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

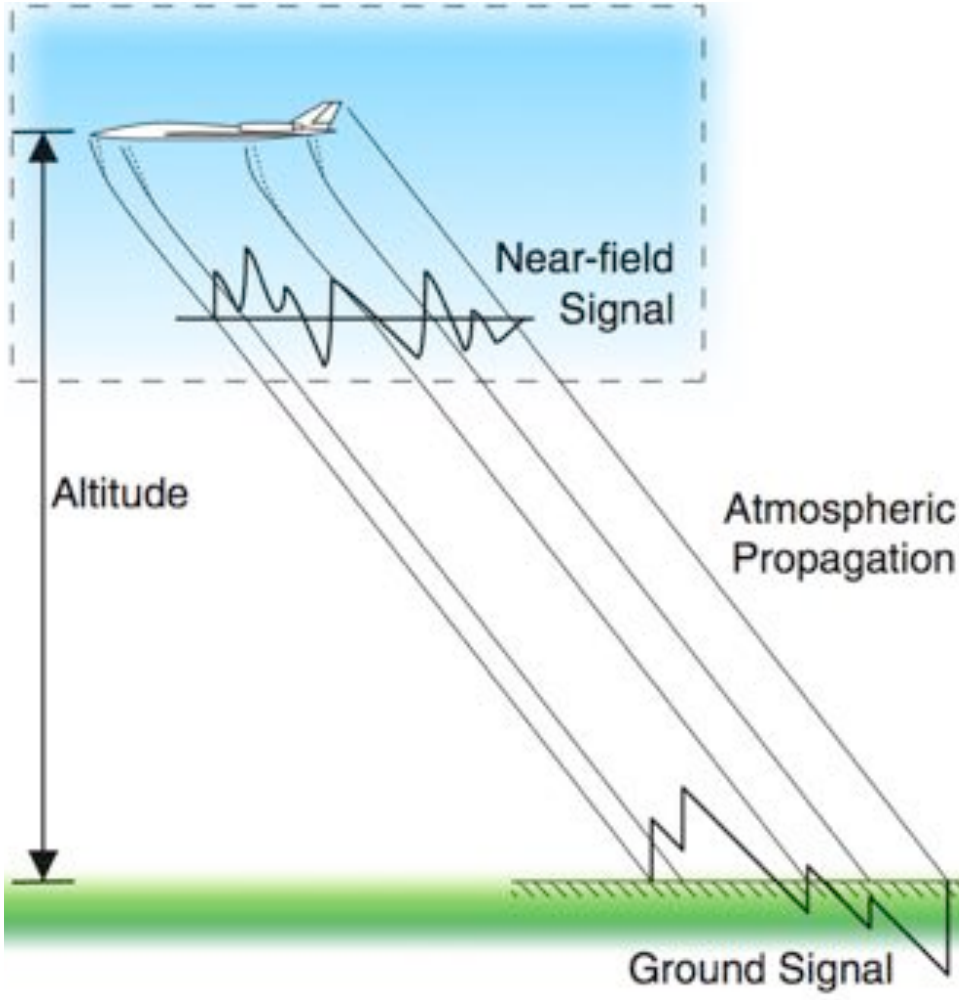
Background

- 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

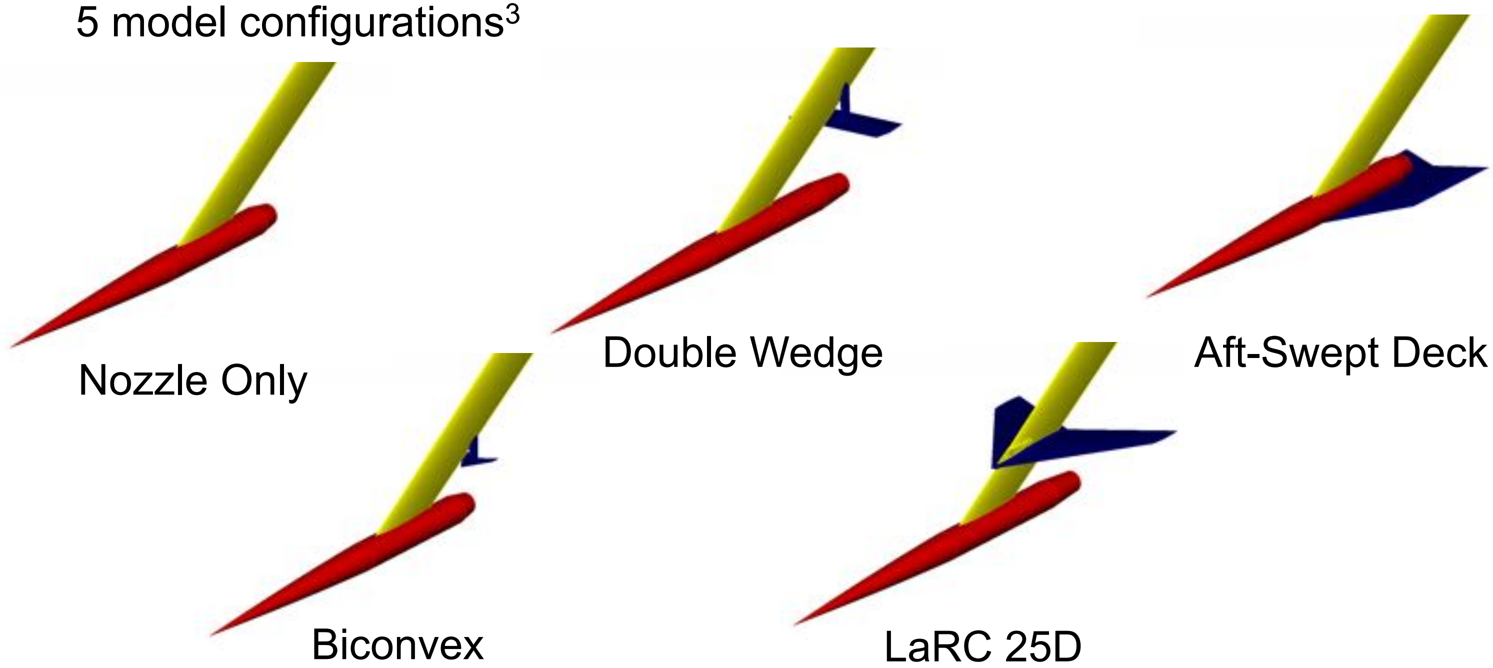
- Pressure signature prediction has two components
  - Using CFD to calculate the near-field pressure signature
  - Using atmospheric propagation codes to propagate the signal to the ground

