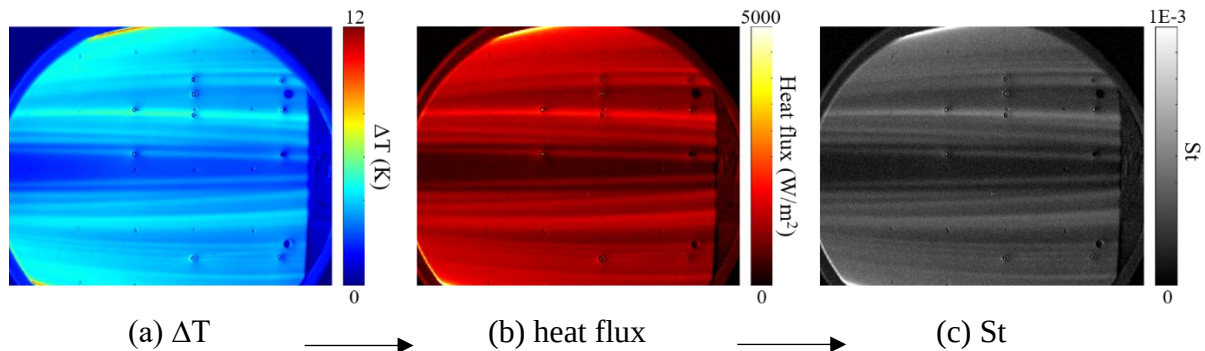


Figure 3-1. Purdue BAM6QT results at $Re/m = 12.4 \times 10^6$ with flow from left to right. The back 6 inches of the model is shown with IR. (a) is the change in temperature, (b) is the calculated heat flux using Texas A&M's 1-D heat transfer code, and (c) is the conversion to Stanton number

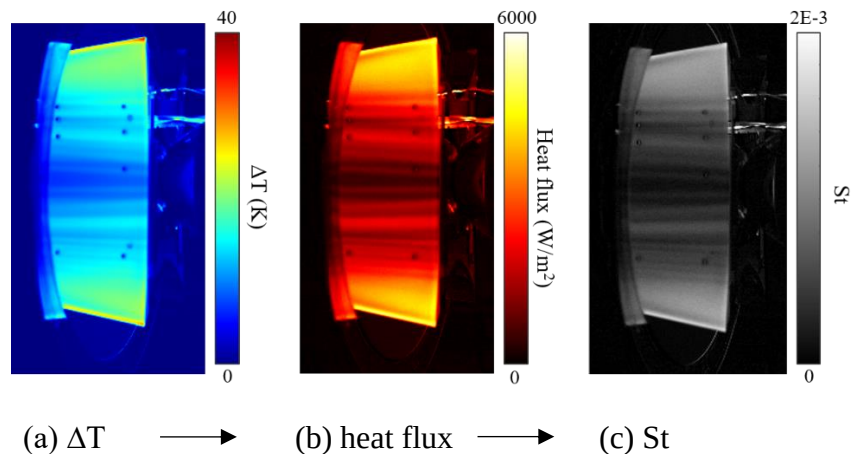


Figure 3-2. TAMU M6QT results at $Re/m = 10.2 \times 10^6$ with flow from left to right. The back 2.5 in of the BOLT model is seen with IR. (a) is the change in temperature, (b) is the calculated heat flux using Texas A&M's 1-D heat transfer code, and (c) is the Stanton number.

The TSP heat flux results imaged and processed at Purdue are shown in Figure 3-3 at $Re/m = 10.2 \times 10^6$. The small dots seen on the surface are the registration marks located 1-inch apart. The two larger, more defined dots on the surface downstream on the model are the location of the Schmidt Boelter gauges used to calculate heat flux from the TSP. The less-defined dots scattered about the surface are due to reflections within the facility. Even with different Reynolds numbers and diagnostics techniques, the quiet tunnel results from Purdue show similar distinct structures. The same streaks, whether at a Reynolds per meter of 10.2×10^6 or 12.4×10^6 , are visible in the IR and TSP, respectively. The inner 60% of the model in the M6QT at TAMU also compares with

the Purdue results of both diagnostic techniques. No onset of transition to turbulence is seen in any of the quiet flow results.

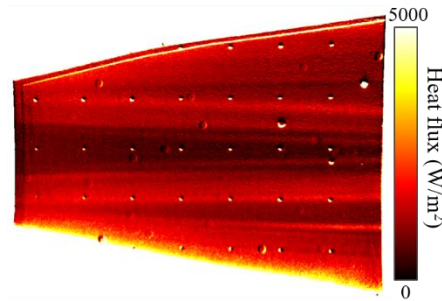


Figure 3-3. Temperature sensitive paint results on BOLT at Purdue University from a previous wind tunnel campaign of $Re/m = 10.2 \times 10^6$. Color map is in heat flux.

University of Minnesota ran a quiet dynamic numerical simulation (DNS) of a laminar CFD solution of BOLT at TAMU M6QT conditions for the case of $Re/m = 10.2 \times 10^6$. The steady state CFD results are obtained using a low dissipation, 4th order Kinetic Energy Consistent (KEC) numerical scheme for unstructured grids [32]. The wall is assumed to be isothermal at 300 K and is not altered to include any numerical roughness effects so as to maintain a 'smooth' wall. The computed solution is for half the domain due to the inherent symmetry and consists of approximately 160 million volume cells. For more details see [33]. Results from the laminar CFD solution for the 33% scale BOLT model are shown in Figure 3-4. The results show complex surface flow topology with predominant streaks. A secondary flow pattern is established from the concave curvature with highly-swept leading-edges such that the flow is pushed inward toward the center of the model, an effect similar to what was seen on elliptic cones [34, 35]. The boundary layer upon the center region of BOLT is relatively large, but towards the leading-edge region it is fairly thin. Thus, crossflow instabilities are a possibility in this region.

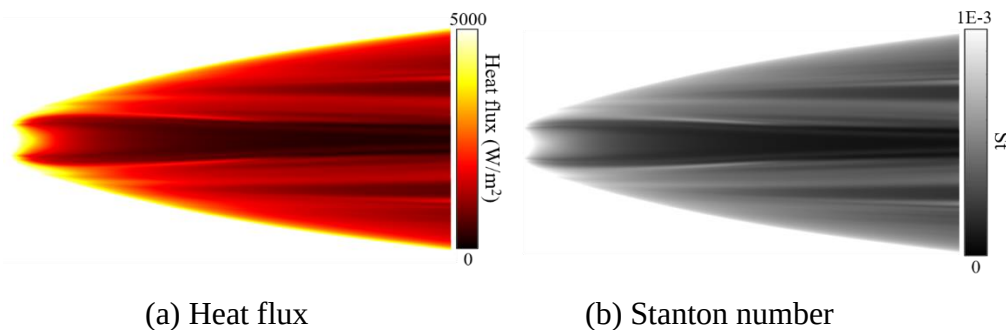


Figure 3-4. CFD laminar flow solution of the 33% scale BOLT model provided by The University of Minnesota. (a) the provided heat flux solution and (b) conversion to Stanton number at Texas A&M.

Each facility obtained remarkable IR imagery, where the flow features compared well with one another. For now, slice extractions of heat flux at the same position on BOLT were taken and

plotted with respect to one another for each of the diagnostics of each group. For reference of this slice, the CFD from Minnesota is given in Figure 3-5, where the white line denotes the position of the slice. The slice was taken approximately 0.4 inches from the back, just in front of the PCB instrumentation on the PEEK surface of the machined wind tunnel model. The y-axis in Figure 3-6 represents the heat flux and the x-axis is the non-dimensional length scale of BOLT. For comparison, TAMU IR on the 3-D printed polycarbonate model is shown, Purdue TSP and IR on the machined model is shown, and CFD results from Minnesota and TAMU are shown.

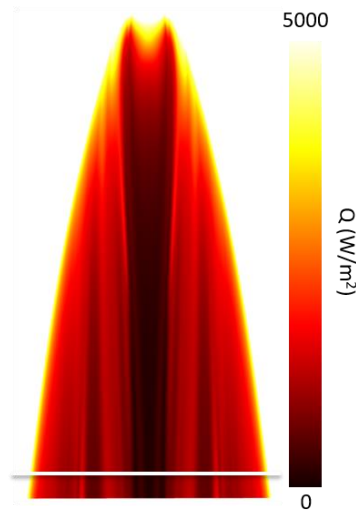


Figure 3-5. Slice extraction location on CFD from University of Minnesota.

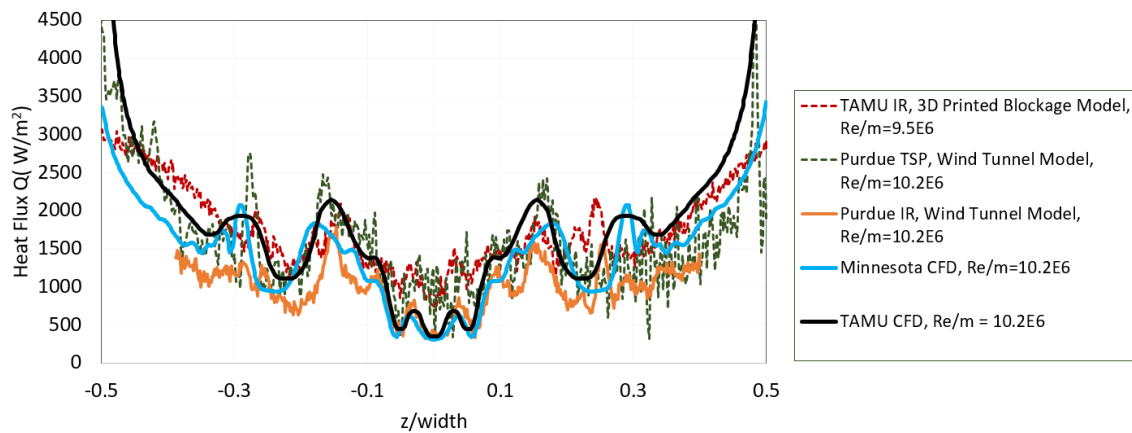


Figure 3-6. Line plot comparison of slice extractions on the machined 33% model, the 31% polycarbonate blockage model, TSP on the machined 33% model, and CFD from Minnesota and TAMU.

The results in Figure 3-6 are consistent across the research groups. Each group experiences similar streak-like structures at similar locations and the plot verifies it. Although the TSP is a bit noisy on the right-hand side of the graph, the left-hand side shows a cleaner line for comparison

with the rest of the groups. The results in Figure 3-6 are rather promising. With different calculation techniques from two CFD groups and different diagnostic techniques in two wind tunnel facilities, heat flux values compared considerably as well as the location of the streaks on the surface.

