



Computational Support of 9x7 Wind Tunnel Test of Sonic Boom Models with Plumes

James C. Jensen⁺, Marie Denison⁺, Don Durston⁺, and
Susan E. Cliff⁺

AIAA-2017-0042

AMS Seminar– Ames Research Center– March 14, 2017

⁺NASA Ames Research Center, Moffett Field, CA

Outline



- Background
- Methodology
- Pre-Test CFD
- Post-Test CFD
 - Current Methods
 - Detailed Study
 - Recent Developments
- Conclusions

Background



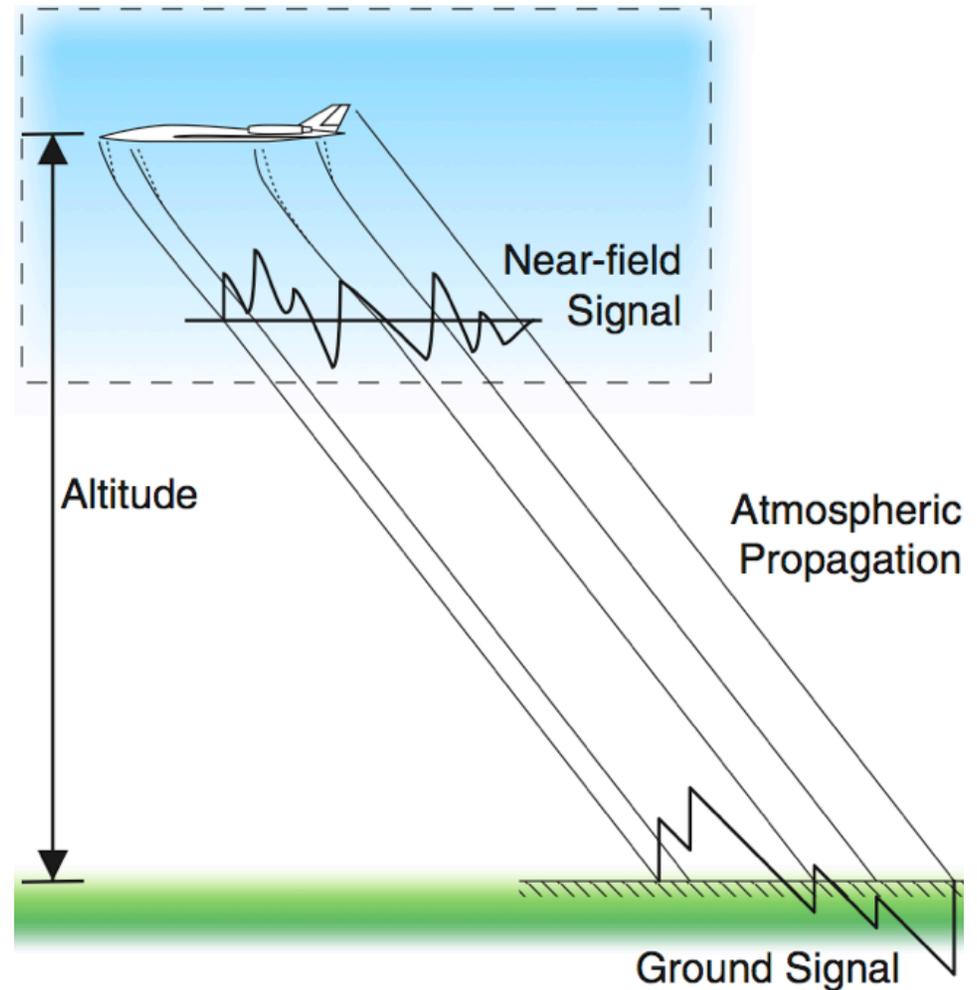
- NASA has been working with industry to develop technologies to enable civilian supersonic aircraft to fly over land
- An ultra low sonic boom aircraft is required to enable FAA certification of commercial supersonic transports
- Accurately predicting the noise produced by low boom aircraft is critical



Background

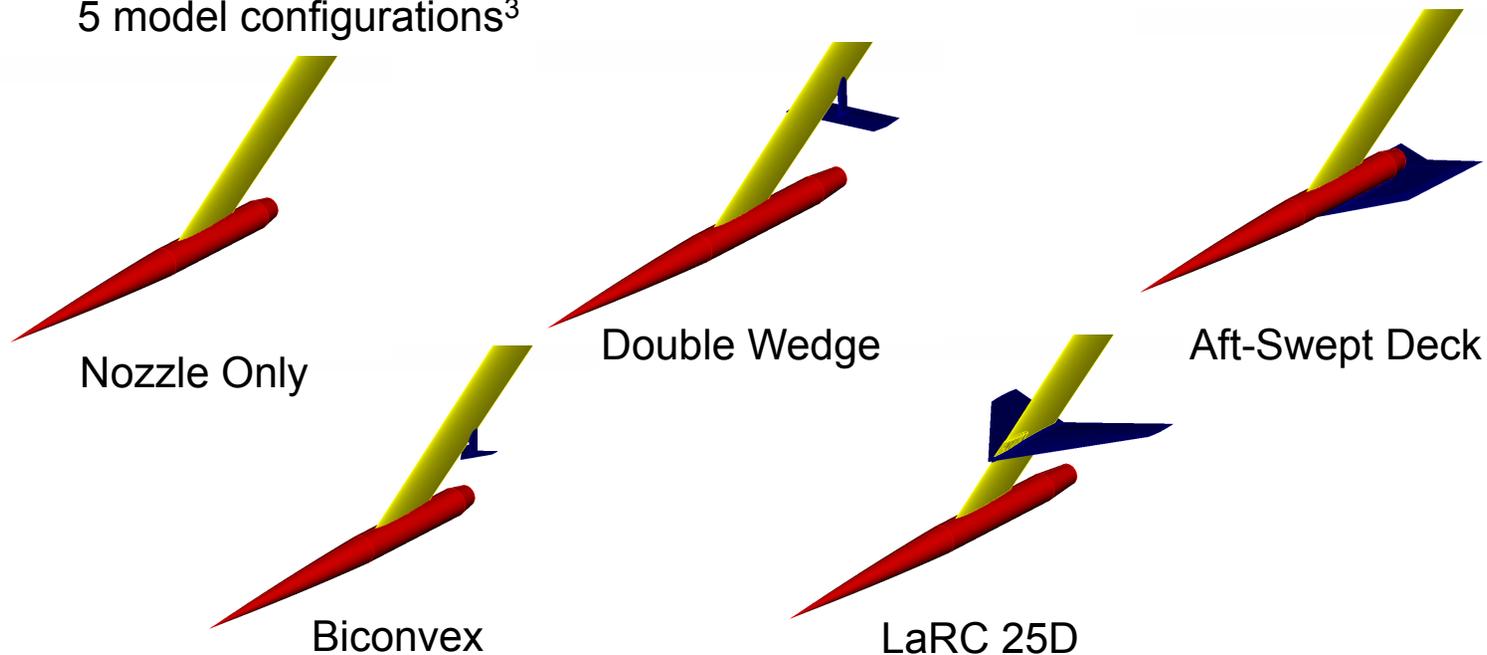


- Pressure signature prediction has two components
 - Using CFD to calculate the near-field pressure signature
 - Using atmospheric propagation codes to propagate the signal to the ground
- Aft signature prediction is a challenge due to the
 - Boundary layer effects
 - Interactions of the nozzle plume with shock waves produced by other surfaces of the geometry



Background

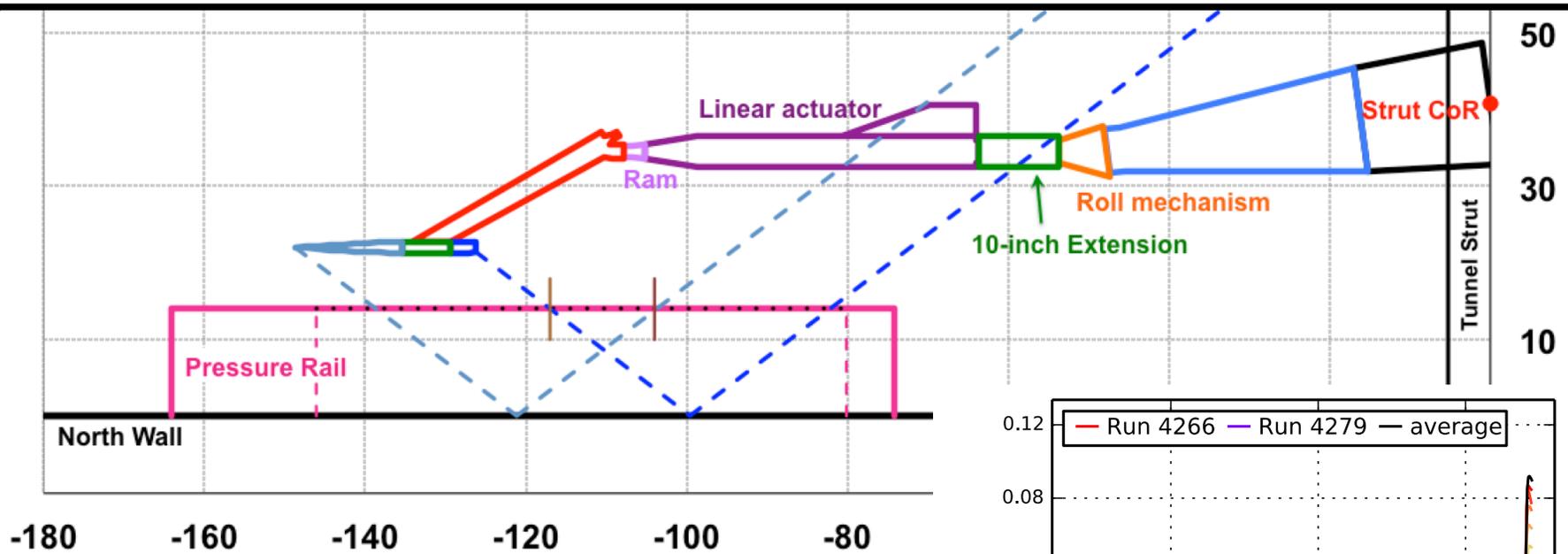
- A small-scale experiment was performed in the NASA Glenn 1x1 Foot Supersonic Tunnel to examine shock/plume interaction (2014)¹
 - Pressure signature taken at 1 nozzle diameter (1.0")
 - 1 model configuration
- A larger-scale shock/plume test was performed in the NASA Ames 9x7 Foot Supersonic Wind Tunnel that provided data for models that were representative of realistic aft geometries (2016)²
 - Pressure signatures up to 23 nozzle diameters (35") with a 1.5" nozzle diameter
 - 5 model configurations³



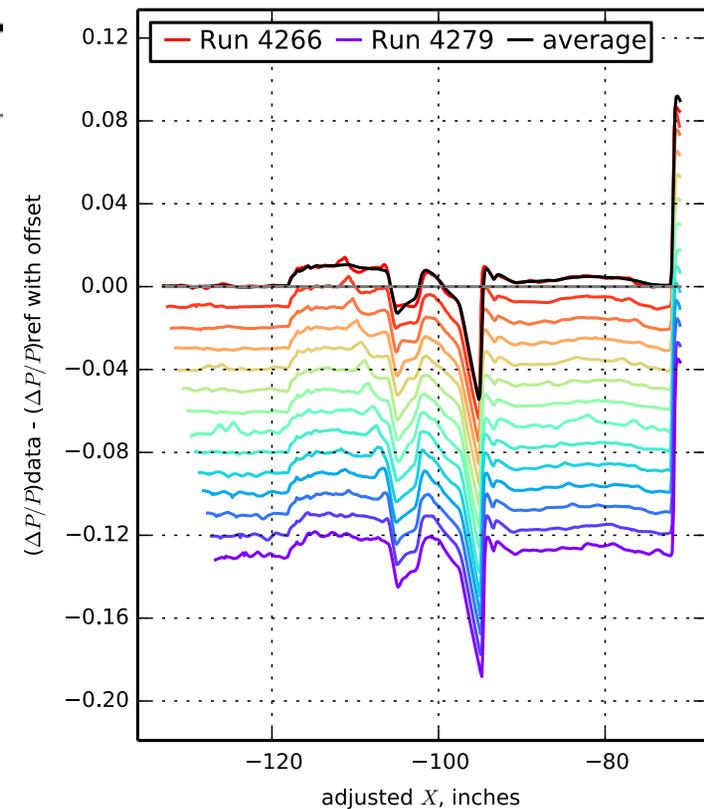
¹Castner et al, AIAA 2015-1046, ²Durston et al, AIAA 2017-0041, ³Cliff et al, AIAA 2016-2035



Background



- 160+ experimental runs
- Two Mach numbers
 - 1.6 and 2.0
- 4 Model altitudes
 - 8", 15", 23", and 35"
- 3 Roll angles
 - 0°, 15°, and 30°
- 2 Model angles of attack
 - 0° and 3°
- 4 Nozzle Pressure Ratios
 - (4, 8, 10, and 14)



Outline

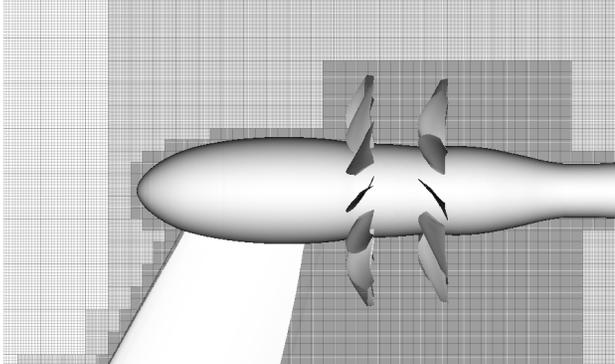
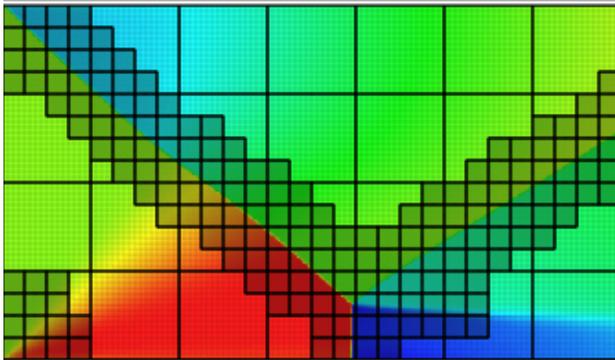


- Background
- **Methodology**
- Pre-Test CFD
- Post-Test CFD
 - Current Methods
 - Detailed Study
 - Recent Developments
- Conclusions

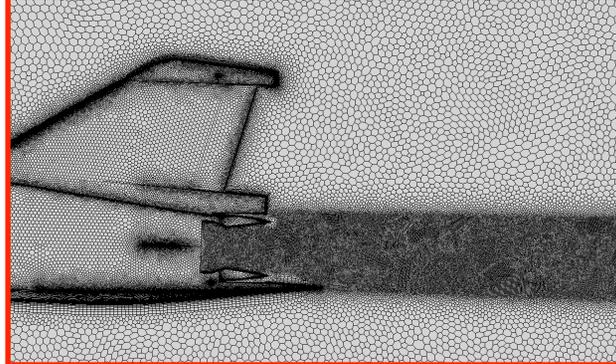
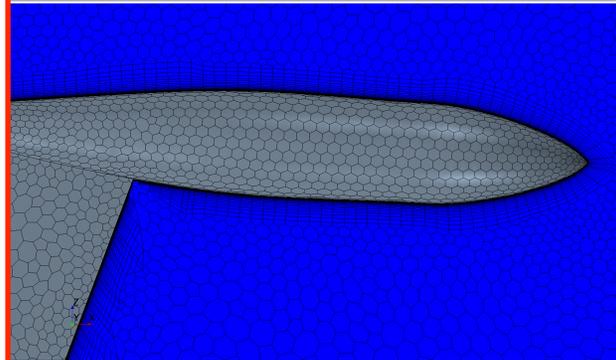
Methodology: Solver

Launch Ascent and Vehicle Aerodynamics (LAVA) framework⁴

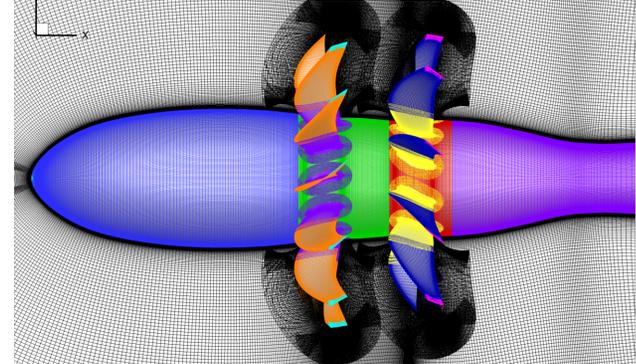
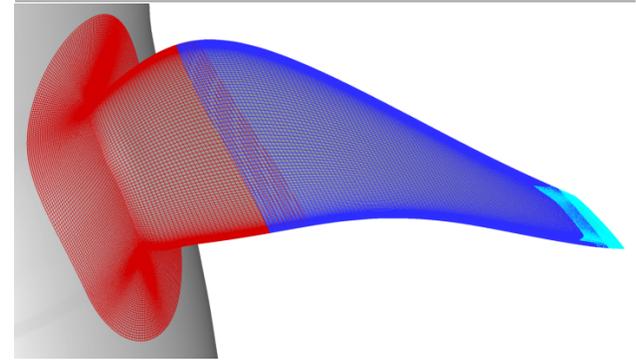
Structured Cartesian with AMR



Unstructured Arbitrary Polyhedral



Structured Curvilinear



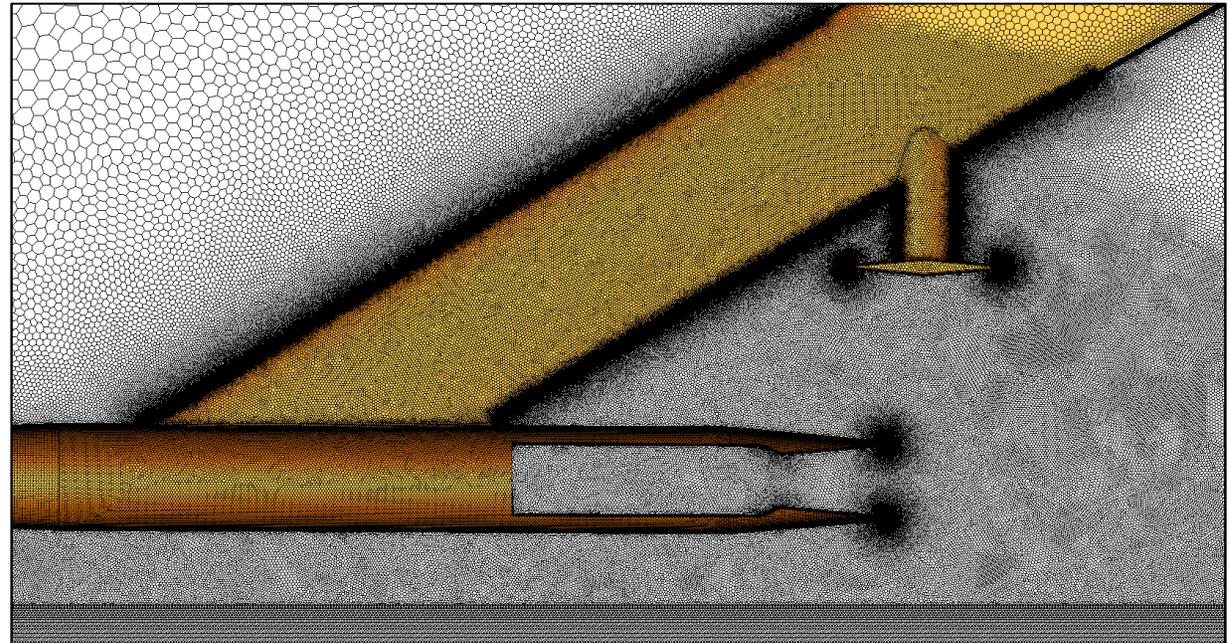
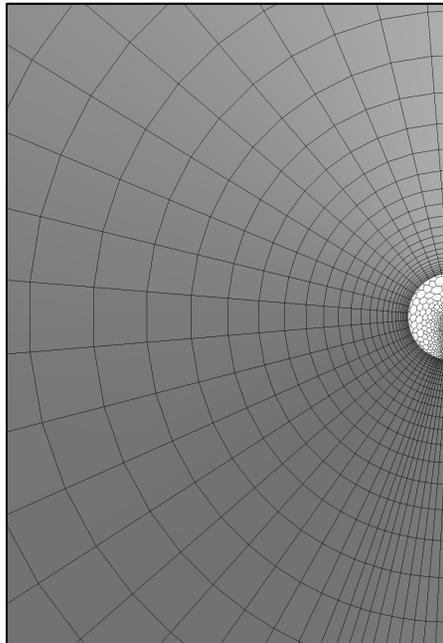
- Unstructured Polyhedral Solver (2nd Order)
 - Steady Reynolds-Averaged Navier-Stokes (RANS)
 - Spalart-Allmaras turbulence model

⁴Kiris et al, Aerospace Science and Technology Journal 2016, Computational Framework for Launch, Ascent, and Vehicle Aerodynamics

Methodology: Mesh Generation



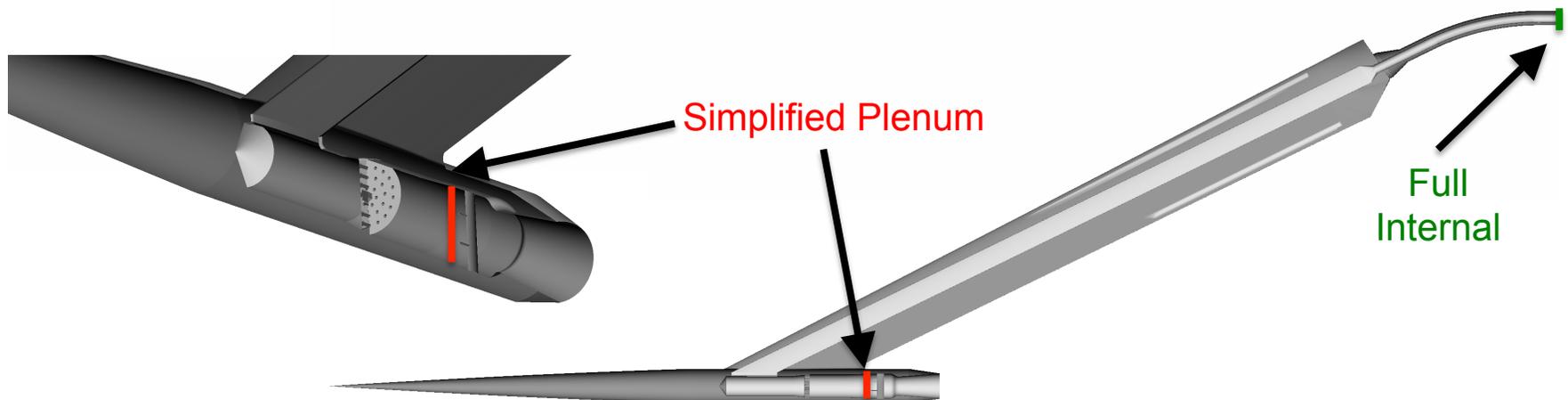
- The mesh was generated using a near body “core cylinder” mesh and Mach aligned extruded prismatic layers
- A Mach aligned refinement region exists between the model and the extruded layer boundary
- Angle of attack is treated by rotating the model relative to the free stream flow vector



Methodology: Boundary Condition



- Two types of boundary condition placements were used
 - Averaged total pressure and temperature surface at the nozzle plenum (**simplified plenum**)
 - Averaged total pressure and temperature surface at the top of the strut (**full internal flow**)
- Simplified plenum BC was used for the pre-test external design study and for the current methods post-test CFD
- Internal flow BC was used for the pre-test internal flow path design study and for detailed post-test CFD



Outline



- Background
- Methodology
- **Pre-Test CFD**
- Post-Test CFD
 - Current Methods
 - Detailed Study
 - Recent Developments
- Conclusions

Pre-Test CFD Objectives

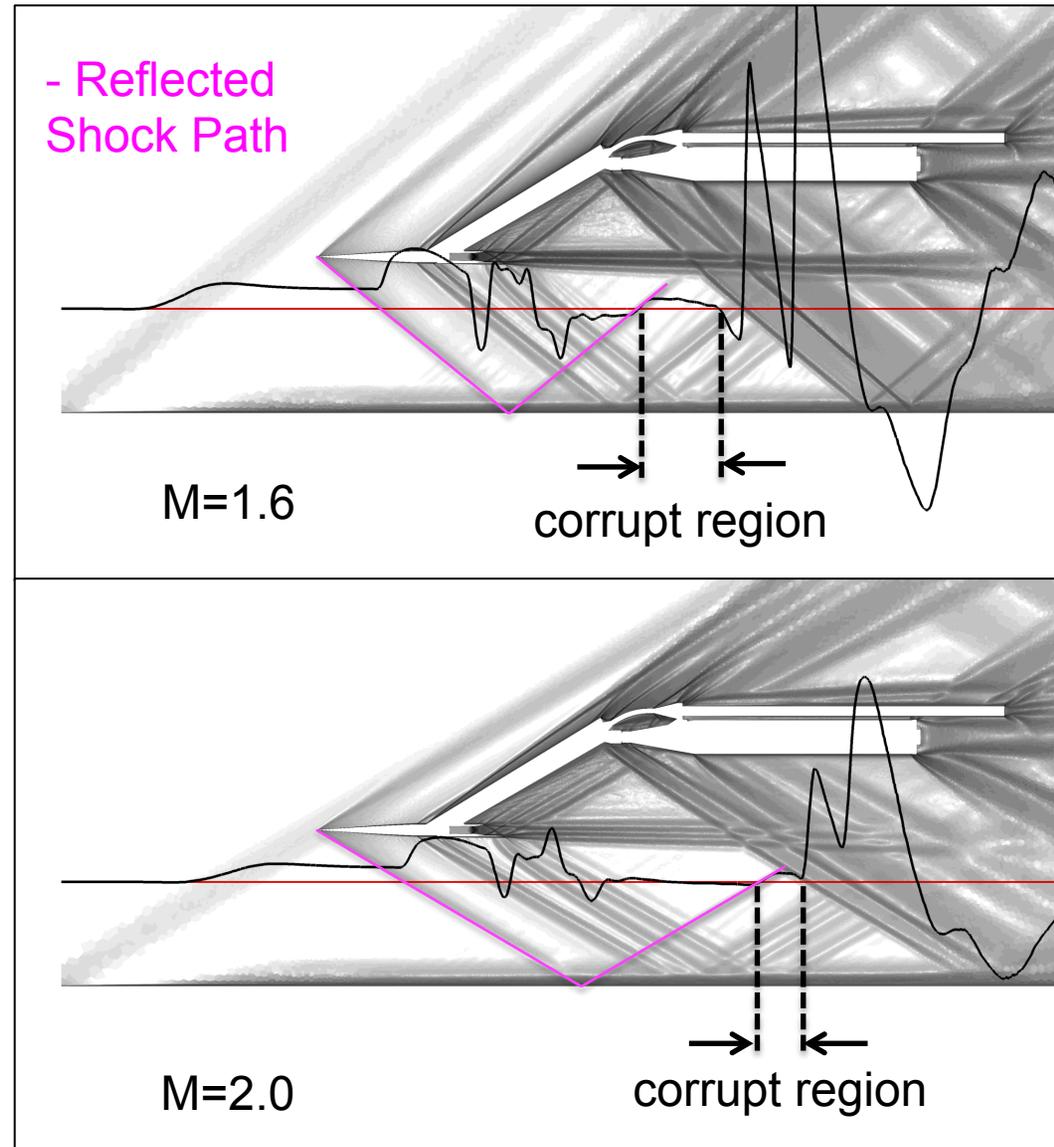


- A goal of the experiment was to provide a large database of validation data that can be used to compare to CFD
- Use CFD simulations to improve the quality of data that would be gathered during the test
 - Examine the effects of the model nose shock reflecting off the walls of the tunnel and contaminating the plume data
 - Examine the internal flow of the test rig to ensure the experiment had a well behaved plume

Nose Shock Reflection from Wall

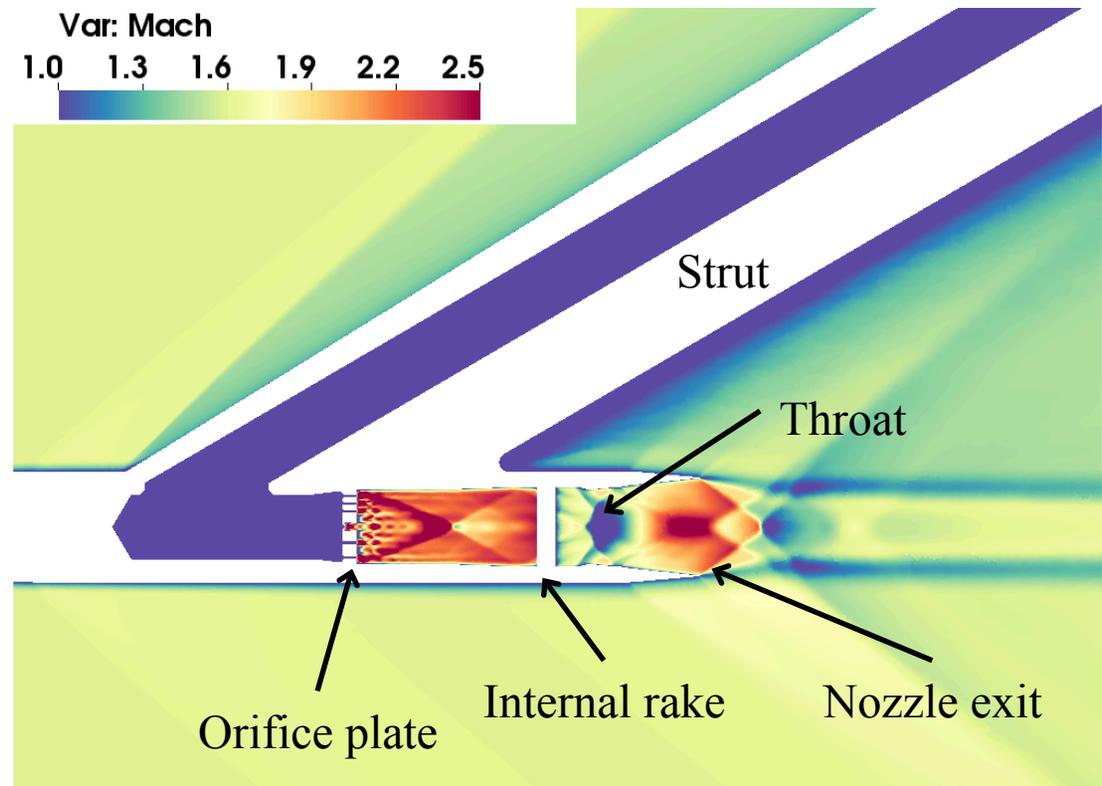


- M=1.6 signature is compromised by nose shock
- M=2.0 shifts reflected shock aft, providing greater extent of contamination-free plume
- Results lead to an increase in the number of M=2.0 runs during the experiment