3.1.2 Conventional Tunnel Results

The following conventional wind tunnel results will be shown at various Reynolds numbers. The color scheme follows the same format as the quiet tunnel results. The full-view IR results of BOLT in the 20-inch Mach 6 Air Tunnel at NASA Langley are shown in Figure 3-7. Flow over the model in the images is from left to right with the small circles on the surface designating surface pressure transducers. Right away, it can be seen that the flow structures on the surface of BOLT in a conventional wind tunnel facility are completely different than the structures in a low-disturbance environment. This effect was previously seen on the HIFiRE-5 wind tunnel model. Instead of distinct streaks in the streamwise direction seen previously in quiet flow (Figure 3-1), two wedge-like structures originate from the back of the model and travel upstream with an increase in Reynolds number. The higher heating designates breakdown to turbulence has occurred.

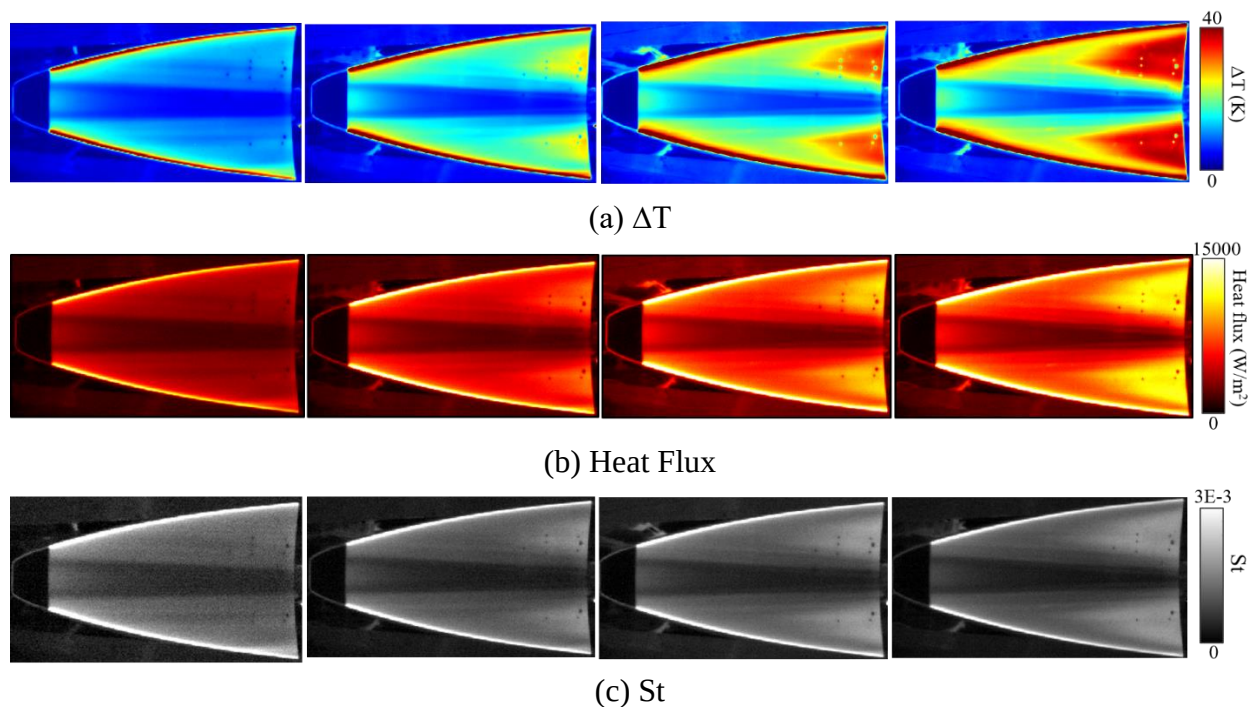


Figure 3-7. NASA Langley Mach 6 IR Thermography (Left to right: $Re/m = 3.5 \times 10^6$, 5.6×10^6 , 6.7×10^6 , 8.0×10^6). The entire model is shown in the facility. (a) is the change in temperature, (b) is the calculated heat flux using Texas A&M's 1-D heat transfer code, (c) is the Stanton number.

Similar results were seen on the surface of BOLT in the ACE facility at Texas A&M. With the flow from left to right, the turbulent structure on the surface in Figure 3-8 travels upstream on

the model as the Reynolds number is increased. The heat flux in the ACE wind tunnel facility is a third of the magnitude of that in the 20-inch Mach 6 tunnel. This factor is due to the initial conditions within the facility, i.e. the temperature in the facility at Langley is higher.

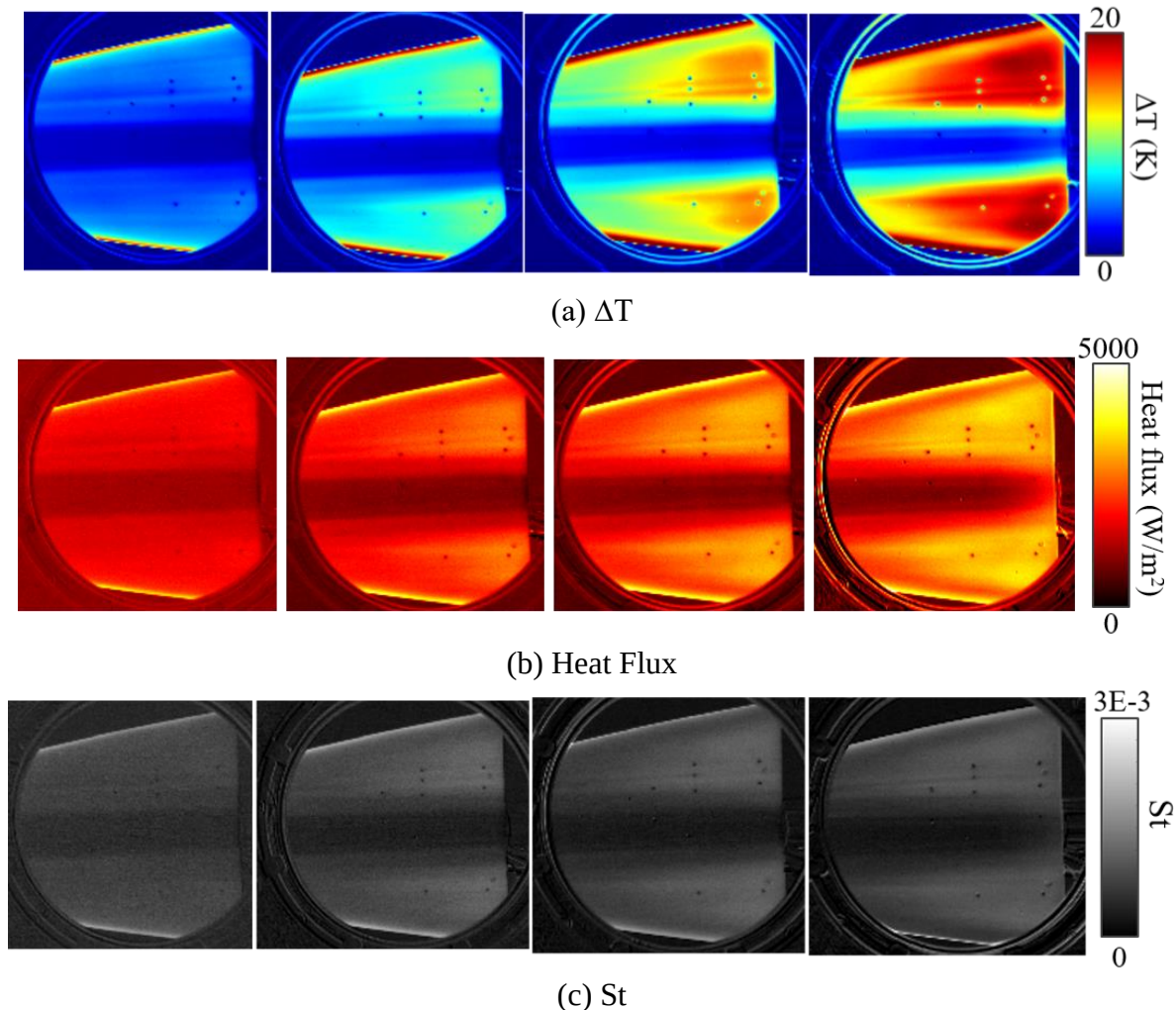


Figure 3-8. TAMU ACE Mach 6 IR Thermography results on the 33% Model (Left to right: $Re/m = 3.5 \times 10^6$, 5.6×10^6 , 6.7×10^6 , 8.0×10^6). The back 6 in of the model is shown. (a) is the change in temperature, (b) is the calculated heat flux using Texas A&M's 1-D heat transfer code, and (c) is the Stanton number.

The results in the BAM6QT at Purdue under noisy conditions are shown in Figure 3-9 at two Reynolds numbers, $Re/m = 3.5 \times 10^6$ and 8×10^6 . The flow is from left to right where the dots located on the surface represent 1-inch registration marks. Due to malfunctioning of the contraction Kulite in Purdue's tunnel, freestream conditions were inaccessible to convert to Stanton number. The wedge-like structure on the surface at a higher Reynolds number is similar to the results from Texas A&M and NASA Langley. However, the freestream disturbances located in the BAM6QT are much higher than the levels in ACE and the 20-inch Mach 6. This effect was

previously compared between A&M facilities and the BAM6QT conditions in Figure 2-10. The wedge itself is farther upstream on the model due to higher freestream disturbances within the facility. With a short run time, the heating on the model is low in the BAM6QT.