Internal Flow Issues

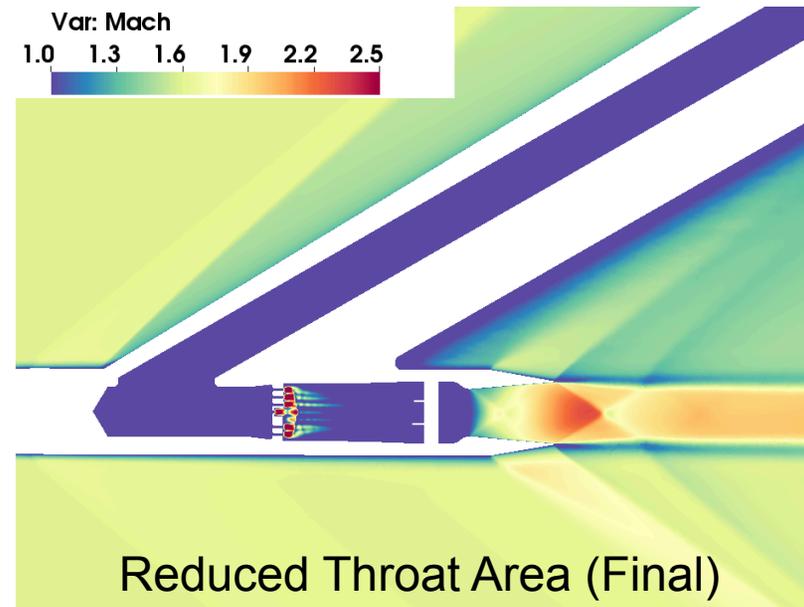
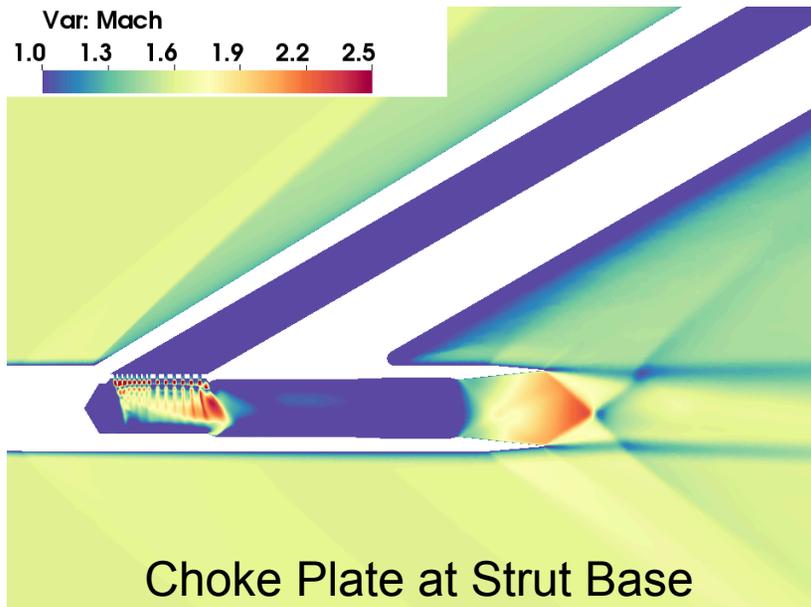
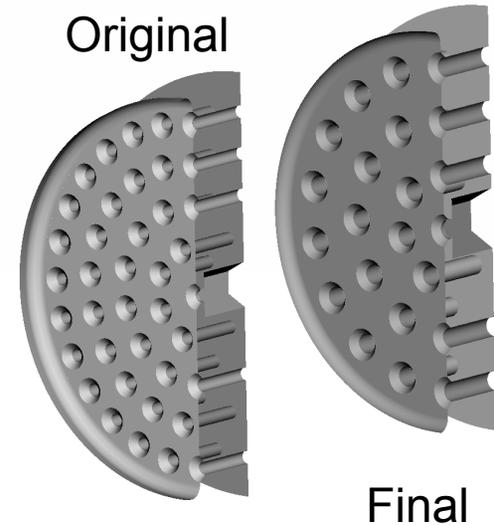
- Original nozzle design was found to have supersonic flow between the orifice plate and the internal rake, additionally, the nozzle throat was not choked
- This would lead to poor quality measurements due to:
 1. Needing to use a shock correction to the data measured at the internal rake, which was used to determine the nozzle pressure ratio (NPR)
 2. Extraneous shocks would propagate from the nozzle into the plume
 3. Lack of choking at the throat would lead to a less uniform plume that would be more sensitive to the flow conditions



Internal Flow Path Design



- The CFD study involved over 30+ RANS runs with different orifice plate designs (including up to 4 staggered plates), nozzle profiles and flow conditions
- CFD revealed the need to make changes to the orifice plate itself
- Two designs met the desired requirements:
 - Placing the choke plate at the nozzle and strut intersection
 - Proportionally reducing the area of the nozzle exit and throat.



Pre-Test CFD Summary



- The LAVA unstructured solver was successfully used to help improve the experiment:
 - The simulations of wind tunnel wall effects showed that it was necessary to have more test runs than originally planned at the $M=2.0$ condition
 - The internal flow simulations were used to help redesign the nozzle to realize a cleaner plume and subsonic flow in front of the internal Pitot probe

Outline



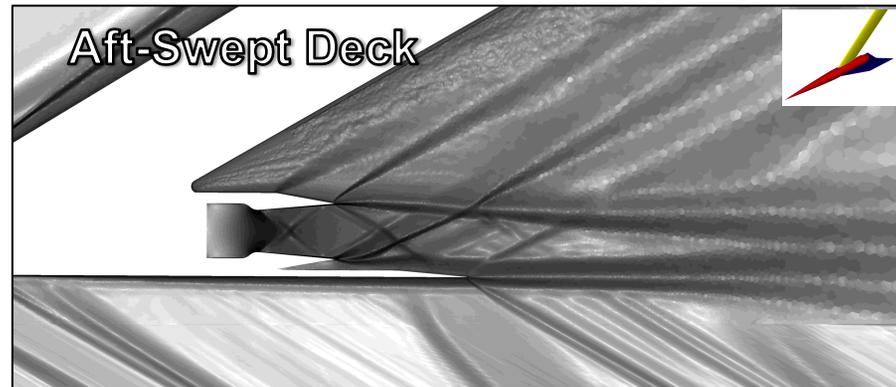
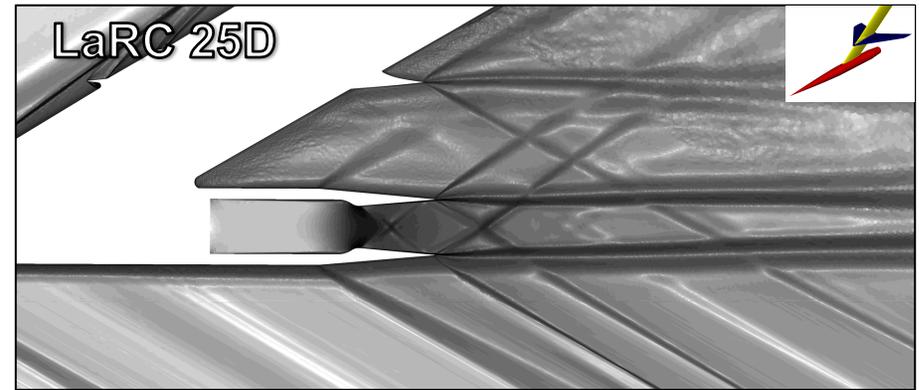
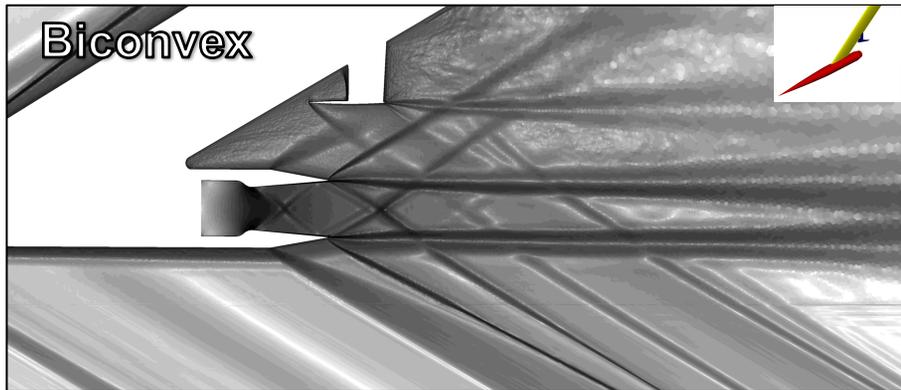
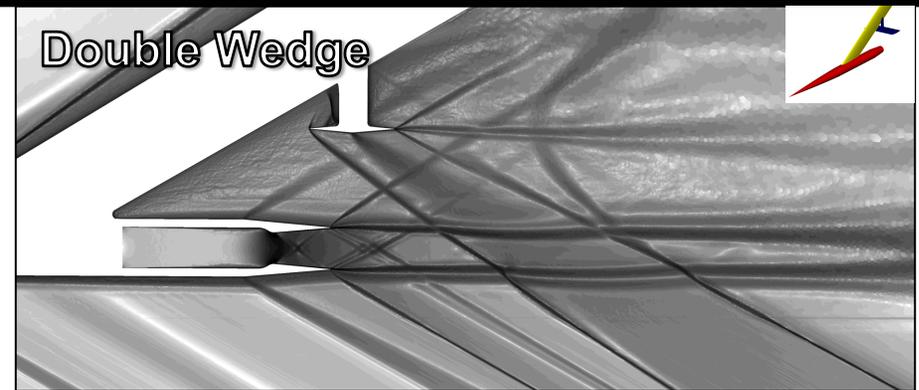
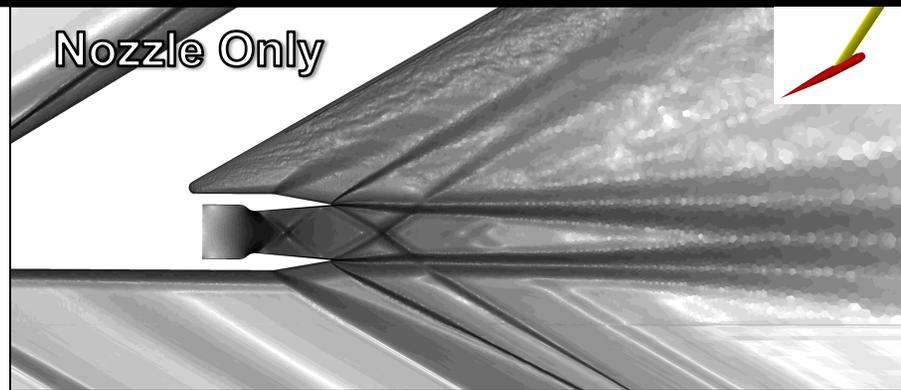
-
- Background
 - Methodology
 - Pre-Test CFD
 - Post-Test CFD
 - Current Methods
 - Detailed Study
 - Recent Developments
 - Conclusions

Post-Test CFD Current Methods Objectives



- Compare CFD results generated with a simplified plenum boundary condition to the wind tunnel test data and assess their validity
 - All test configurations
 - Mach number effect
 - Angle of attack effect
 - Model altitude effect
 - Off-track effect
 - Nozzle pressure ratio effect

Model Density Gradient Comparison

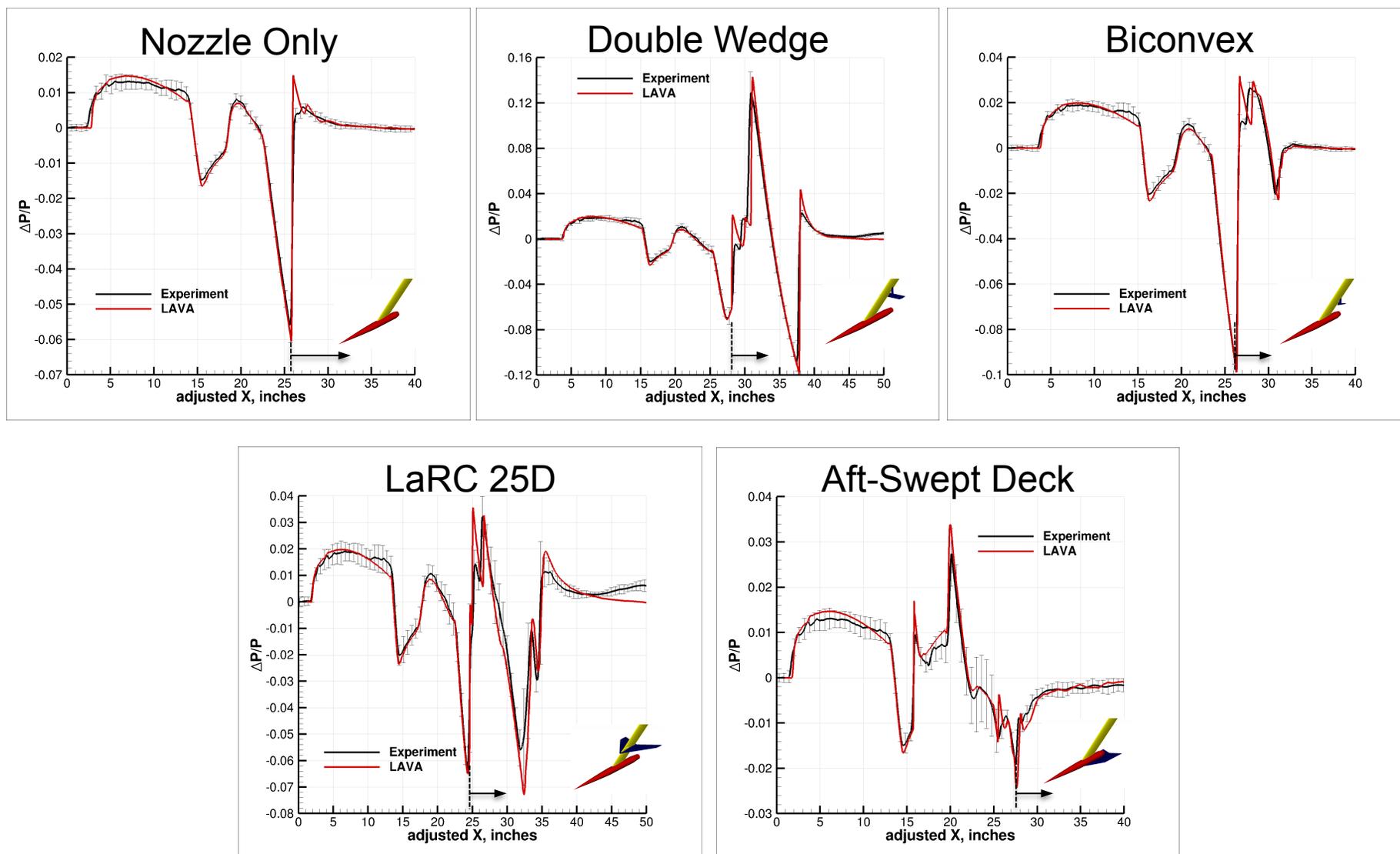


M=2.0 and NPR=8
(Double Wedge
NPR=10)