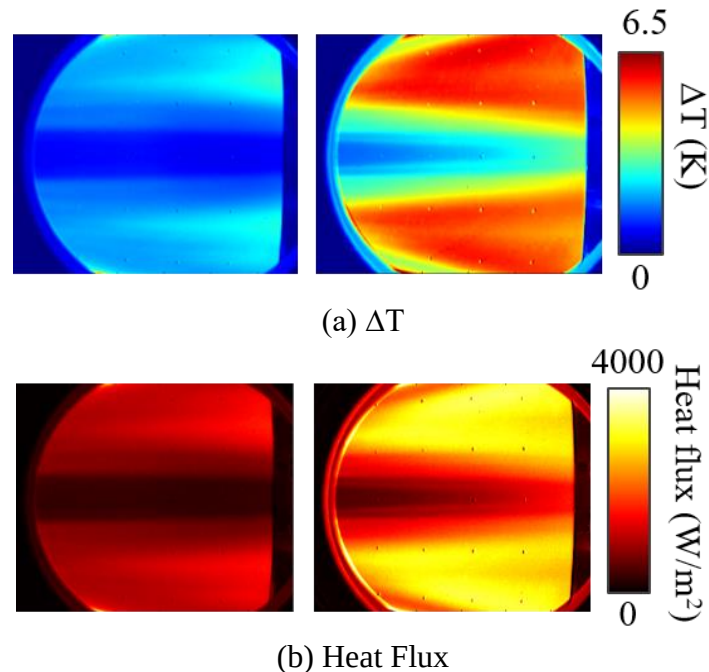


Figure3-9. Purdue (Noisy) Mach 6 IR thermography results on the 33% scale machined model (Left to right: $Re/m = 3.5 \times 10^6, 8.0 \times 10^6$). The back 6 inches of the model is shown. (a) is the change in temperature and (b) is the calculated heat flux using Texas A&M's 1-D heat transfer code

A 31% scale BOLT blockage wind tunnel model made of polycarbonate material was tested at two Mach numbers in the ACE facility. Mach 5.4 and Mach 7.4 lie within the Woomera flight trajectory and were desired for results. Figure 3-10 and Figure 3-11 are IR results at Mach 5.4 and Mach 7.4, respectively, in the ACE facility. The two circles located on the model were from a previous campaign involving surface pressure transducers. These results will not be shown here. With flow from left to right, wedge-like structures travel upstream on the model as the Reynolds number increases. This verifies the same effect of turbulence not only occurs at Mach 6, but also at lower and higher Mach numbers. The extra streaks visible in Figure 3-11 at Mach 7.4 are due to a slight melting effect of the leading edge at high Reynolds numbers and high temperatures.

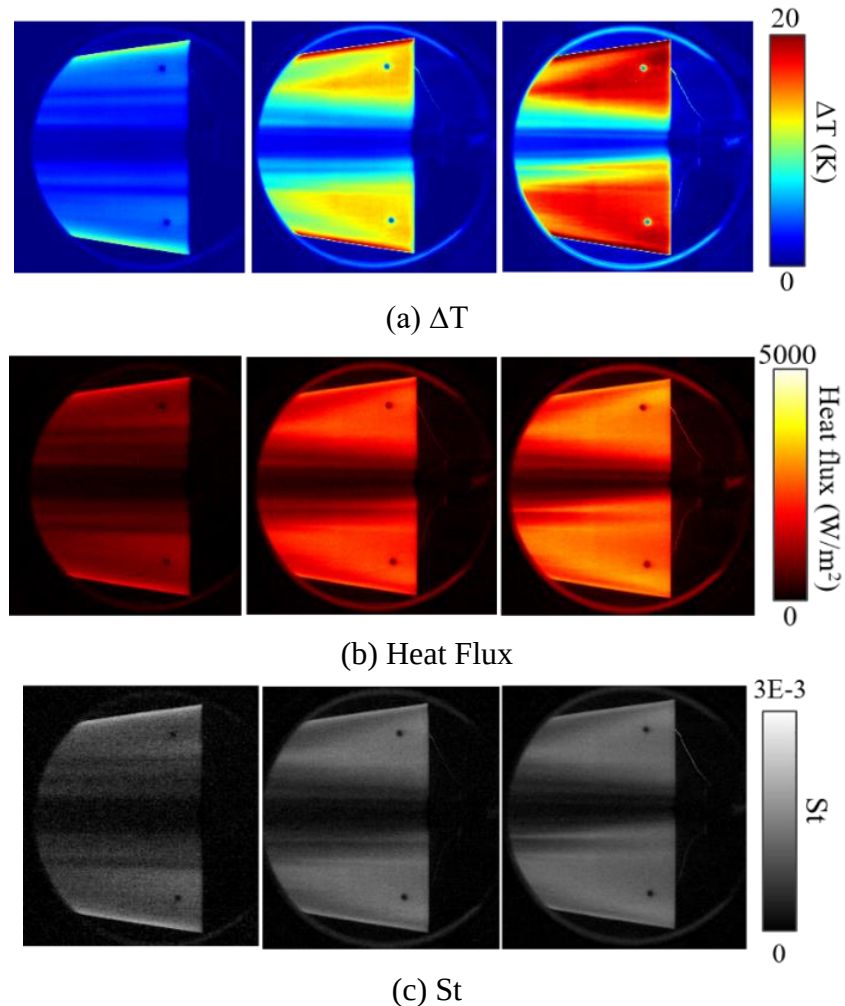


Figure 3-10. TAMU ACE Mach 5.4 IR thermography on the 31% blockage model of polycarbonate material (Left to right: $Re/m = 3.5 \times 10^6$, 5.6×10^6 , 6.8×10^6). (a) is the change in temperature, (b) is the calculated heat flux using Texas A&M's 1-D heat transfer code, and (c) is the Stanton number.

Testing at CUBRC was incorporated into this report to gain a better understanding of the differences and similarities on the surface of BOLT in a high-enthalpy environment on a full-scale model. With the capability of producing flight-like velocities, the environment in the LENS II facility is that of conventional flow. A full-scale BOLT model was tested at Mach 7.4 with a Reynolds per meter of 8×10^6 . The heating observed on the solid aluminum model, shown in Figure 3-12 with flow from left to right, was captured by TSP on the surface. The heat flux data was provided by JHUAPL. Even at high-enthalpy conditions, the turbulent wedges are present and the locations qualitatively compare with Figure 3-11c at the same Mach number and Reynolds number. The turbulent structures on the surface of BOLT in ACE, the BAM6QT (noisy conditions), 20-inch Mach 6, and LENS II are similar even at different Mach and Reynolds numbers.

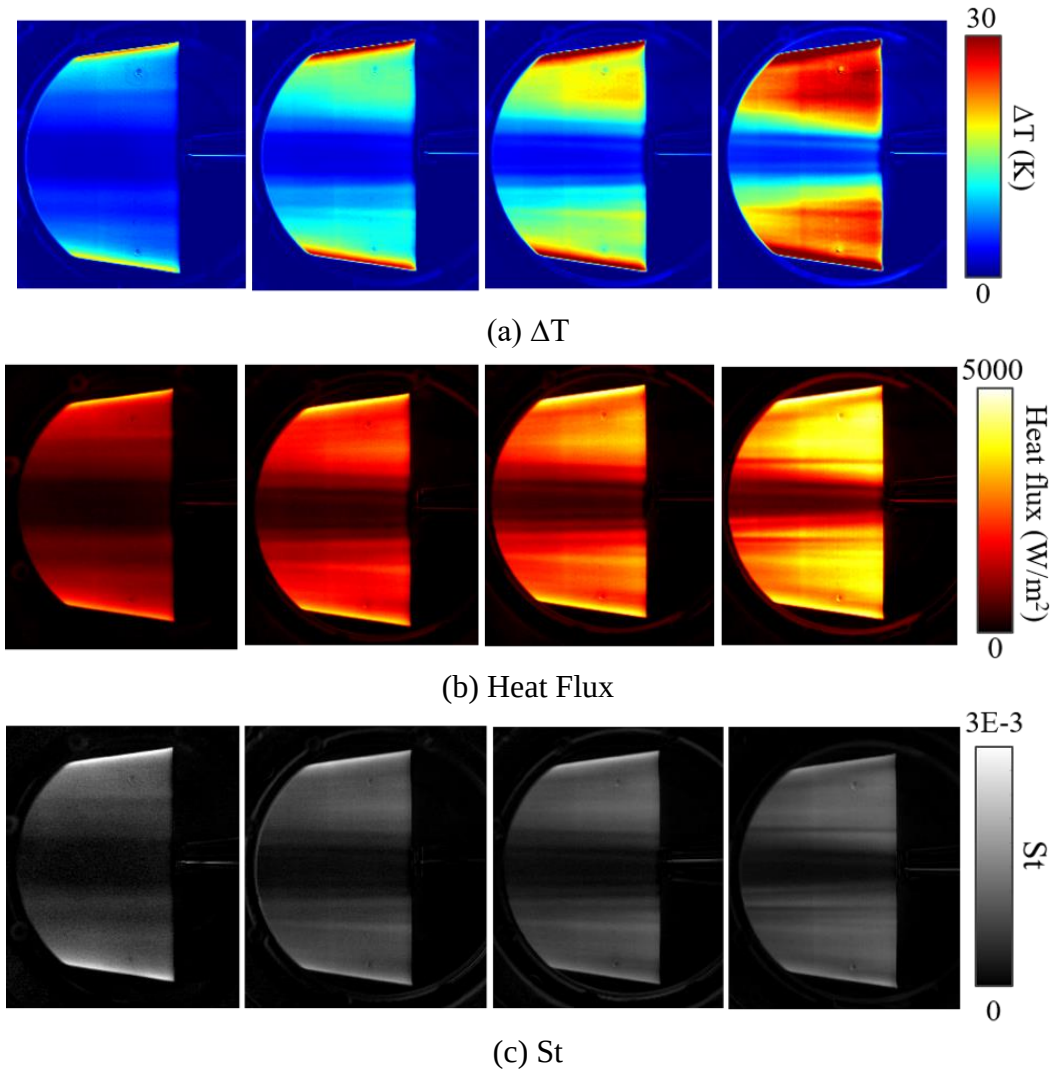


Figure 3-11. TAMU ACE Mach 7.4 IR thermography on the 31% blockage model of polycarbonate material (Left to right: $Re/m = 3.5 \times 10^6$, 5.6×10^6 , 6.8×10^6 , 8.0×10^6). (a) is the change in temperature, (b) is the calculated heat flux using Texas A&M's 1-D heat transfer code, and (c) is the Stanton number.

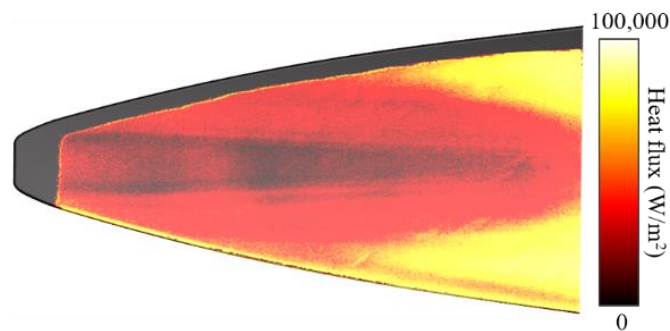


Figure 3-12. CUBRC Mach 7.4 TSP on the full-scale BOLT model in the LENS II facility at $Re/m = 8.0 \times 10^6$.

3.2 Heat Transfer Code and Schmidt Boelter Gauge Comparison

As previously mentioned in section 2.3, Texas A&M National Aerothermochemistry and Hypersonics Laboratory developed a MATLAB code for converting infrared temperature maps into heat flux. In this section, a comparison between the code and Schmidt Boelter gauge data will be assessed. The data analyzed involved roughness ‘trip-heights’ on the model to instigate transition that was tested at NASA Langley. Due to malfunction of the Schmidt Boelter gauges for ‘smooth’ (non-tripped) wind tunnel runs, wind tunnel runs with a roughness present will be analyzed. For this report, however, the roughness heights are not of interest; the heat flux values downstream on the model are.

The process involved the usual method of computing heat flux using A&M’s in-house heat transfer code for Langley images. The values in the IR image computed near the region of the Schmidt Boelter gauge were compared to the actual gauge data. The location of the gauge on the PEEK surface is shown for reference again in Figure 3-13, although the image itself is flipped to better correlate with the IR images.



Figure 3-13. Location of the Schmidt Boelter gauge for analysis

Six wind tunnel runs were chosen for comparison and are shown in Figure 3-14. The X and Y refer to pixel location in the image, the *Index* label is the heat flux value in W/m^2 , and R , G , B refer to the color mapping. Roughness trips can be seen on the model for different configurations in each image. The Schmidt Boelter gauge is located directly behind the point of measurement in Figure 3-14, seen as a red dot (also circled in Figure 3-13). The data from the gauge was provided to TAMU from NASA Langley for each wind tunnel run. The overall comparison between the two measurements, IR and sensor, are listed in Table 3-1. The data between the code and the sensor measurement agree within 6%.

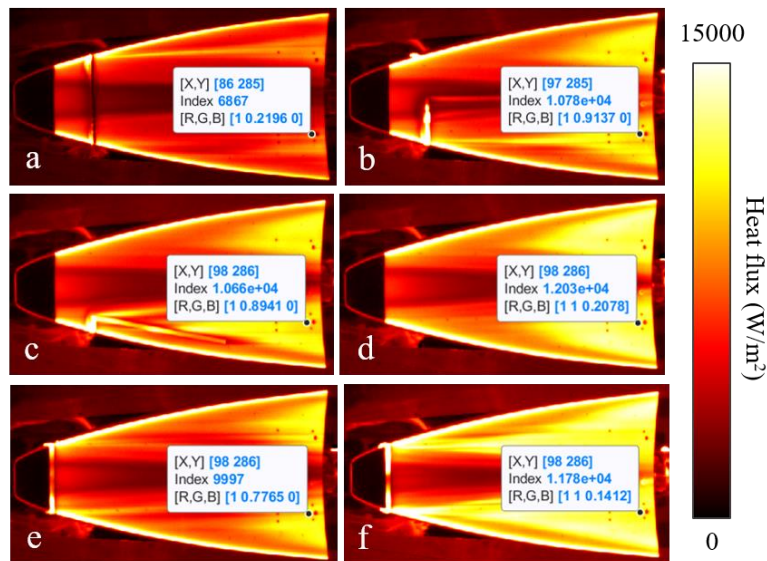


Figure 3-14. Heat flux data on the model for various wind tunnel runs. Reynolds number for each image from (a) to (f) is $\text{Re}/\text{m} = 4.9 \times 10^6$, 7.95×10^6 , 7.88×10^6 , 9.38×10^6 , 7.89×10^6 , and 9.47×10^6 , respectively. Images portray different roughness configurations.

Table 3-1. Heat transfer results and Schmidt Boelter gauge results

Reynolds Number	IR Heat Flux Data (kW/m^2)	Schmidt Boelter Data (kW/m^2)	Percentage %
4.90×10^6	6.87	6.50	5.39
7.95×10^6	10.78	10.93	1.39
7.88×10^6	10.66	10.62	0.38
9.38×10^6	12.03	12.56	4.41
7.89×10^6	10.00	9.96	0.40
9.47×10^6	11.78	11.32	3.90

However, when comparing other wind tunnel runs, the difference between the two variables varied more at the lower and higher range of Reynolds numbers within the campaign.

The frames chosen for comparison during the run would also alter the measurement. Thus, for this analysis, frames near 1.3 seconds into the run were chosen for less variation and uncertainty in the results. Schmidt Boelter measurements can be off from the theoretical heat transfer value by 20-25% as previously measured at Purdue on a straight, sharp cone at 0° AoA [36]. These results are preliminary and were a side objective proposed by the authors.

3.3 Surface Pressure Transducers

Surface pressure spectra were acquired with Kulite and PCB pressure transducers to help identify instability mechanisms. The IR visualization provided a basis to locate the instrumentation on the model.

3.3.1 Surface Pressure Spectra – Conventional Flow

In this section, PSD plots for surface pressure spectra are shown for conventional flow. The TAMU ACE and Langley 20-inch Mach 6 provided Reynolds number sweeps, while at Purdue a fixed Reynolds number condition is provided. All voltage measurements were calibrated to pressure. The legend in each PSD plot represents the spectra at a specific Reynolds number. The processing techniques and filtering for each facility were discussed earlier in Chapter 2, i.e. the sharper roll-off in the data from Langley is due to filter cut-off. Although there are 35 sensor locations on the model, only a few important locations will be discussed in this section. Appendix E contains the rest of the data for the surface pressure transducers. Not every facility collected data at every location which will be seen in the results.

The off-centerline Kulites proved to capture an instability in conventional flow in each facility. The Kulites in locations 1, 9 and 27 stand out with a modal growth around 30-40 kHz. The mode, first captured by K1, is present in all facilities and seen in Figure 3-15. In Figure 3-15a, the mode appears at a Reynolds per meter of 3.5×10^6 and continues to grow as the Reynolds number is increased. This is due to the nozzle sidewalls transitioning in the ACE facility and incorporating higher freestream disturbances on the surface of BOLT. The same mode is captured in the facilities at both Purdue and Langley and is seen in Figure 3-15b-c. The mode is present for all Reynolds numbers in Langley as well as the single data point spectra at Purdue. As the flow travels downstream on the model, K27 is the next off-centerline location to capture the same instability. The mode shows existence in the ACE facility around $Re/m = 3 \times 10^6$ and is present in the Purdue facility at the designated Reynolds number in Figure 3-16. Then traveling farther downstream on the model to K9, the mode is visible in all three facilities in Figure 3-17. Similar to the growth in K27, the mode begins to grow at $Re/m = 3 \times 10^6$ in the ACE facility. In Figure 3-17c, the instability is present at lower Reynolds numbers, but the freestream noise environment within the 20-inch Mach 6 masks the results at higher Re/m . The locations of these three Kulite sensors, K1, K9 and K27, prove to be of importance for future analyses.

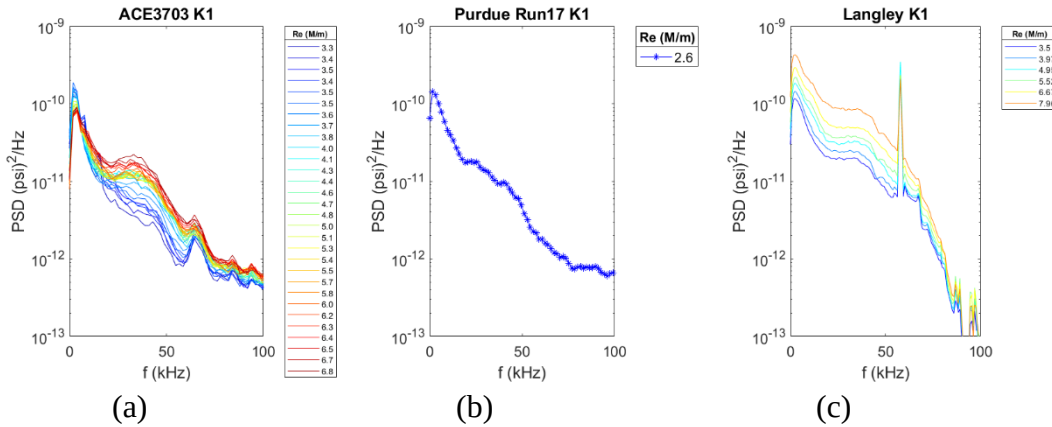


Figure 3-15. Results of surface pressure spectra in 3 conventional flow facilities for K1; (a) Texas A&M ACE Tunnel, (b) Purdue BAM6QT operated in conventional mode, (c) NASA Langley 20-inch Mach 6

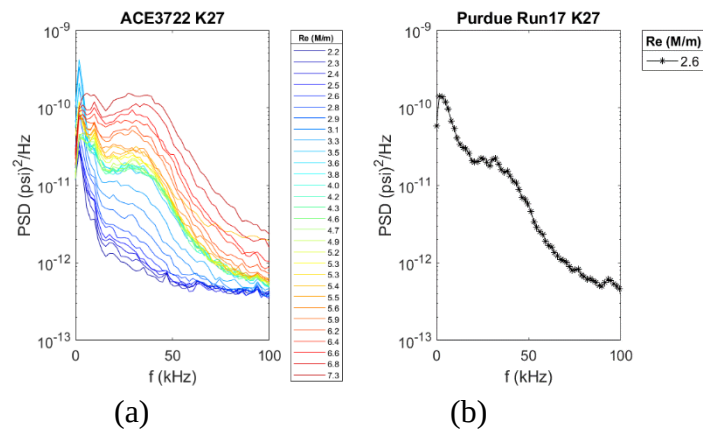


Figure 3-16. Results of surface pressure spectra in 2 conventional flow facilities for K27; (a) Texas A&M ACE Tunnel, (b) Purdue BAM6QT operated in conventional mode

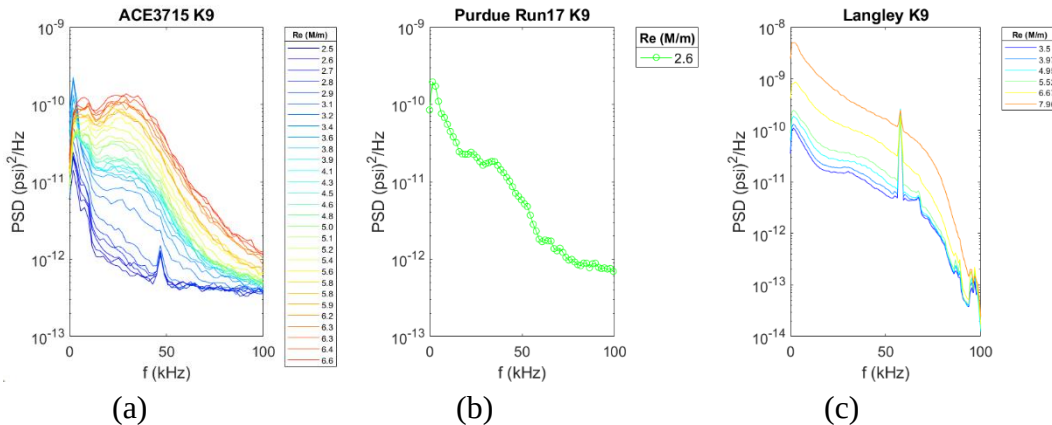


Figure 3-17. Results of surface pressure spectra in 3 conventional flow facilities for K9; (a) Texas A&M ACE Tunnel, (b) Purdue BAM6QT operated in conventional mode, (c) NASA Langley 20-inch Mach 6

The spectra from the PCB sensors did not show defined modal growth as previously seen in the off-centerline Kulites. Instead, broadband growth is seen as the Reynolds number is increased in the facility. All groups featured noise interference in the PCB sensors when mounted in the aluminum surfaces of the subscale model. Therefore, the majority of the PCB sensors are deemed unusable for data comparison. These data are located in Appendix E. In Figure 3-18, both A&M and Langley experience a broad growth with no modes apparent with an increase in Reynolds number. At the same streamwise location and mirrored to P8 is P11 in Figure 3-19. The data at P11 verifies the same results of P8 in the ACE facility. Similar broadband content is also observed in P25 and P31 in Figure 3-20 and Figure 3-21, respectively. The peaks seen in the spectra at Purdue in Figure 3-20b are due to oscillations in the sensor from electronic noise.