Model Signature Comparison

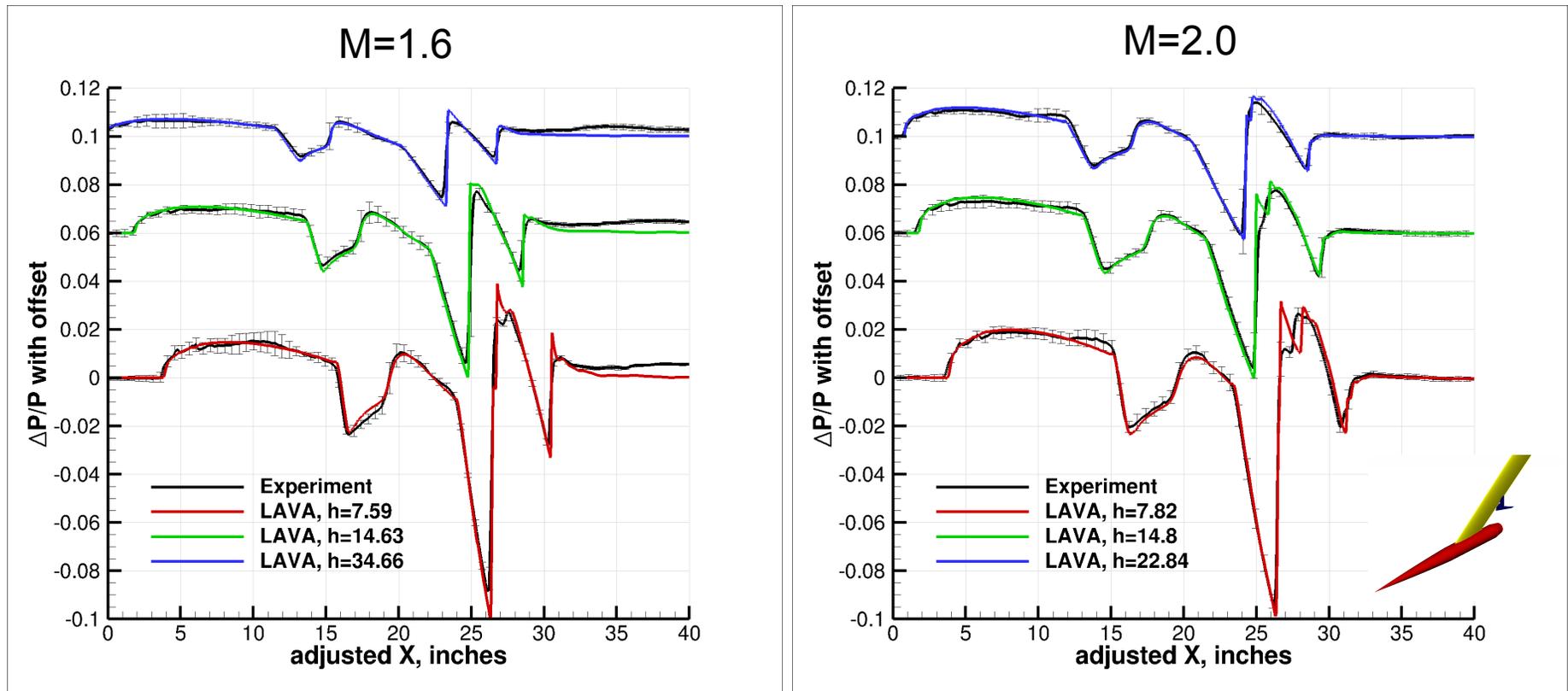
M=2.0, H=8" (Aft-Swept Deck H=15"), and NPR=8 (Double Wedge NPR=10)



Mach Number Comparison



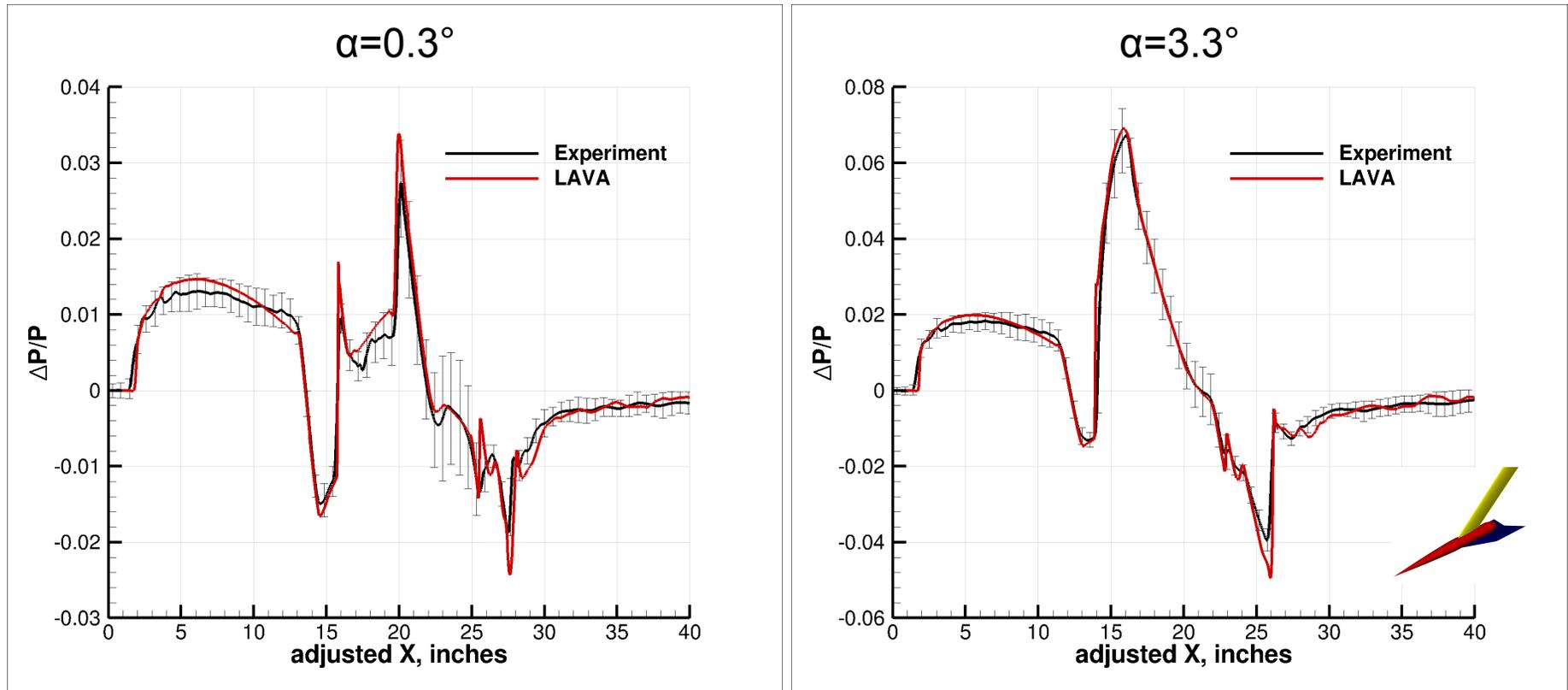
Biconvex Configuration at NPR=8



- The Mach number does not have a negative impact on the comparison between the experiment and the CFD

Angle of Attack Comparison

Aft-Swept Deck at $M=2.0$, $H=15''$, and $NPR=8$

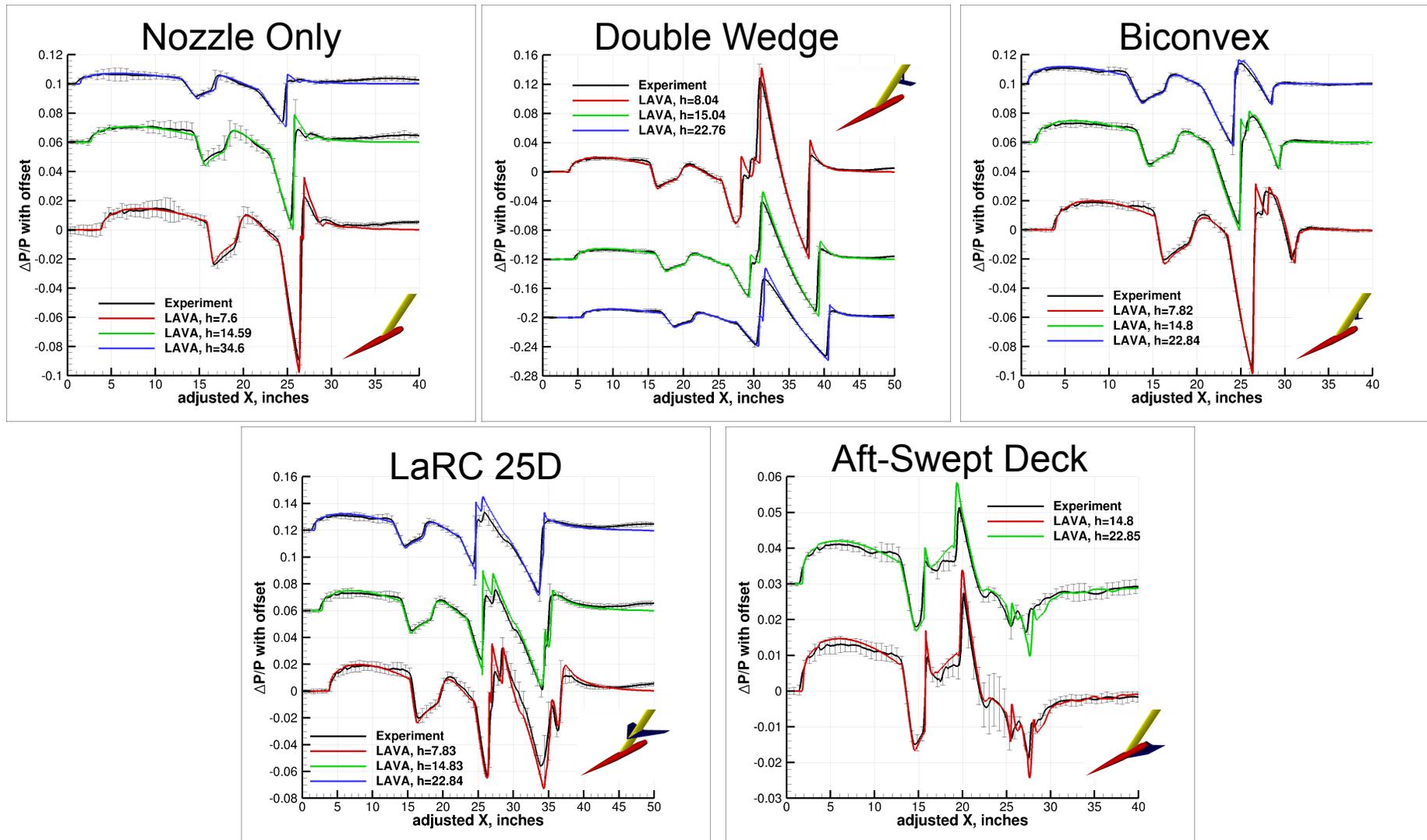


- Changing the angle of attack does not negatively affect the comparison between the experiment and the CFD

Altitude Comparisons

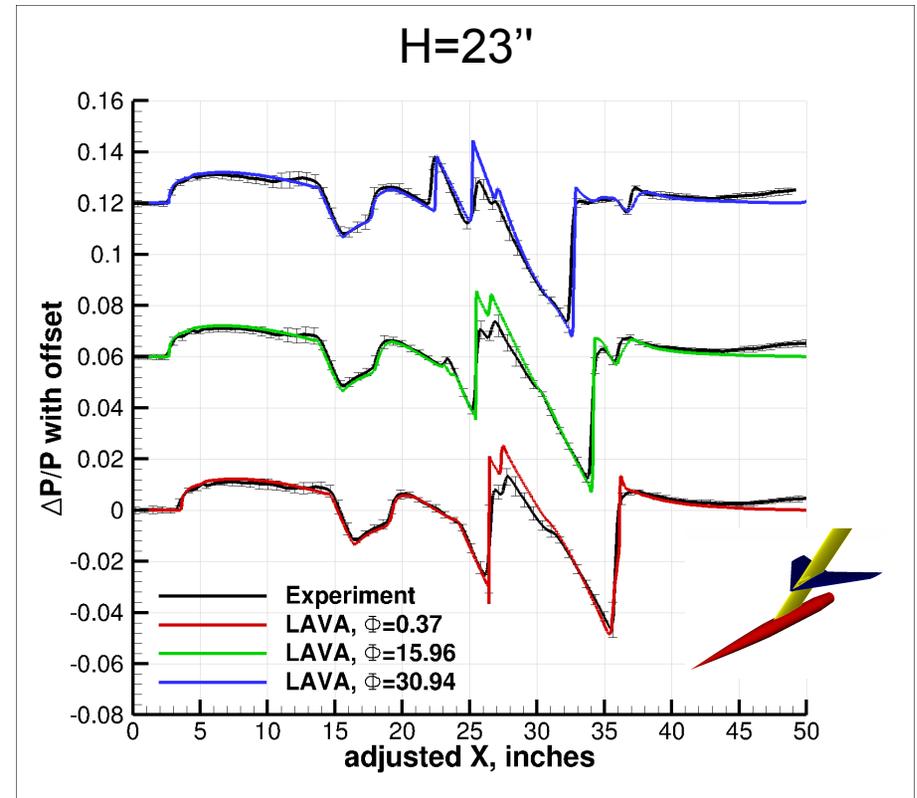
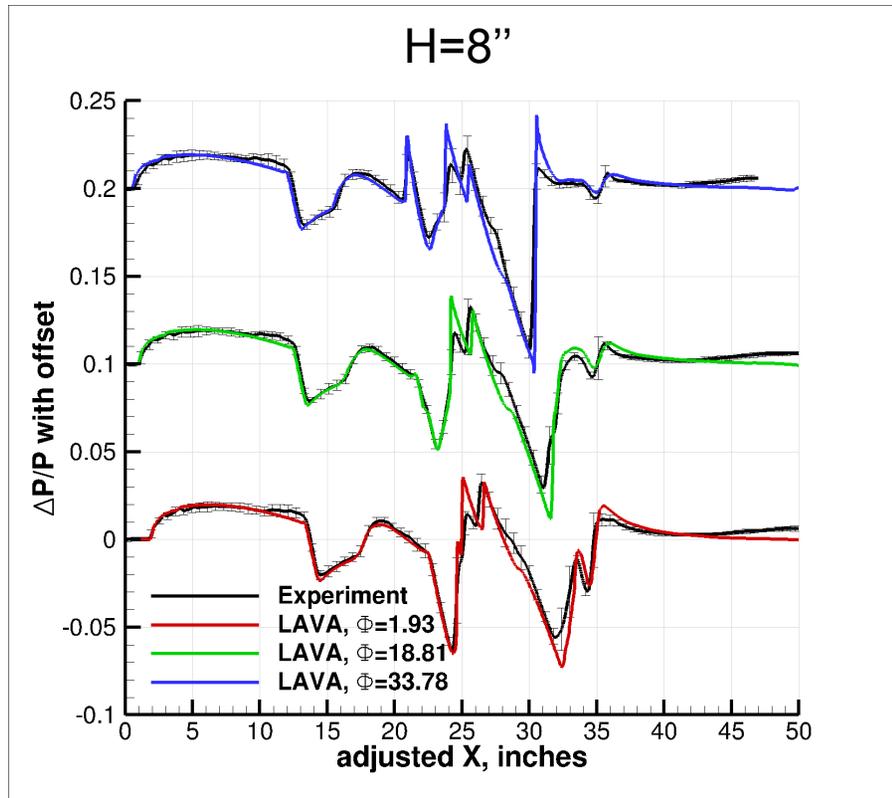