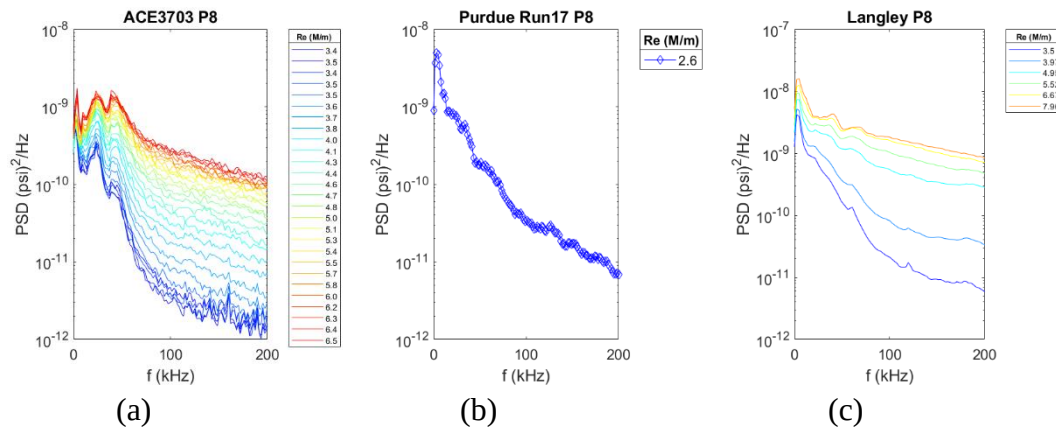


Figure 3-18. Results of surface pressure spectra in three conventional flow facilities for P8; (a) Texas A&M ACE Tunnel, (b) Purdue BAM6QT operated in conventional mode, (c) NASA Langley 20-inch Mach 6 Tunnel.

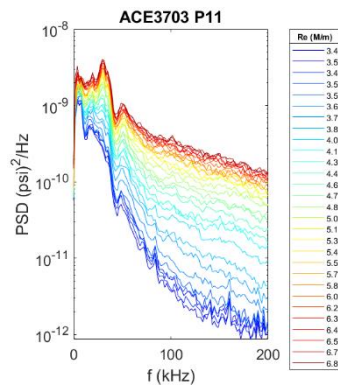


Figure 3-19. Results of surface pressure spectra in the Texas A&M ACE Tunnel for P11.

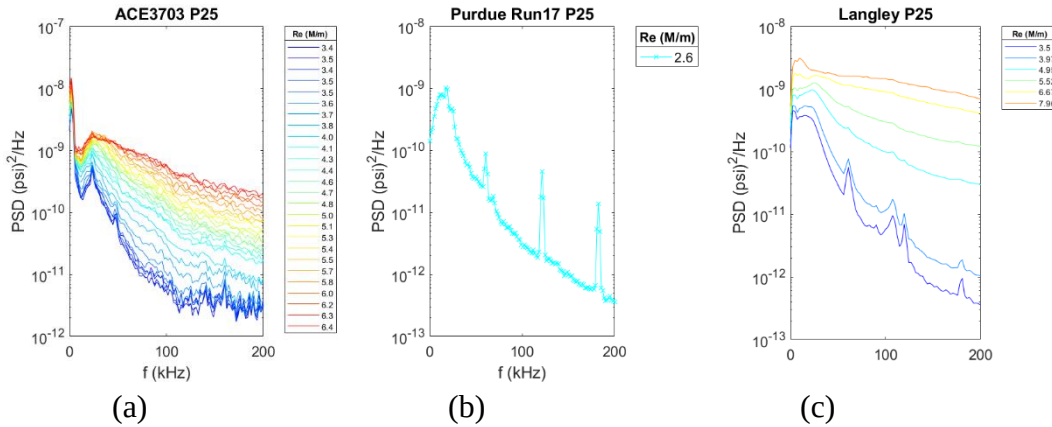


Figure 3-20. Results of surface pressure spectra in three conventional flow facilities for P25; (a) Texas A&M ACE Tunnel, (b) Purdue BAM6QT operated in conventional mode, (c) NASA Langley 20-inch Mach 6 Tunnel.

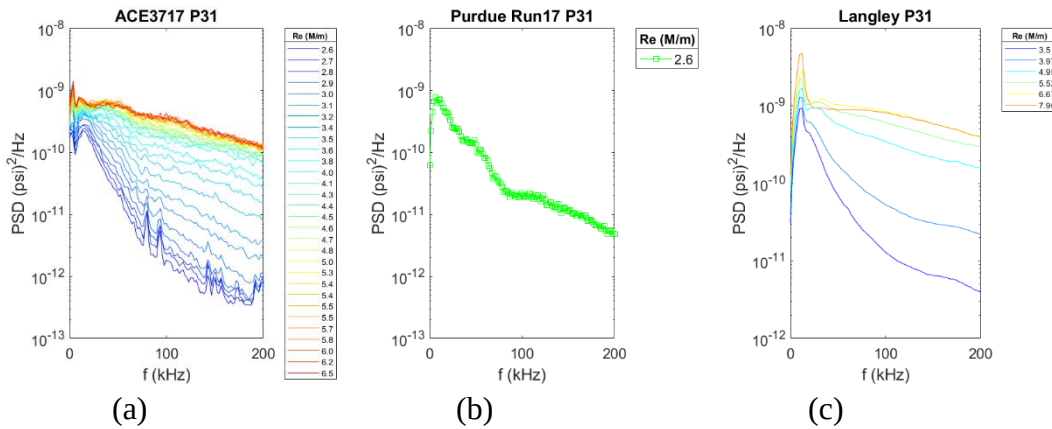


Figure 3-21. Results of surface pressure spectra in three conventional flow facilities for P31; (a) Texas A&M ACE Tunnel, (b) Purdue BAM6QT operated in conventional mode, (c) NASA Langley 20-inch Mach 6 Tunnel.

3.3.2 Surface Pressure Spectra – Quiet Flow

Quiet flow results in the M6QT and the BAM6QT for BOLT are presented in this section. It should be noted that Run 10 at Purdue contains an incorrectly installed nosetip. The other sensor results, (located in Appendix F) are taken from two different wind tunnel runs, Run 10 and Run 16, due to sensor malfunction. Each facility acquires data differently as previously stated in Table 2-3. However, the final results at Purdue are seen differently under quiet flow. Instead of a gradual ‘bump’ in the data to signify growth at a certain frequency that was seen by all facilities for conventional flow, sharp peaks at the frequency of suspected growth will be visible instead. This feature is quite different from their conventional flow results, so caution should be taken when analyzing the data in this section.

Taking a look at the same off-centerline Kulites analyzed for conventional flow, a similar 30 – 40 kHz mode is seen in quiet flow. The mode is first visible in the spectra in Figure 3-22 for both facilities. The mode is not seen in K1 until higher Reynolds numbers in the M6QT. Moving downstream to K27 in Figure 3-23, the mode is only apparent in the spectra at Texas A&M. This may be an effect from the nosetip installation. So far, the results are similar to what was observed under conventional flow. Lastly along the centerline, little to no growth is seen in K9 in Figure 3-24a but Figure 3-24b shows some content.

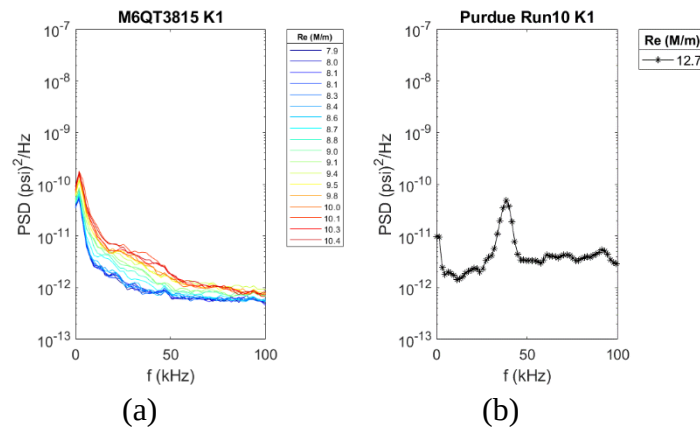


Figure 3-22. Results of surface pressure spectra in two quiet flow facilities for K1; (a) Texas A&M Mach 6 Quiet Tunnel and (b) the BAM6QT

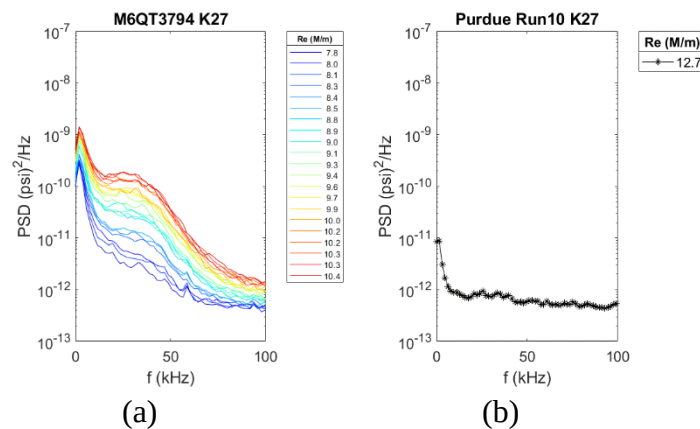


Figure 3-23. Results of surface pressure spectra in two quiet flow facilities for K27; (a) Texas A&M Mach 6 Quiet Tunnel and (b) the BAM6QT

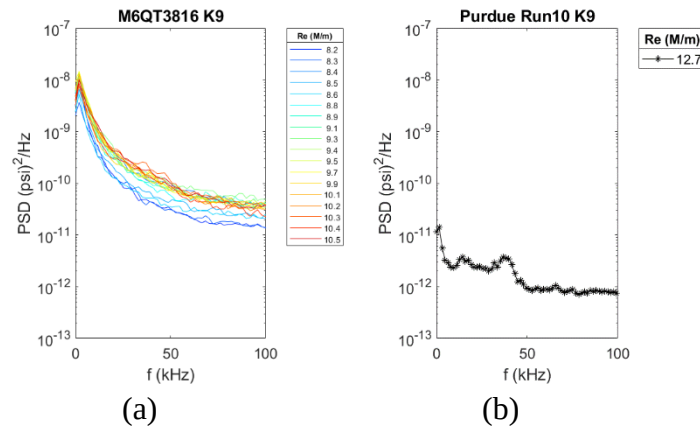


Figure 3-24. Results of surface pressure spectra in two quiet flow facilities for K9; (a) Texas A&M Mach 6 Quiet Tunnel and (b) the BAM6QT

In conventional flow, no modal growth was observed in K10 (see Appendix E). However, a 30 – 40 kHz mode is seen in both the M6QT and the BAM6QT in Figure 3-25. This result is still of discussion at this location. Model orientation, such as a slight yaw, could have caused a slight variation in the flow over the sensors during these wind tunnel runs. The nosetip installed before Run 10 may be a culprit to the added amplitudes in Figure 3-25b.