M=2.0 and NPR=8 (Double Wedge NPR=10)



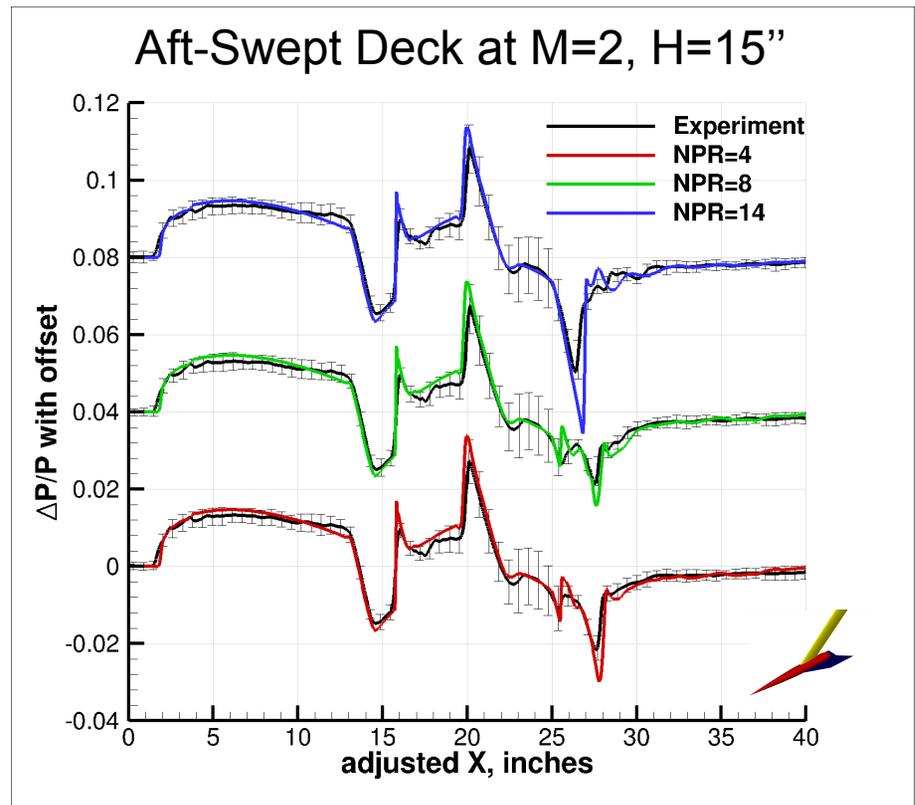
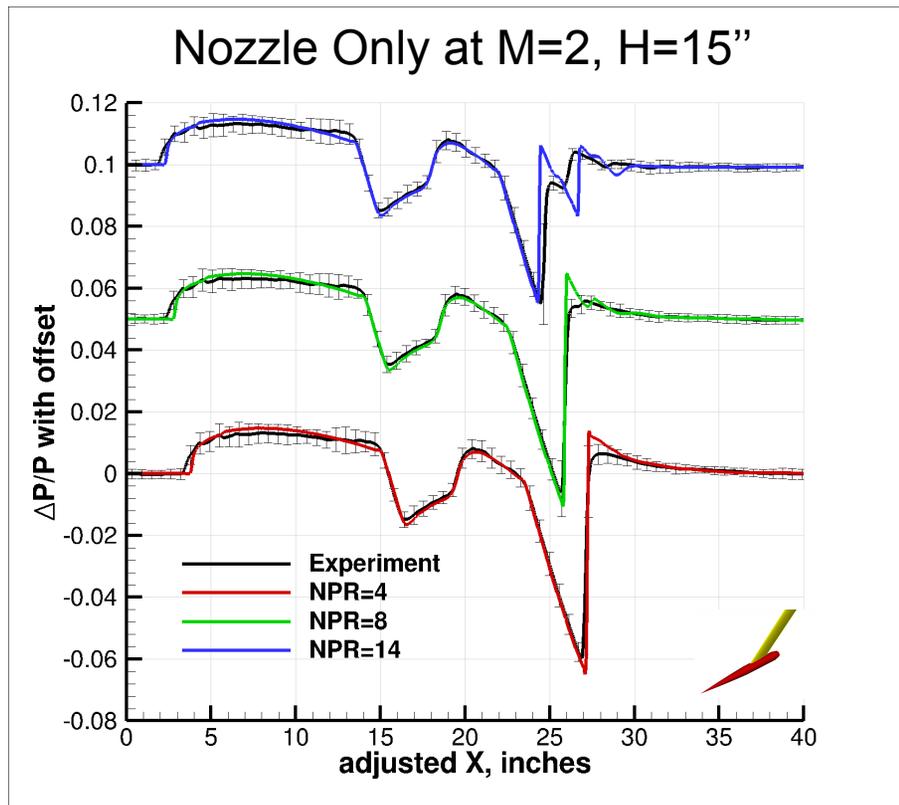
Off-Track Comparisons

LaRC 25D at M=2.0 and NPR=8



- The differences between the experimental and CFD signatures are not worsened by increasing the off-track angle

NPR Comparisons



- As the NPR is increased the differences between the CFD and experimental signatures also increase
- This seems to be caused by the use of the simplified plenum boundary conditions
- NPR=4 and 8 still match the experiment well but NPR=14 does not

Post-Test CFD Current Methods Summary



- The results generated using the plenum boundary condition compared well to the experimental data
 - Changing the conditions of the simulations (Mach and angle of attack) as well as the location of where the data was extracted (model altitude and off-track angle) did not negatively affect the comparison between CFD and experiment

- The higher the NPR the worse the comparison between the CFD and experiment

Outline

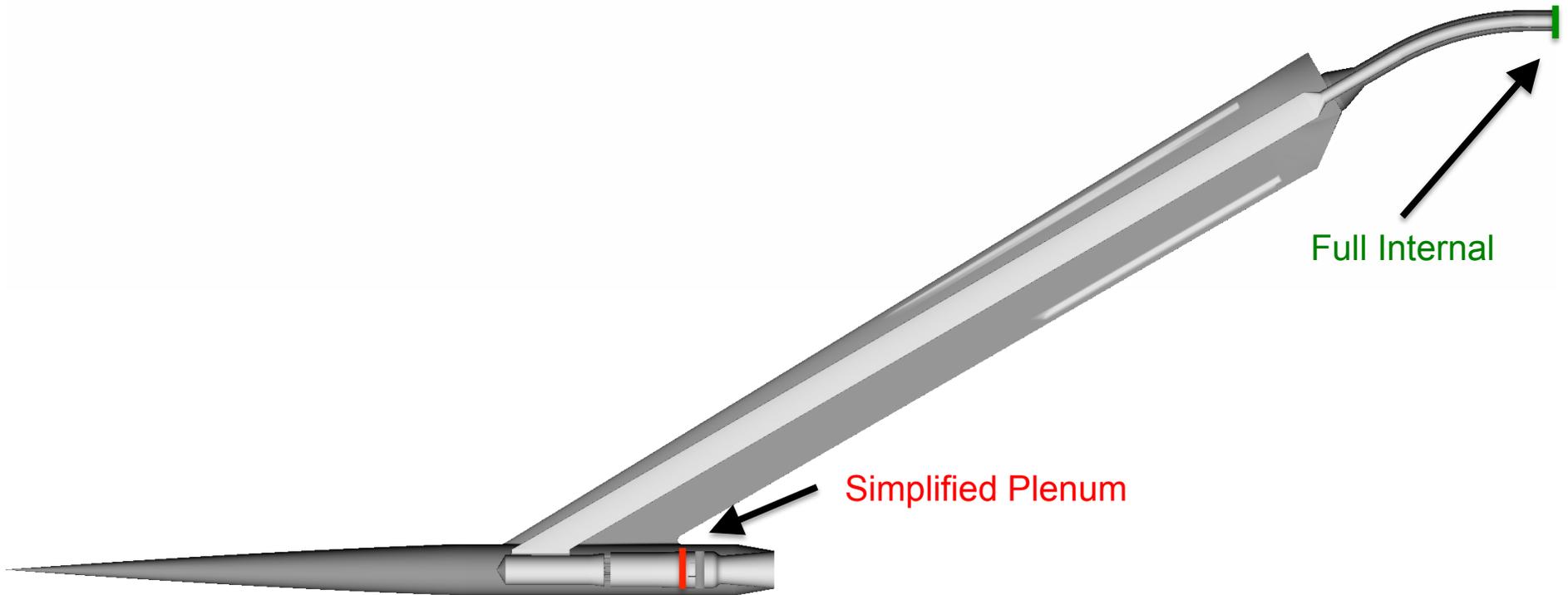


-
- Background
 - Methodology
 - Pre-Test CFD
 - Post-Test CFD
 - Current Methods
 - **Detailed Study**
 - Recent Developments
 - Conclusions

Detailed CFD Study with Internal Flow Path



- Compare the simplified plenum and full internal CFD results with the experiment to assess their validity

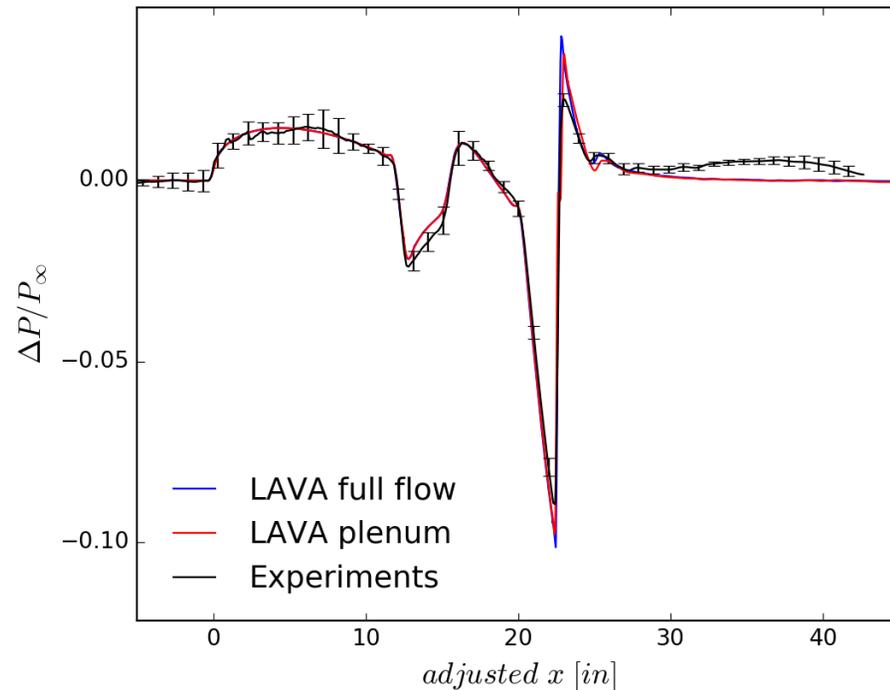


Pressure Signature Comparisons

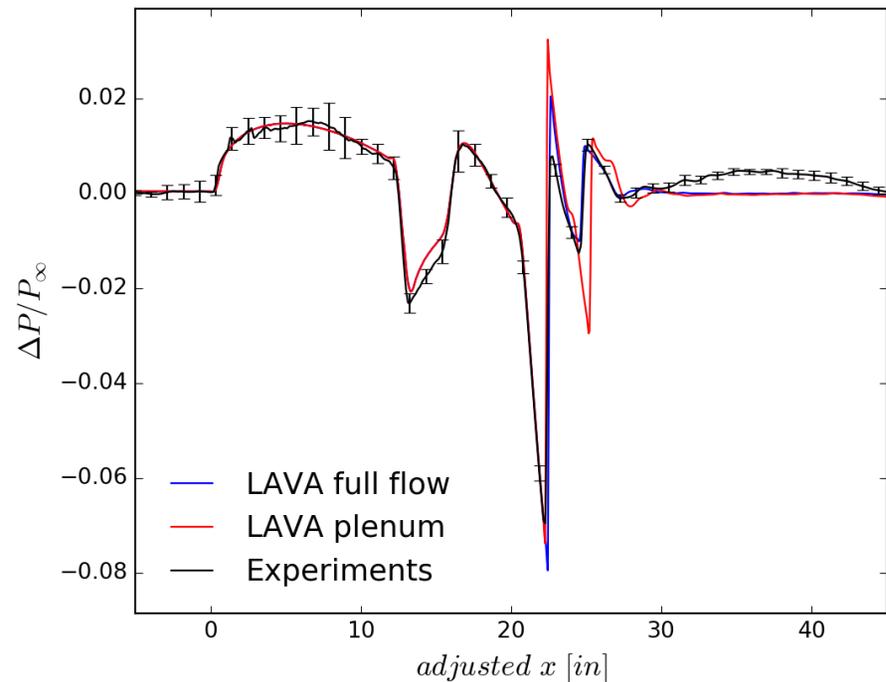


Nozzle Only at M=1.6 and H=15''

NPR=8



NPR=14

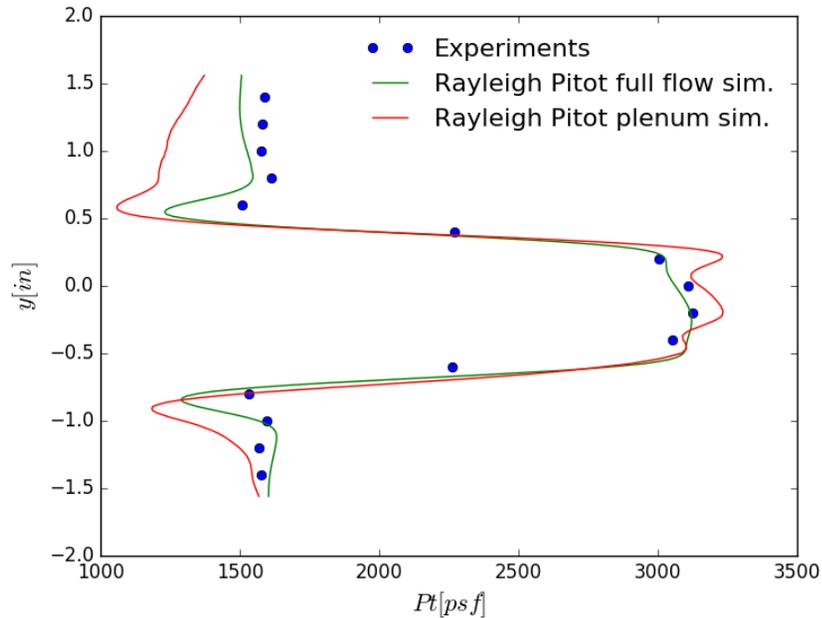


- At NPR=8 the full flow, plenum, and experiment are all in close agreement but at NPR=14 the plenum boundary condition signature has significant differences as compared to the experiment

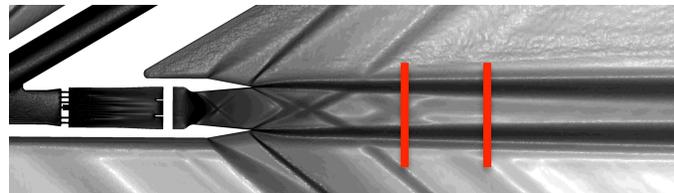
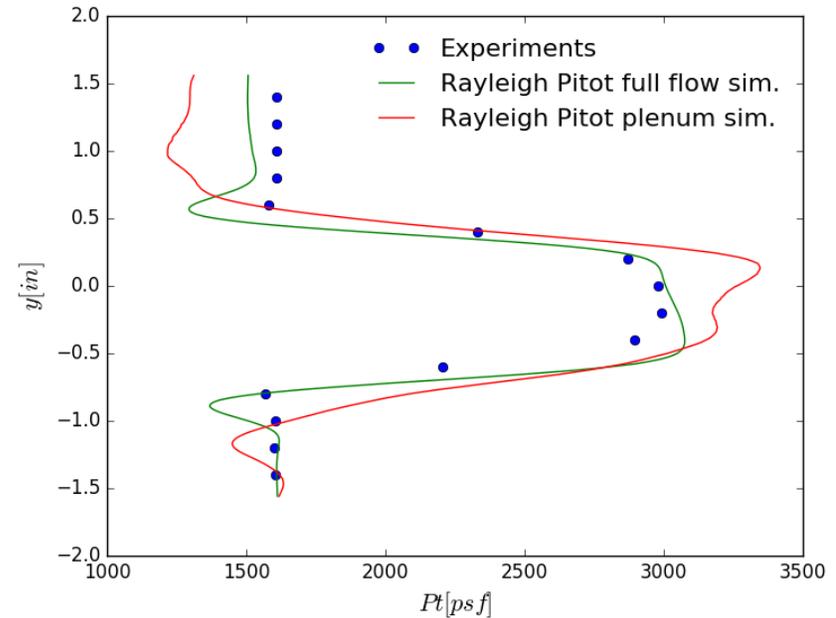
Plume Rake Comparisons (M=1.6, NPR=14)



NPR=14, Distance to nozzle exit = 3.2 in



NPR=14, Distance to nozzle exit = 5.1 in

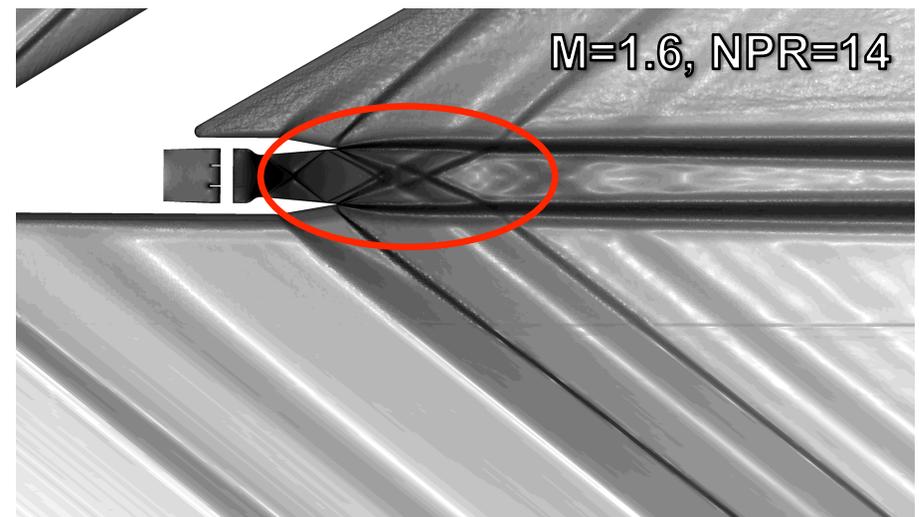
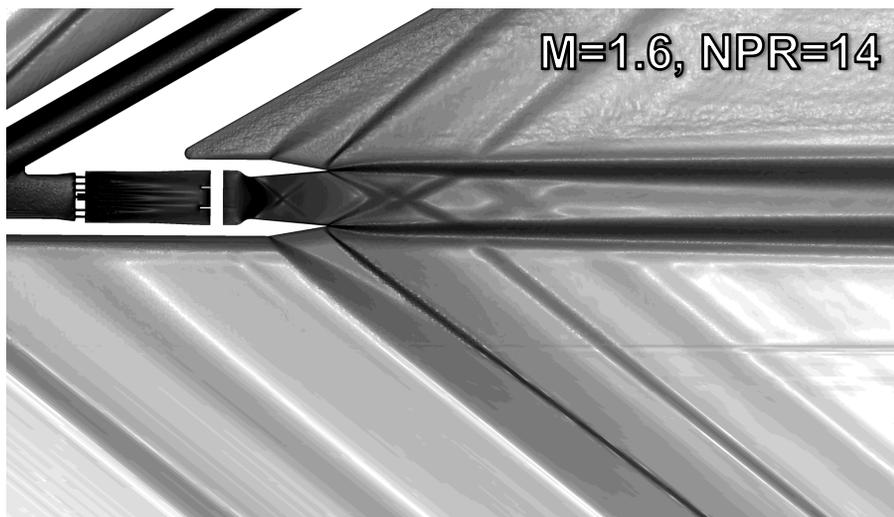
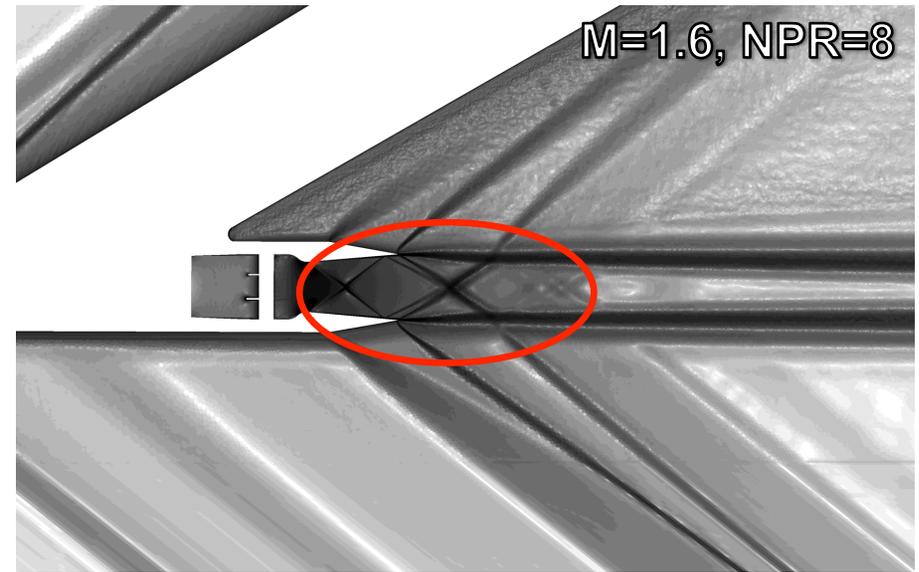
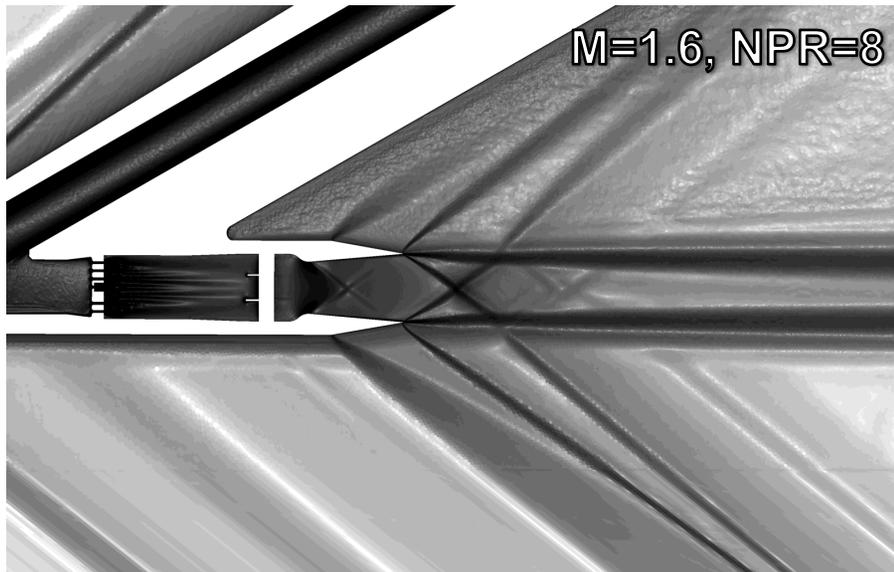


- The probe data are approximated from the simulations by applying the Rayleigh Pitot tube normal shock correction at the location of the probes
- Included corrections for the drooping and AOA of the rake

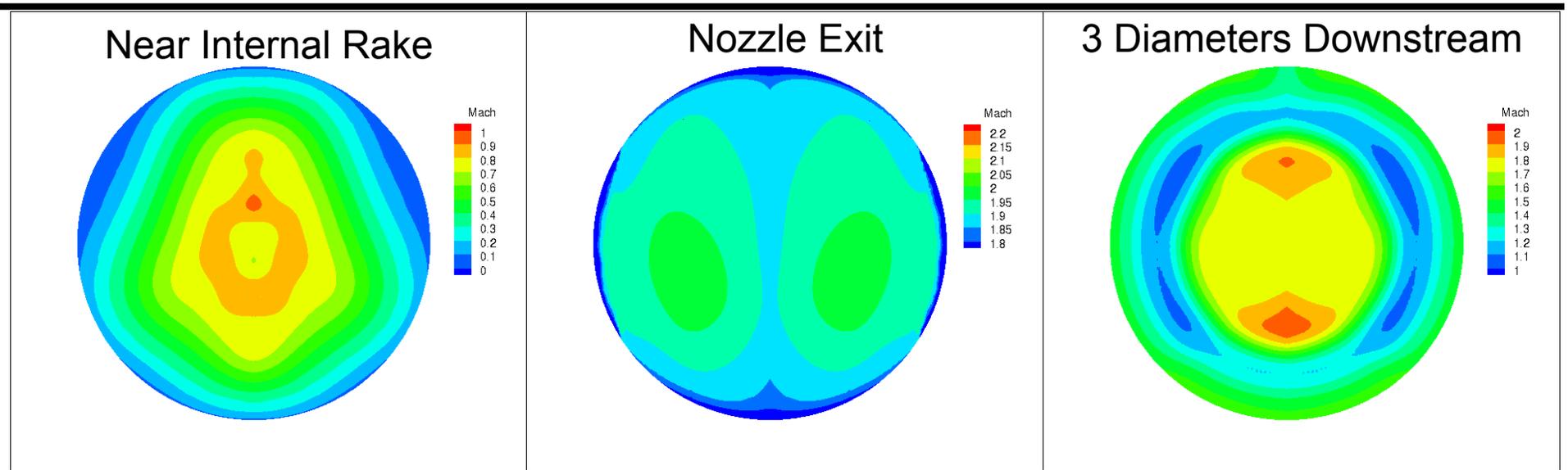
Density Gradient Comparison (Nozzle Only)

Full Internal Flow

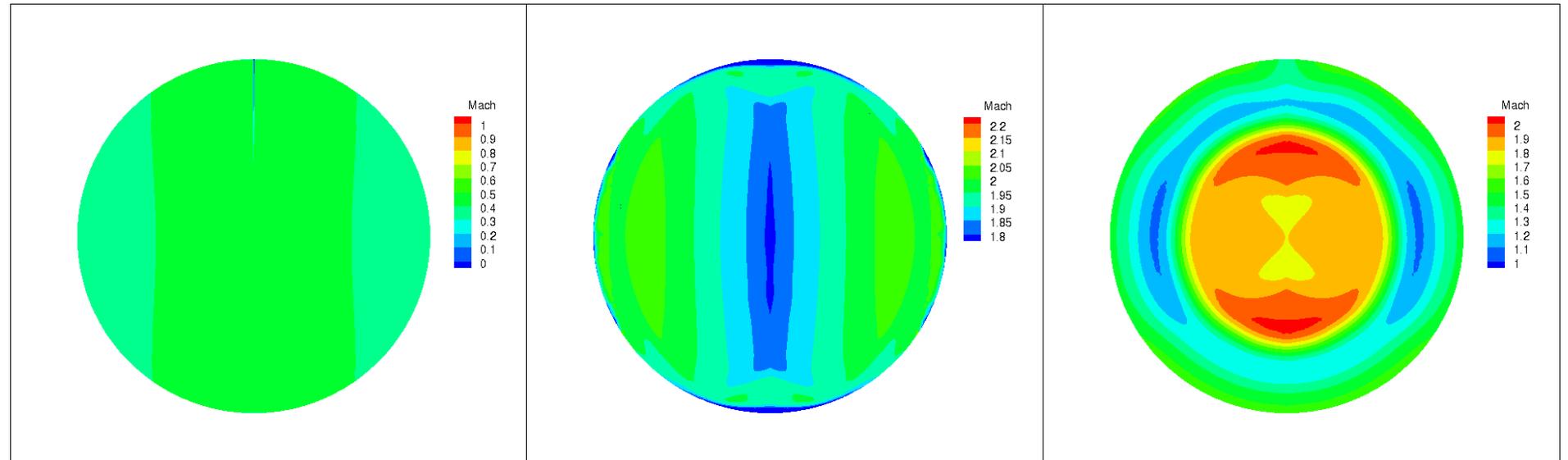
Plenum



Internal Mach Contour Comparisons (Nozzle and NPR=8)