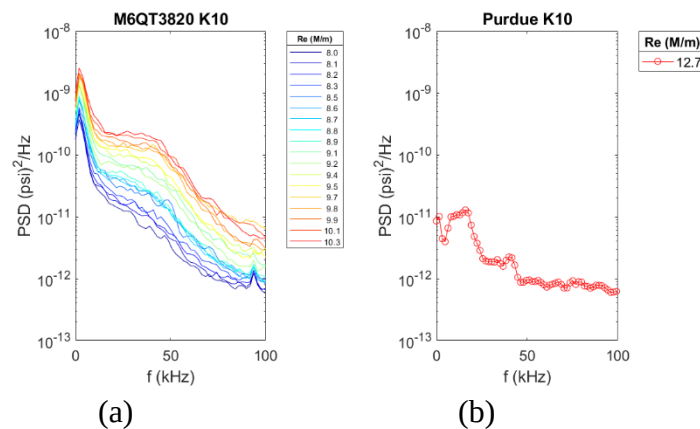


Figure 3-25. Results of surface pressure spectra in two quiet flow facilities for K10; (a) Texas A&M Mach 6 Quiet Tunnel and (b) the BAM6QT

The results of the PCB sensors mounted in the aluminum surfaces in quiet flow also contained noise interference similar to the conventional flow results. Thus, only results in the PEEK surface will be shown in this section. The results for P8 and P11 in Figure 3-26 and Figure 3-27, respectively, were similar to the results in conventional flow. Broadband growth is observed in both of the sensors in the M6QT, as well as mirrored results between the two locations. No modal growth was observed in any of the PCB sensors for both facilities.

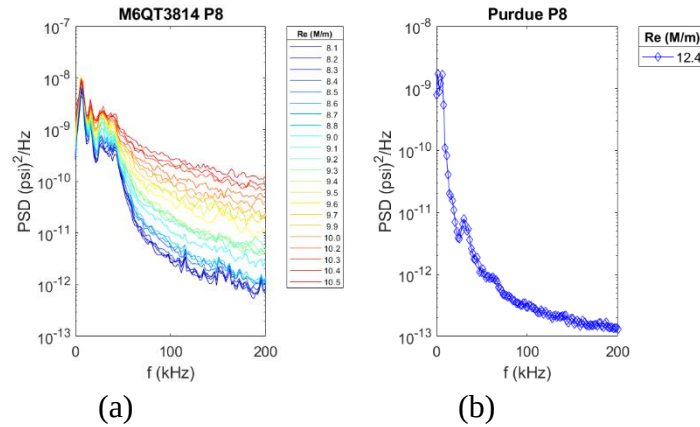


Figure 3-26. Results of surface pressure spectra in two quiet flow facilities for P8; (a) Texas A&M Mach 6 Quiet Tunnel and (b) the BAM6QT

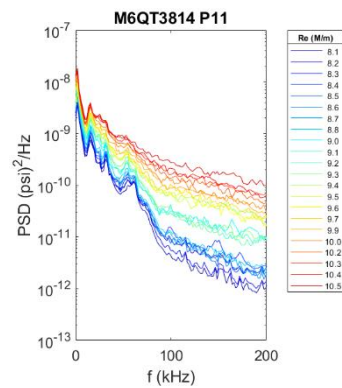


Figure 3-27. Results of surface pressure spectra in the Texas A&M M6QT for P11.

3.4 Off-Body Hot-wire Measurements

The hot-wire probe is mounted within the M6QT on a three-axis traverse previously shown in Section 2. A photograph of the TAMU BOLT model installed in the M6QT with a hot-wire present is shown in Figure 3-28. Before tunnel start-up and shutdown, the wire is hidden behind the exit of the nozzle, out of the flow, to limit wire breakage. Measurements of Cartesian coordinates were converted to cylindrical for off-body measurements of BOLT.



Figure 3-28. Hot-wire located above the surface of BOLT in the M6QT.

Hot-wire measurements were performed on the TAMU BOLT model to qualitatively compare to the off-body flow structures from the simulation results by the University of Minnesota [37, 38]. The focus of the measurements was the vortical mode region to better quantify the spatial distribution of the spectral content within the roll-up structure for simulation validation. The streamwise location of interest was $x = 240$ mm from the leading edge of the 33% scale model. A blown-up view of the vortical roll-up structure is shown in Figure 3-29. The mean mass flux contour is shown in Figure 3-29b, with the dominant frequency at this location plotted in Figure 3-29c. The x-axis represents the z-location in meters from off-centerline. The y-axis represents the location from the center of the geometry in meters, not the location from off the surface. The roll-up structure in is of interest. In summary, these plots show how the flow instabilities, indicated by larger fluctuation levels, are coupled to the secondary flow structure. This result is consistent with the mean flow analyses of Mullen et al [39].

An experimental hot-wire campaign was performed to map the vortex region shown in Figure 3-29. Listed in Table 3-2 is a summary of the traverse run measurement locations. The z- and y-locations correspond to those in Figure 3-29a (right image) [37], but in millimeters. All hot-wire results were acquired at a unit Reynolds number of 9.9×10^6 , which matches the conditions for the DNS data.

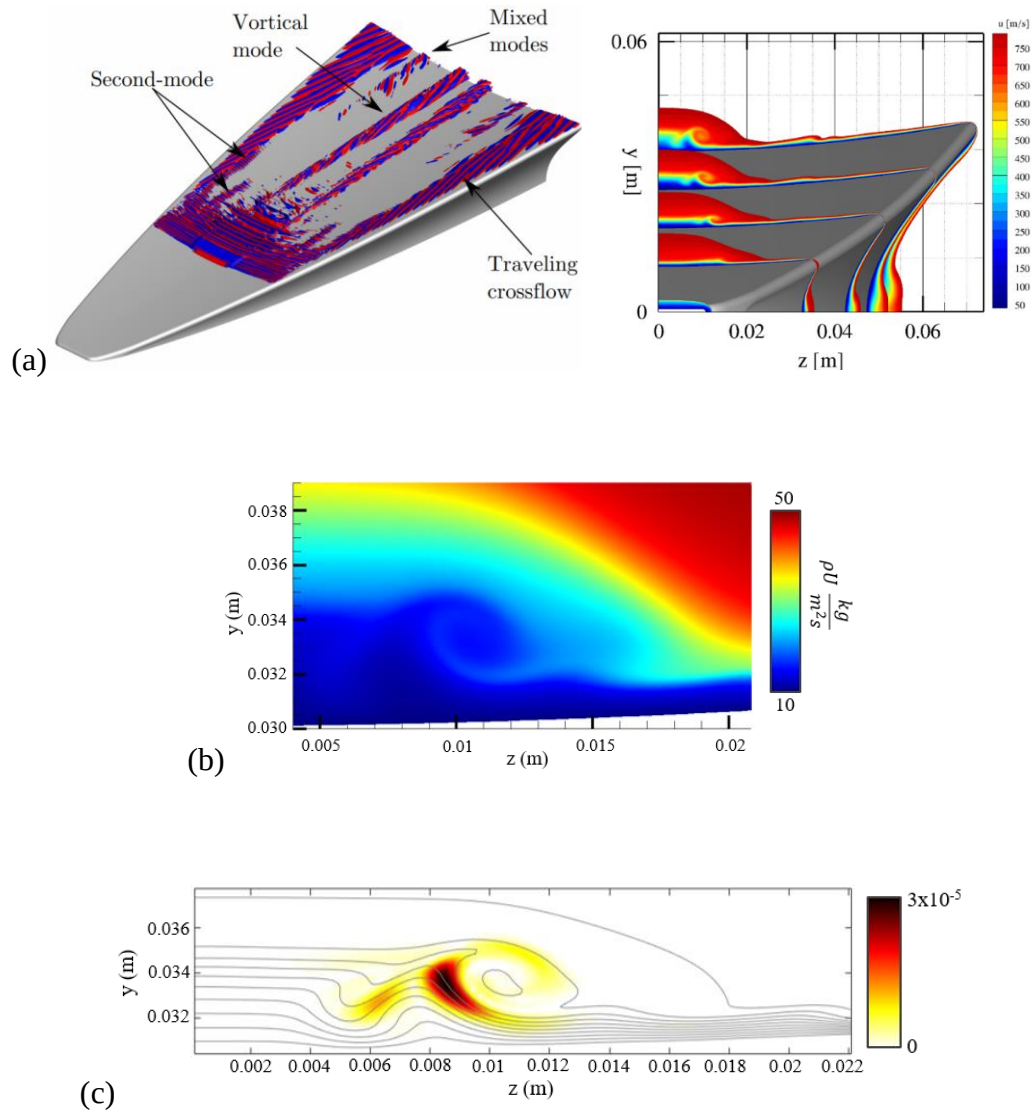


Figure 3-29. Numerical simulations of BOLT instabilities. (a) Isosurface of streamwise velocity perturbation on a 33% scale BOLT geometry at $M = 6.0$, $Re/m = 9.9 \times 10^6$ at $x = 0.24$ m. (b) Mass flux¹, (c) Magnitude of the u'_ξ mode.²

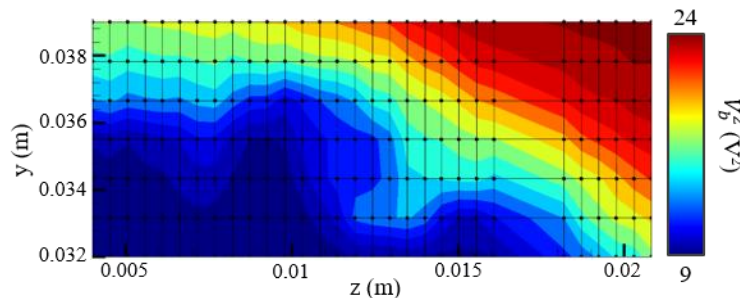
¹ Data provided by John Thome, University of Minnesota, 2019.

² Courtesy of Anthony Knutson, University of Minnesota, 2019.

Table 3-2 Hot-Wire Measurement Locations

Run #	Z-location (mm)	Y-location (mm)
4098	8.7 9.3 9.8	32-39
4099	10.3 10.8 11.4	32-39
4100	11.9 12.4 12.9	32-39
4101	13.5 14 14.5	32-39
4103	7.2 7.7 8.2	32-39
4104	5.6 6.1 6.6	32-39
4105	4 4.5 5.1	32-39
4108	15 15.6 16.1	32-39
4112	18.2 18.7 19.2	32-39
4113	19.8 20.3 20.8	32-39

The first comparison is the mean flow. At high over heat ratio, the hot-wire probe is primarily sensitive to mean mass flow [40], and for the relatively low-density flow, the King's law exponent has been found to be close to unity [41]. With this, the mean mass flux is roughly proportional to the square of mean bridge voltage from the anemometry. Hence, the square of the bridge voltage from the experiments is shown in Figure 3-30. Hot-wire measurement locations are represented by the black dots in Figure 3-30 with the corresponding mesh of the contour. The x-axis represents the z-location in meters from the centerline of the model and the y-axis is from the center of the geometry in meters. The data between $z = 16.1$ - 18.2 mm was influenced by noise and thus discarded from the linear interpolation. The contour results are compared to the simulation in Figure 3-29. The location and structure of the vortex, as well as the boundary layer edge and freestream environment locations in Figure 3-30 are qualitatively similar to those in Figure 3-29b. The experimental resolution was insufficient to fully resolve the detailed structure as shown in Figure 3-29a. The experiments appeared to have been shifted 1-2 mm as compared to the computation, which may have been the result of a slight misalignment of the wind tunnel model. Nonetheless, the vortex structure was present in the experiment.

**Figure 3-30.** Visualization of the vortex structure from the hot-wire measurements.

The second comparison is a 2-D contour map of the fluctuations within the vortex structure. The hot-wire RMS of the fluctuating voltage normalized by the mean voltage is plotted in Figure 3-31a. This ratio is approximately proportional to the mass flux fluctuation ratio [40]. The RMS was calculated by numerically integrating the PSD spectra across all frequencies. The fluctuation energy within the vortex structure is apparent and resembles the shape in Figure 3-29b. Additional content is seen closer to the surface by the hot-wire between 15-20 mm that is not present in the simulations in Figure 3-29c. The data that was discarded may be of influence in amplifying these smaller structures. To quantify the frequency content within the vortex structure, power spectra at various y -locations are plotted in Figure 3-31b at $z = 10.8$ mm. The frequency scale is from 10-100 kHz with each PSD line corresponding to a specific y -location. Modal growth is apparent in the range of 25-40 kHz in Figure 3-31b as the hot-wire moves in and out of the structure shown in Figure 3-31a, that is for $y = 33$ -37 mm. Strain-gauging is apparent at 20 and 60 kHz, but do not affect the overall results. The PSD spectra were also integrated across the frequency range of 25-40 kHz. These results are plotted in Figure 3-32. Within this frequency band, the top of the vortex structure in Figure 3-32 has faded, content to the left of the structure is now visible, and the structure from $z = 15$ -20 mm is not as prominent. The overall structure from the experiments resembles that of the computation in Figure 3-29c. To help clarify the peak behavior spatially, line plots of the normalized energy at 35 kHz and the RMS energy within the 25-40 kHz band are plotted in Figure 3-33a-b, respectively, for $z = 10.8$ mm, which corresponds to the high energy location in Figure 3-29b. The peak and RMS fluctuation values increase by a factor of approximately 2.5 within the vortex region ($y = 32$ -37 mm). Overall, the spectral content and location within the mean flow structure agree with the simulations from the University of Minnesota.

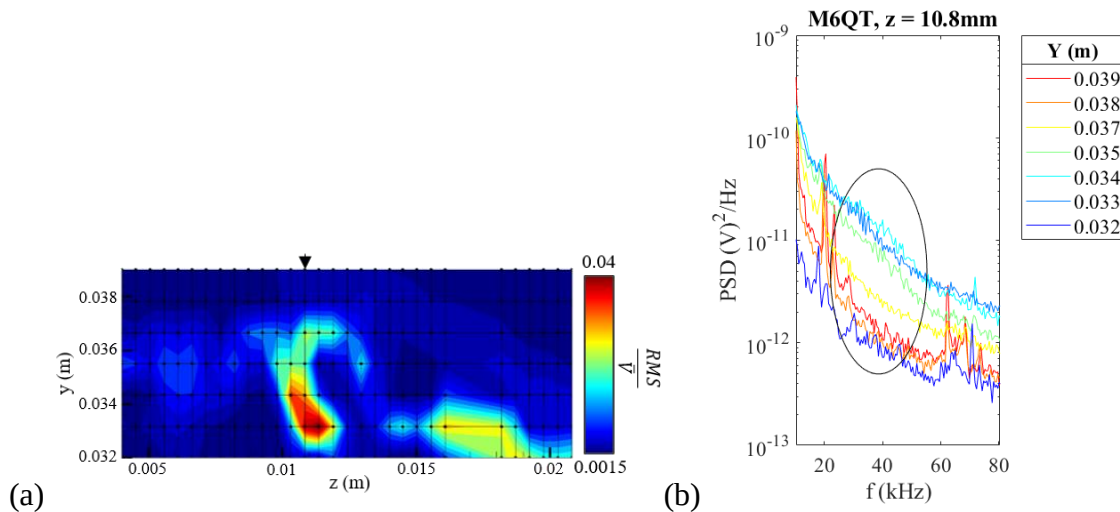


Figure 3-31. Hot-wire results. (a) $\text{RMS}/\bar{V}$ with arrow at $z=10.8$ mm, (b) PSD spectra at location $z=10.8$ mm

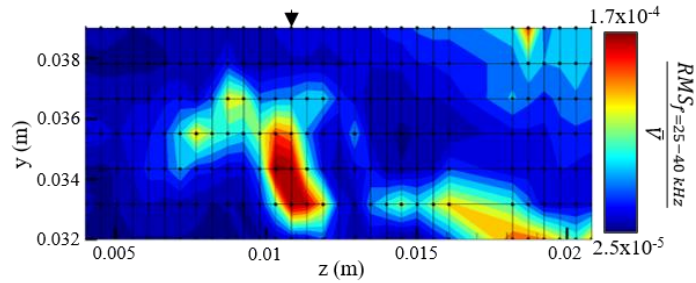


Figure 3-32. Hot-wire $RMS/\bar{V}$ results between the frequency band 25-40 kHz with arrow at $z=10.8\text{mm}$.

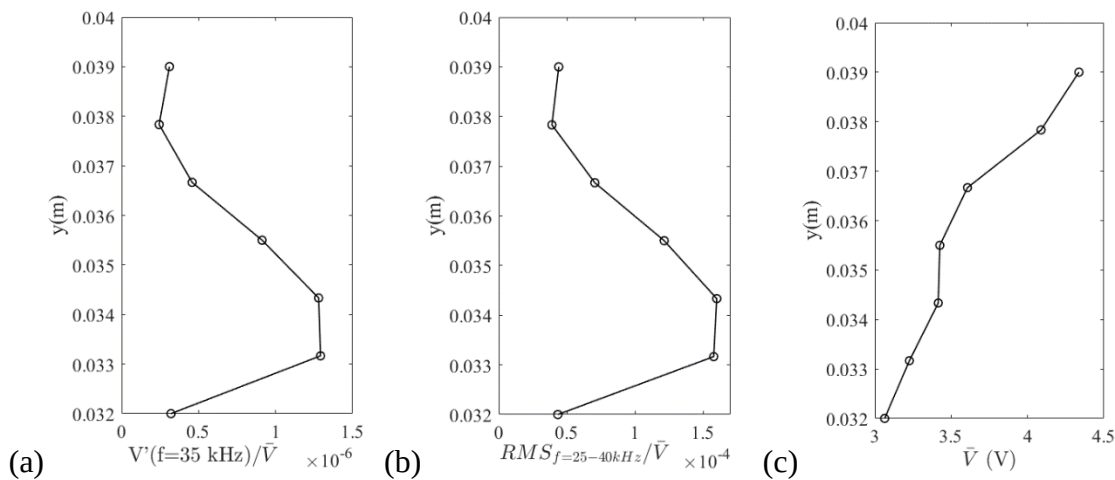


Figure 3-33. Hot-wire results at location $z=10.8$ mm. (a) $V'/\bar{V}$ at 35 kHz, (b) $RMS/\bar{V}$ results between the frequency band 25-40 kHz, (c) $\bar{V}$ (volts)

REFERENCES

1. Kostak, H., et al., *Freestream Disturbance Effects on Boundary Layer Instability and Transition on the AFOSR BOLT Geometry*, in *AIAA Scitech 2019 Forum*. 2019.
2. Kostak, H., Bowersox, R., "Hypersonic Boundary Layer Off-Body and Surface Measurements on the AFOSR BOLT," AIAA-2020-1043, AIAA SCITECH Forum, Jan. 2020.
3. Berridge, D., Kostak, H., McKiernan, G., King, R., Wheaton, B., Wolf, T., and Schneider, S., 2018 "Hypersonic Ground Tests with High-Frequency Instrumentation in Support of the Boundary Layer Transition (BOLT) Flight Experiment," AIAA 2019-0090.
4. Kostak, H. and Bowersox, R., "Pre-Flight Ground Test Analyses of the Boundary Layer Transition (BOLT) Flight Geometry," *Journal Spacecraft and Rockets*, Vol. 58, No.1, Jan.-Feb., 2021, pp. 67-77.
5. Berridge, D.C., et al., *Hypersonic Ground Tests In Support of the Boundary Layer Transition (BOLT) Flight Experiment*, in *2018 Fluid Dynamics Conference*. 2018.
6. Berridge, D.C., et al., *Hypersonic Ground Tests With High-Frequency Instrumentation In Support of the Boundary Layer Transition (BOLT) Flight Experiment*, in *AIAA Scitech 2019 Forum*. 2019.