Mach Profiles for Full Internal Flow

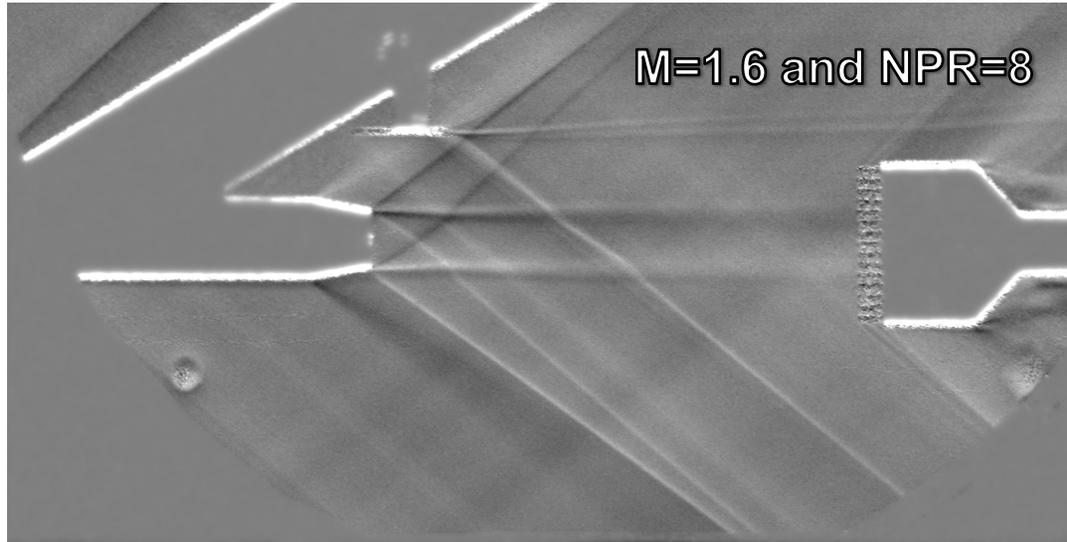


Mach Profiles for Simplified Plenum

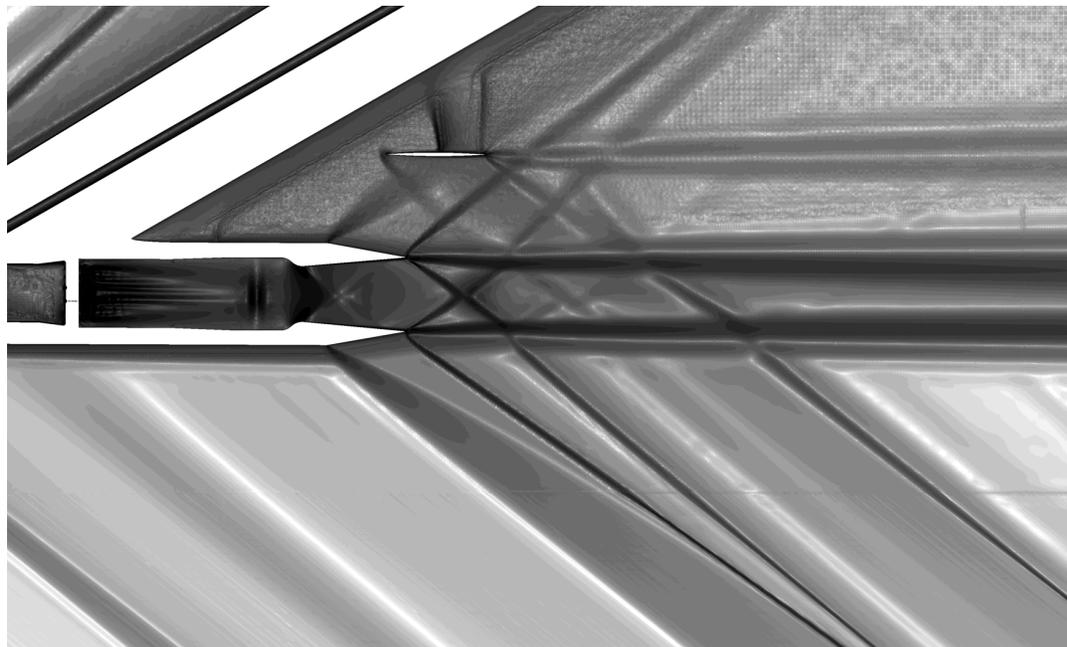
Flow Visualization Comparison (Biconvex)



RBOS



Density
Gradient Slice
From CFD



Detailed CFD Summary and Conclusions



- Full flow simulations show significant differences in the plume region relative to the simplified plenum simulations
- Using the full internal flow generated a better match with the experiment for all of the NPR values

Outline

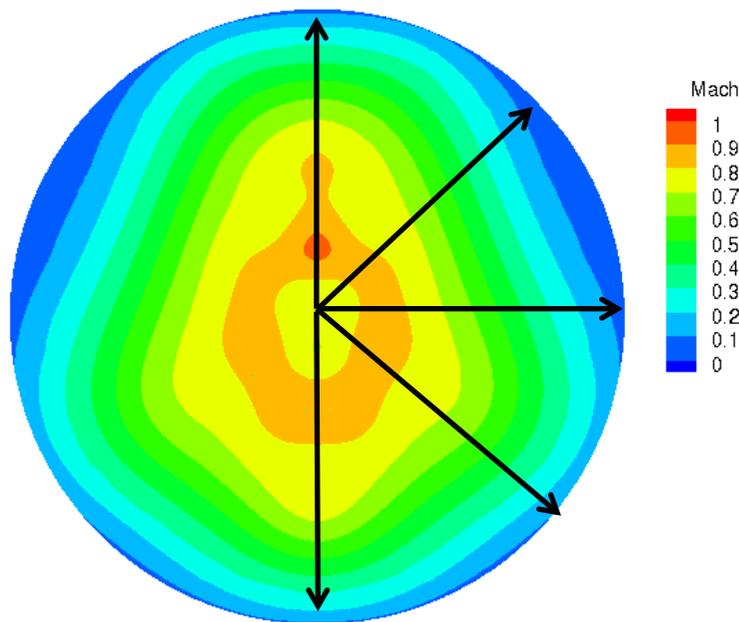


- Background
- Methodology
- Pre-Test CFD
- Post-Test CFD
 - Current Methods
 - Detailed Study
 - **Recent Developments**
- Conclusions

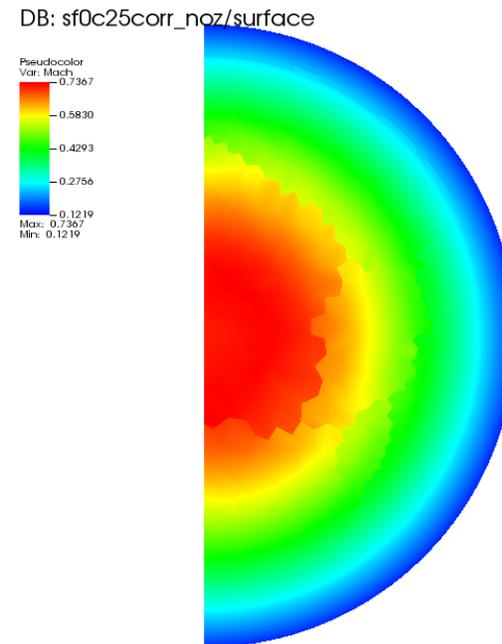
Radial Boundary Condition Implementation



- Extract cut lines from the full flow path solution at the desired location of the inlet BC (e.g., 1 ray at every 45°) and then average them to create the averaged radial profile



Full Flow Solution With Extraction Lines

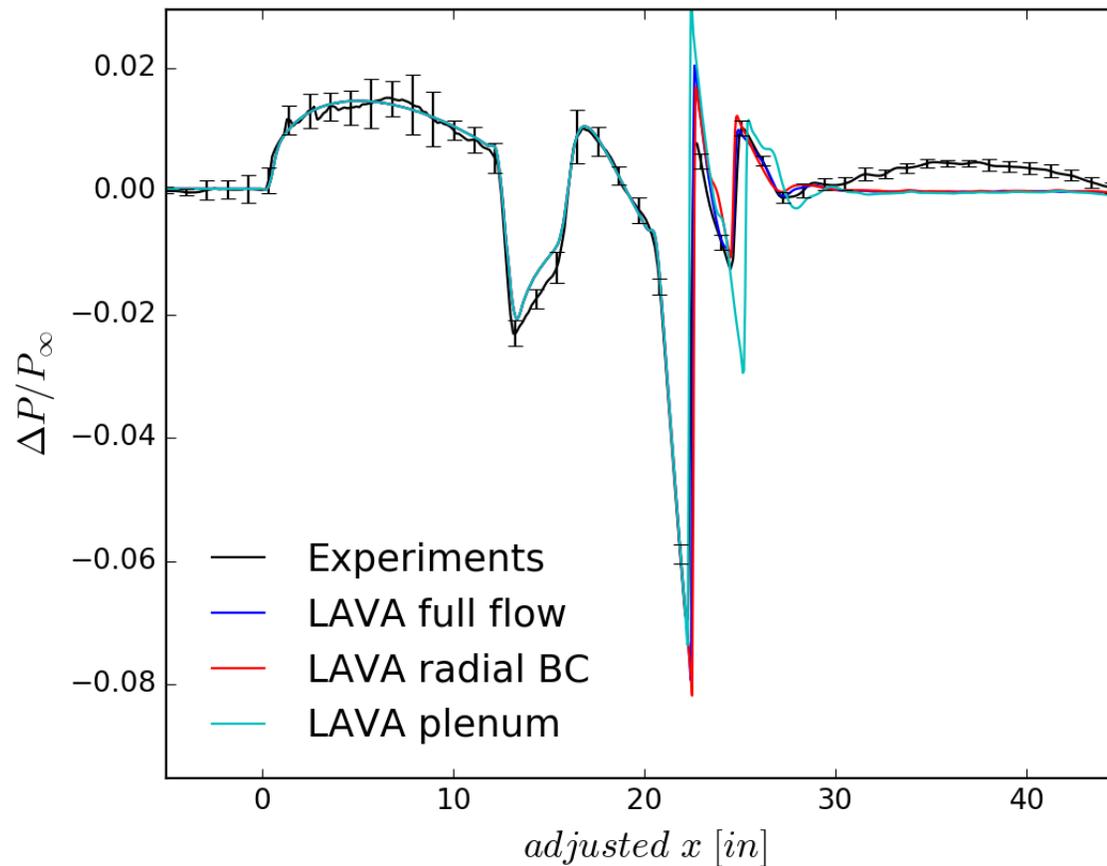


Resulting Radially Varying Boundary Surface

Radial BC Signature Comparison (nozzle only)

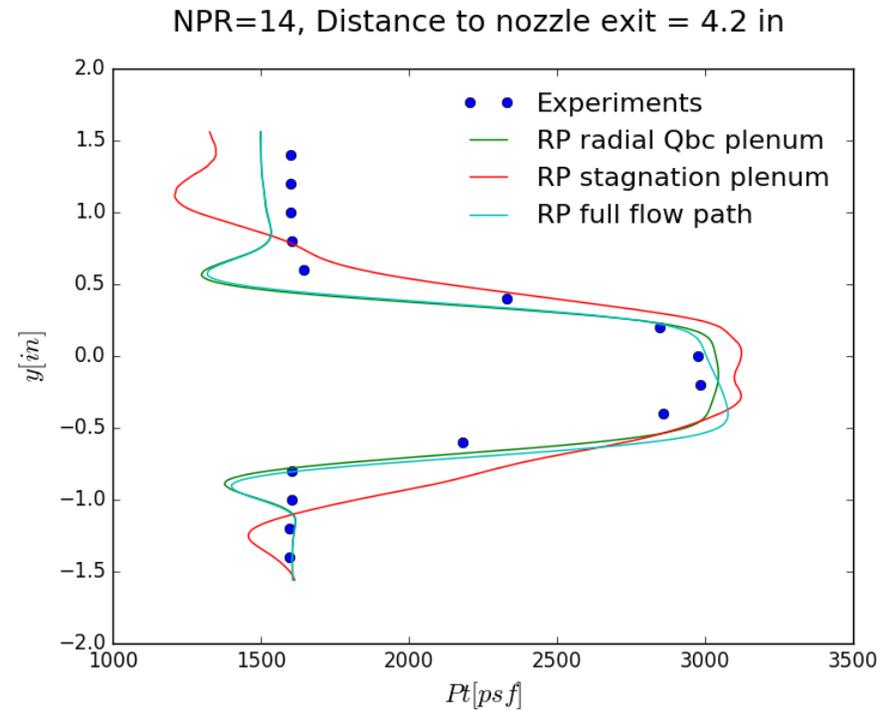
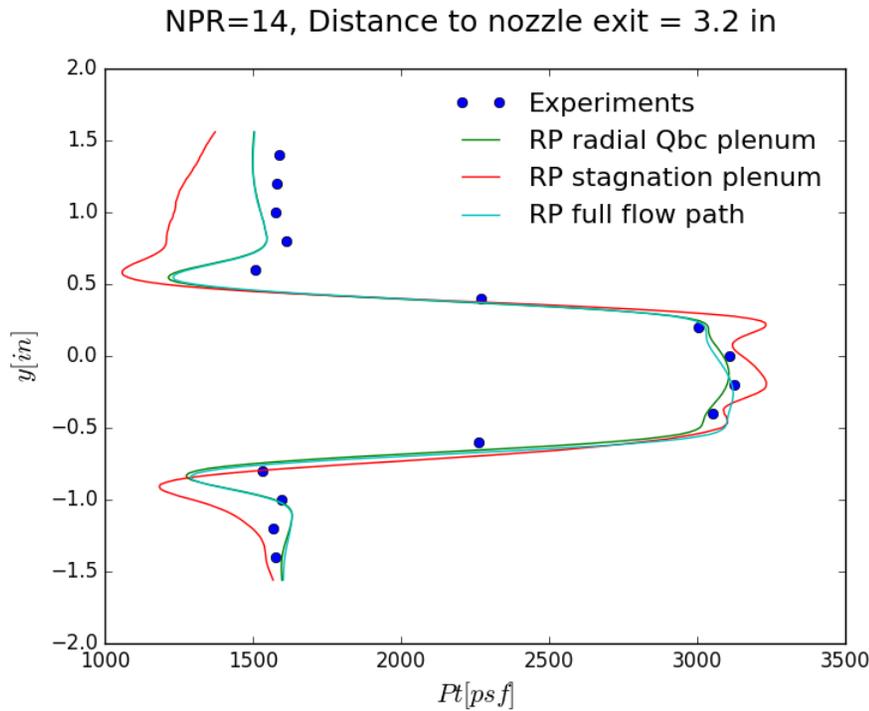


M=1.6, NPR=14, h=7.5951, roll=4.5718°



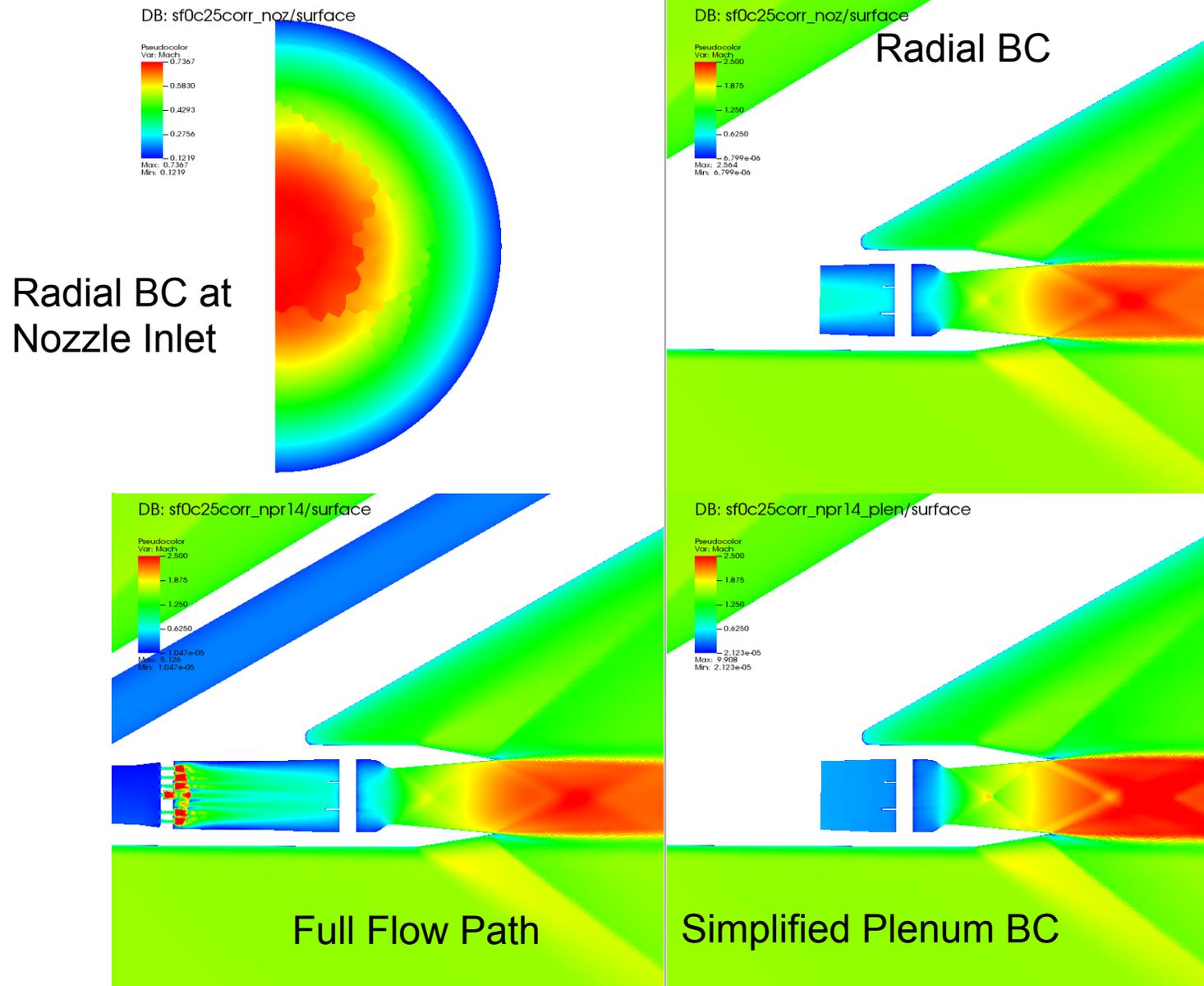
- The radial BC simulation (red) agrees well with the full flow path simulation (blue) in contrast to the constant stagnation BC based on the internal rake measurements (cyan)

Radial BC Rake Comparison (nozzle only)



➤ Satisfying agreement between radial BC and full flow path simulations.

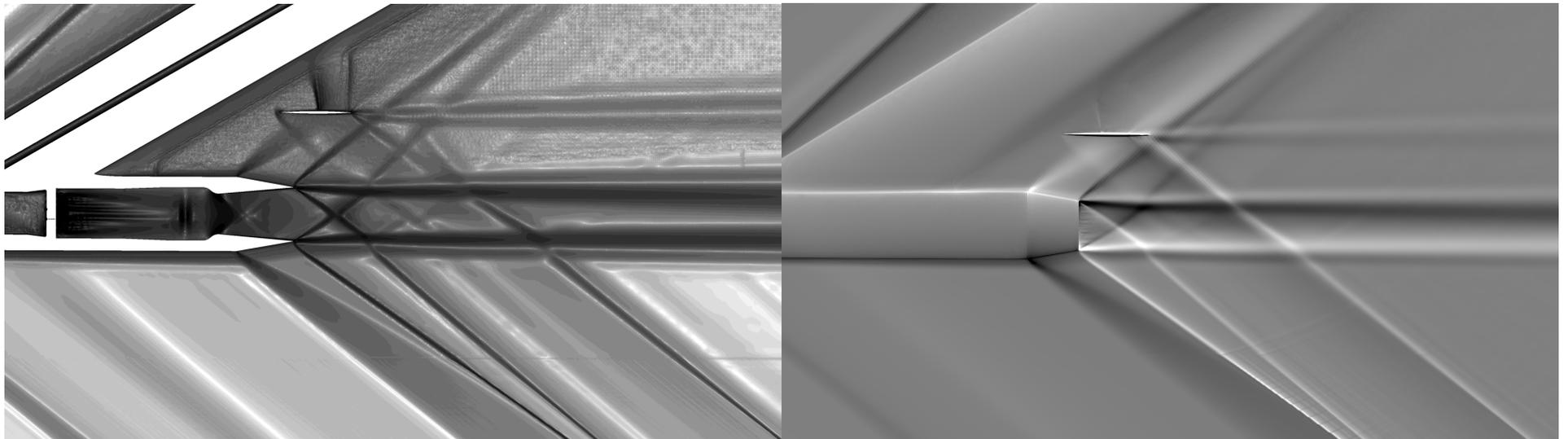
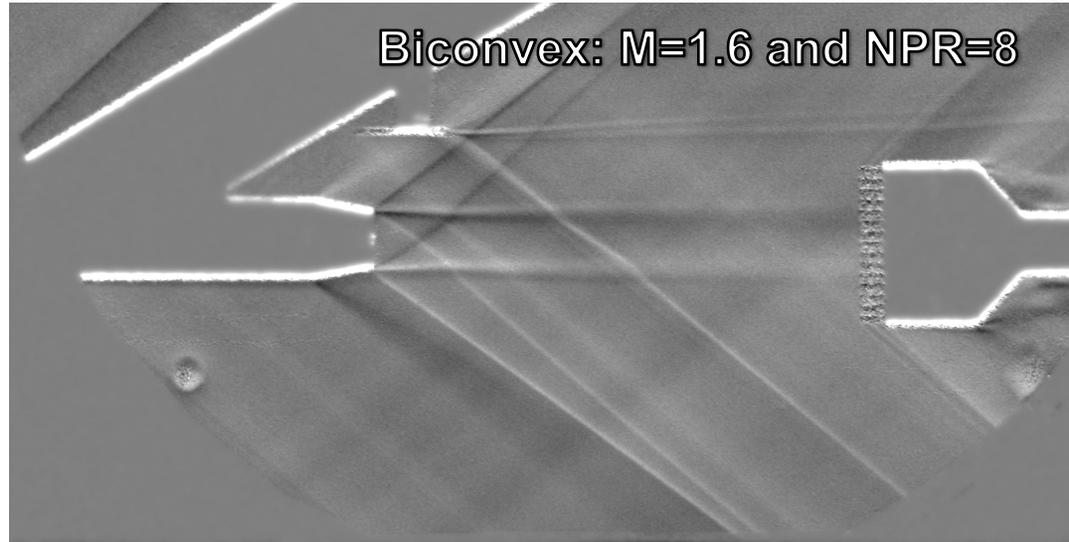
Mach Contour Comparison (nozzle only)



Improved Data Visualization



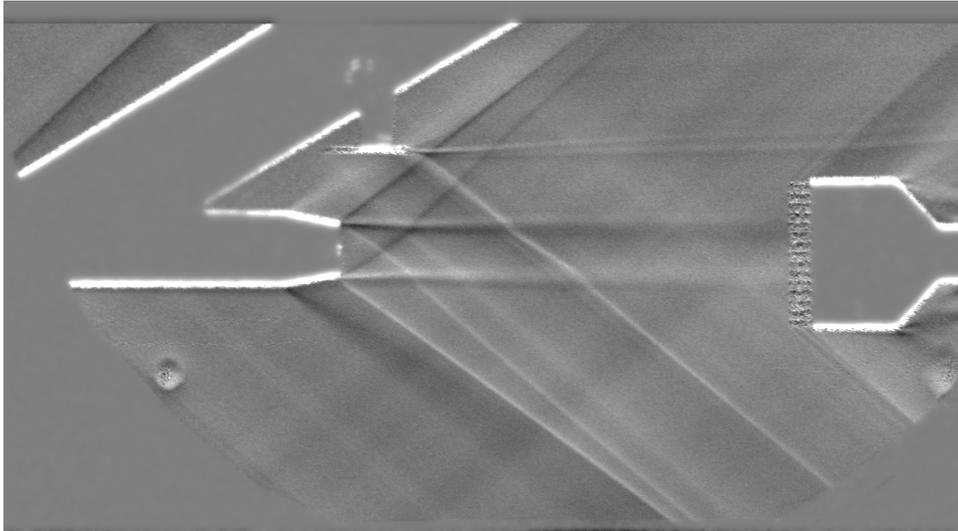
RBOS



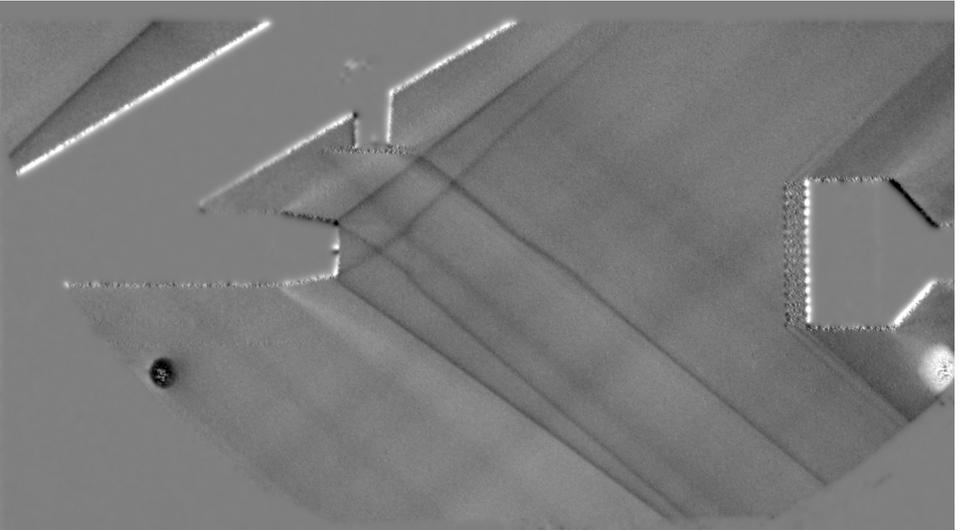
Density Gradient Slice

Numerical Schlieren

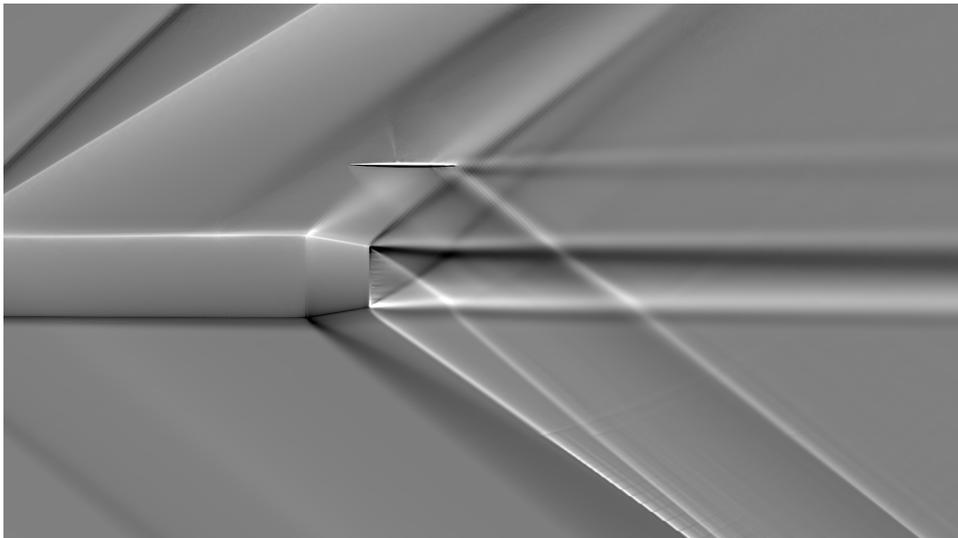
Improved Data Visualization



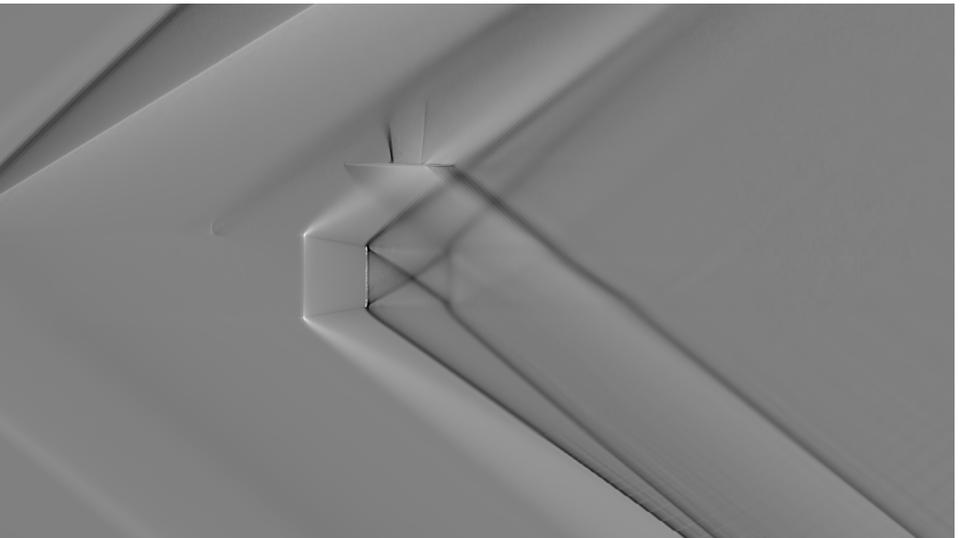
RBOS (v)



RBOS (u)



Numerical Schlieren (v)



Numerical Schlieren (u)

Summary and Conclusions



- The LAVA unstructured solver was successfully used to improve the experiment (wall effect and internal design)
- The results generated using the simplified plenum boundary condition and current meshing practices compared well to the experimental data (better match at lower NPR values)
- Using the full internal flow generated a better match with the experiment for all of the NPR values
- Using the radial BC provides a good comparison with the experiment but with the same mesh as the simplified plenum BC
- New data visualization methods allow for more comparisons between the CFD and experiment

Future Work



- Investigate the effects of the tunnel on the signatures
- Generate solutions using the LAVA curvilinear solver with structured overset meshes



Acknowledgments

- Derek Dalle, Jeff Housman, and Cetin Kiris in the Computational Aerosciences Branch at NASA Ames Research Center
- Pat Moran in the Advanced Computing Branch at NASA Ames Research Center
- Don Morr at Millennium Engineering and Integration Services
- Commercial Supersonic Technologies (CST) project under NASA's Advanced Air Vehicles Program (AAVP)
- NASA Advanced Supercomputing (NAS) center at ARC

Questions?



Lockheed-Martin Low Boom Flight
Demonstrator Concept