7. Leidy, A.N., *An experimental characterization of 3-d transitional shock wave boundarylayer interactions at mach 6*. 2019, Texas A&M University.
8. Hofferth, J., R. Bowersox, and W. Saric, *The Mach 6 Quiet Tunnel at Texas A&M: Quiet Flow Performance*, in *27th AIAA Aerodynamic Measurement Technology and Ground Testing Conference*. 2010, American Institute of Aeronautics and Astronautics.
9. Blanchard, A.E., J.T. Lachowicz, and S.P. Wilkinson, *NASA Langley Mach 6 Quiet Wind-Tunnel Performance*. AIAA Journal, 1997. **35**(1): p. 23-28.
10. Saric, W.S., *Görtler Vortices*. Annual Review of Fluid Mechanics, 1994. **26**(1): p. 379-409.
11. Craig, S.A., *Stability of High-Speed, Three-Dimensional Boundary Layers*. 2015, Texas A&M University.
12. Hofferth, J. and W. Saric, *Boundary-Layer Transition on a Flared Cone in the Texas A&M Mach 6 Quiet Tunnel*, in *50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*. 2012, American Institute of Aeronautics and Astronautics.
13. Hofferth, J.W., *Boundary-Layer Stability and Transition on a Flared Cone in a Mach 6 Quiet Wind Tunnel*. 2013.
14. Tichenor, N.R., *Characterization of the Influence of a Favorable Pressure Gradient on the Basic Structure of a Mach 5.0 High Reynolds Number Supersonic Turbulent Boundary Layer*. 2010, Texas A&M University.
15. Semper, M., B. Pruski, and R. Bowersox, *Freestream Turbulence Measurements in a Continuously Variable Hypersonic Wind Tunnel*, in *50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*. 2012, American Institute of Aeronautics and Astronautics.
16. Mai, C.L.N., *Near-region modification of total pressure fluctuations by a normal shock wave in a low-density hypersonic wind tunnel*. 2014, Texas A&M University.
17. Borg, M.P., et al., *Freestream Effects on Boundary Layer Disturbances for HIFiRE-5*, in *53rd AIAA Aerospace Sciences Meeting*. 2015.
18. Neel, I.T., *Influence of Environmental Disturbances on Hypersonic Crossflow Instability on the HIFiRE-5 Elliptic Cone*. 2019, Texas A&M University.
19. Steen, L.-c.E., *Characterization and development of nozzles for a hypersonic quiet wind tunnel*. 2010, Purdue University.
20. Sweeney, C.J., *Characterization of a hypersonic quiet wind tunnel nozzle*. 2016, Purdue University.
21. Berger, K.T., et al., *NASA Langley Aerothermodynamics Laboratory: Hypersonic Testing Capabilities*, in *53rd AIAA Aerospace Sciences Meeting*. 2015, American Institute of Aeronautics and Astronautics.
22. Wadhams, T., M. MacLean, and M. Holden, *Continuing Experimental Studies of High Speed Boundary Layer Transition in LENS Facilities to Further the Development of Predictive Tools for Boundary Layer Transition in Flight*, in *51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*. 2013, American Institute of Aeronautics and Astronautics.
23. Neel, I.T., et al., *Characterization of Environmental Disturbances on Hypersonic Crossflow Instability on the HIFiRE-5 Elliptic Cone*, in *22nd AIAA International Space Planes and Hypersonics Systems and Technologies Conference*. 2018.
24. Boyd, C.F. and A. Howell, *Numerical investigation of one-dimensional heat-flux calculations*, in *Unknow*. 1994.
25. Borg, M.P. and R.L. Kimmel, *Simultaneous Infrared and Pressure Measurements of Crossflow Instability Modes for HIFiRE-5*, in *54th AIAA Aerospace Sciences Meeting*. 2016.

26. Juliano, T.J., L. Paquin, and M.P. Borg, *Measurement of HIFiRE-5 Boundary-Layer Transition in a Mach-6 Quiet Tunnel with Infrared Thermography*, in 54th AIAA Aerospace Sciences Meeting. 2016.
27. Hubner, J.P., B.F. Carroll, and K.S. Schanze, *Heat-Transfer Measurements in Hypersonic Flow Using Luminescent Coating Techniques*. Journal of Thermophysics and Heat Transfer, 2002. **16**(4): p. 516-522.
28. Liu, T. and J.P. Sullivan, *Pressure and temperature sensitive paints*. T. Liu, J.P. Sullivan. Experimental fluid mechanics. 2005: Springer.
29. Beresh, S.J., et al., *Fluctuating wall pressures measured beneath a supersonic turbulent boundary layer*. Physics of Fluids, 2011. **23**(7): p. 075110.
30. Ort, D. and J.J. Dosch, *Influence of Mounting on the Accuracy of Piezoelectric Pressure Measurements for Hypersonic Boundary Layer Transition*, in AIAA Scitech 2019 Forum. 2019, American Institute of Aeronautics and Astronautics.
31. Craig, S.A. and W.S. Saric, *Experimental study of crossflow instability on a Mach 6 yawed cone*, in 45th AIAA Fluid Dynamics Conference. 2015, American Institute of Aeronautics and Astronautics.
32. Subbareddy, P.K. and G.V. Candler, *A fully discrete, kinetic energy consistent finite-volume scheme for compressible flows*. Journal of Computational Physics, 2009. **228**(5): p. 1347-1364.
33. Thome, J., et al., *Direct numerical simulation of BOLT hypersonic flight vehicle*, in 2018 Fluid Dynamics Conference. 2018.
34. Li, F., et al., *Stability Analysis for HIFiRE Experiments*, in 42nd AIAA Fluid Dynamics Conference and Exhibit. 2012, American Institute of Aeronautics and Astronautics.
35. Kimmel, R.L., J. Poggie, and S.N. Schwoerke, *Laminar-Turbulent Transition in a Mach 8 Elliptic Cone Flow*. AIAA Journal, 1999. **37**(9): p. 1080-1087.
36. Sullivan, J., et al., *Quantitative Global Heat Transfer in a Mach-6 Quiet Tunnel*. 2012.
37. Thome, J., Knutson, A., and Candler, G. V., "Boundary layer instabilities on BoLT subscale geometry," AIAA Paper 2019-0092, Jan. 2019.
38. Knutson, A. L., Thome, J. S., and Candler, G. V., "Numerical Simulation of Instabilities in the Boundary-Layer Transition Experiment Flowfield". *Journal of Spacecraft and Rockets*. published online Dec. 2019.
39. Mullen, C. D., Moyes, A., Kocian, T. S., and Reed, H. L., "Heat Transfer and Boundary-Layer Stability Analysis of Subscale BOLT and the Fin Cone," AIAA Paper 2019-3081, June 2019.
40. Bowersox, R. D. W., "Combined laser Doppler velocimetry and cross-wire anemometry analysis for supersonic turbulent flow". *AIAA Journal*. Vol. 34, No. 11, 1996. pp. 2269-2275. subscale geometry," AIAA Paper 2019-0092, Jan. 2019.
41. Semper, M. T. and Bowersox, R. D. W., "Tripping of a Hypersonic Low-Reynolds-Number Boundary Layer". *AIAA Journal*. Vol. 55, No. 3, 2017. pp. 808-817.

NOMENCLATURE

f	frequency
i	spatial coordinate for heat conduction equation
IR	Infrared
K#	Kulite at location #
m	meter
M	Mach number

n	time coordinate for heat conduction equation
psi	lbs/in ²
p_0	tunnel stagnation pressure
p_{t2}	Pitot pressure in test section (p_{t2} = mean, p'_{t2} = rms)
P#	PCB at location #
Q	heat flux in W/m ²
Re	Reynolds number
r/r_{exit}	fractional lateral distance along nozzle plane
rms	root mean square
St	Stanton number
t	time
T	temperature in kelvin
μm	micrometer
width	z-span of BOLT in inches
X/L	fractional distance along nozzle length
z	z-location on BOLT (in inches or meters)
ΔT	change in temperature in kelvin
'	fluctuation

F. PUBLICATIONS

- Berridge, D., Kostak, H., McKiernan, G., King, R., Wheaton, B., Wolf, T., Schneider, S.: Hypersonic Ground Tests with High-Frequency Instrumentation in Support of the Boundary Layer Transition (BOLT) Flight Experiment. AIAA 2019-0090 (2019)
- Kostak, H., Bowersox, R., McKiernan, G., Thome, J., Candler, G., King, R.: Freestream Disturbance Effects on Boundary Layer Instability and Transition on the AFOSR BOLT Geometry. AIAA-2019-0088 (2019)
- Kostak, H., Bowersox, R.: Hypersonic Boundary Layer Off-Body and Surface Measurements on the AFOSR BOLT. AIAA-2020-1043 (2020)
- Kostak, H., Bowersox, R.: Pre-Flight Ground Test Analyses of the Boundary Layer Transition (BOLT) Flight Geometry. Journal Spacecraft and Rockets, Vol. 58, No.1, pp. 67-77 (2021)
- Moyes, A., Kocian, T., Mullen, D., Reed, H.: Pre-Flight Boundary-Layer Stability Analysis of BOLT Geometry. AIAA-2018-2895, (2018)
- Moyes, A., Reed, H.: Nonlinear Boundary-Layer Stability Analysis of BOLT and HIFiRE-5. AIAA-2019-2972 (2019)
- Moyes, A., Reed, H.: Preflight Boundary-Layer Stability Analysis of BOLT Geometry. Journal of Spacecraft and Rockets, Vol. 58, No. 1, pp. 78-89 (2021)
- Mullen, D., Moyes, A., Kocian, T., Reed, H.: Heat Transfer and Boundary-Layer Stability Analysis of Subscale BOLT and the Fin Cone. AIAA-2019-3081 (2019)

- Mullen, D., Reed, H.: Linear Parabolized Stability Analysis of the BOLT Flight Geometry at Off-Nominal Conditions. AIAA-2020-3027 (2020)
- Mullen, D., Reed, H.: Analysis of the BOLT II – Holden Mission Flight and Wind Tunnel Geometries, AIAA-2021-0852 (2021)
- Mullen, D., Reed, H.: Analysis of the BOLT Flight Geometry at Off-Nominal Conditions, AIAA-2021-1093 (2021)
- Mullen, D., Reed, H.: Computational Modeling and Stability Analysis of BOLT Hypersonic Geometry Including Off-Nominal Conditions,” Theoretical and Computational Fluid Dynamics, August (2021).
- Reed, H.: Progress in Boundary-Layer Stability and Transition. Plenary Speaker, IUTAM Symposium on Laminar-Turbulent Transition, London, United Kingdom, 2-6 September (2019)