- 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



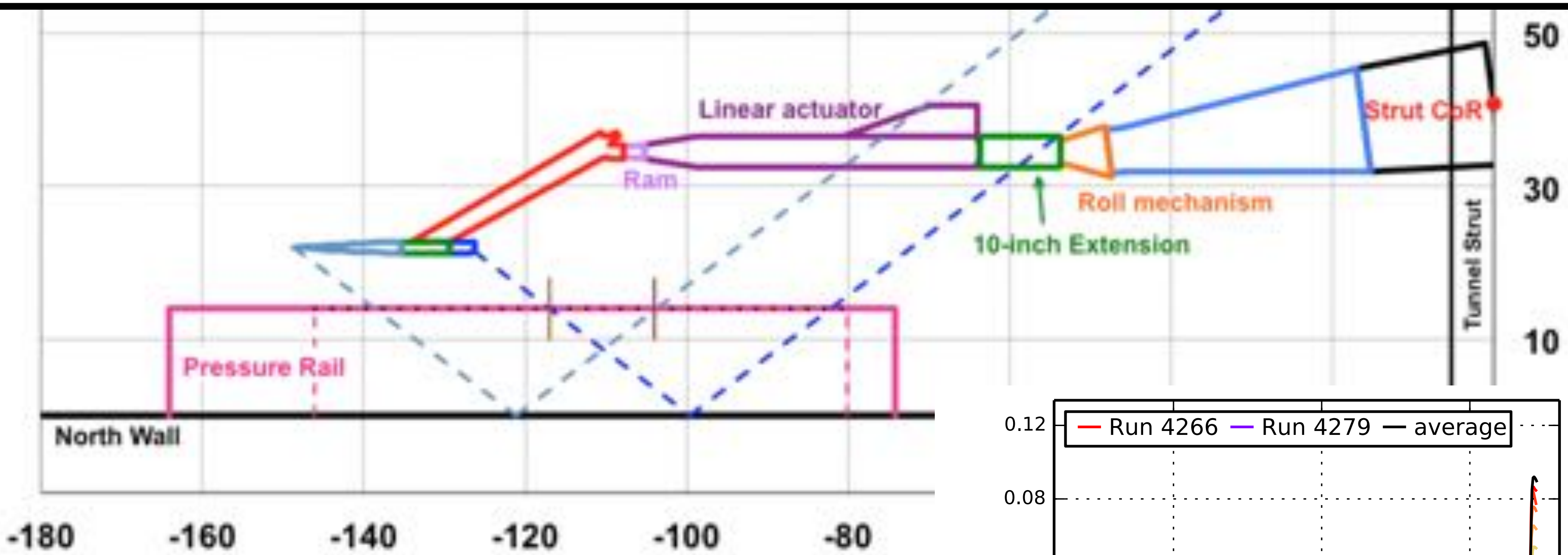
Background

- 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 configurations[3]

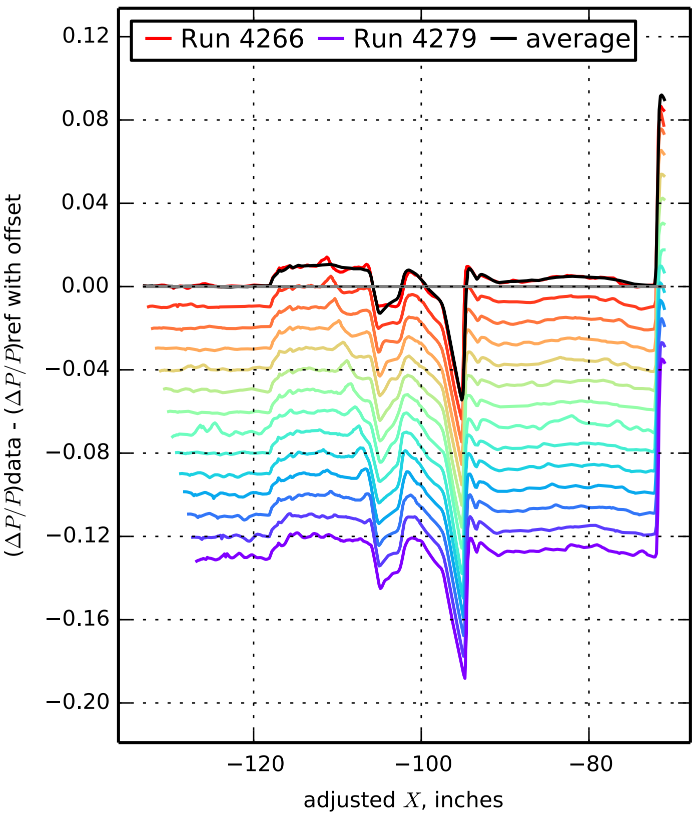


[1]Castner et al, AIAA 2015-1046, [2]Durston et al, AIAA 2017-0041, [3]Cliff et al, AIAA 2016-2035

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)



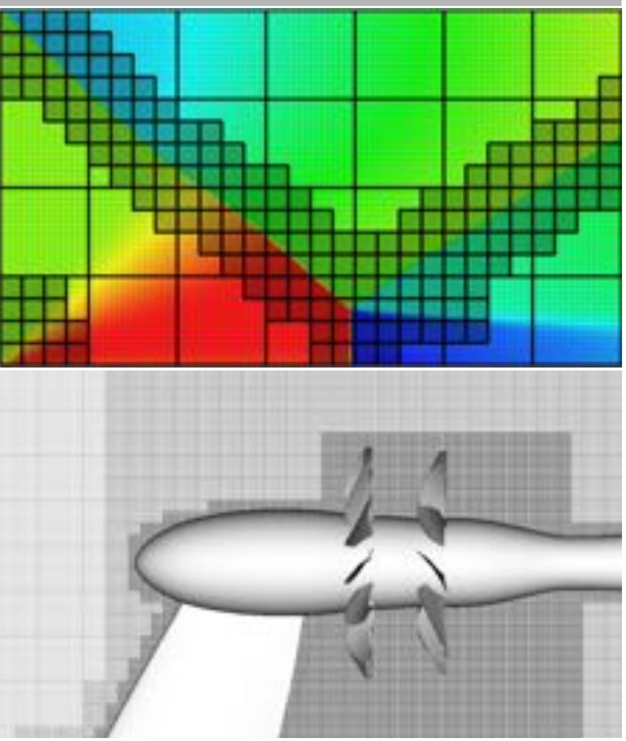
Outline

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

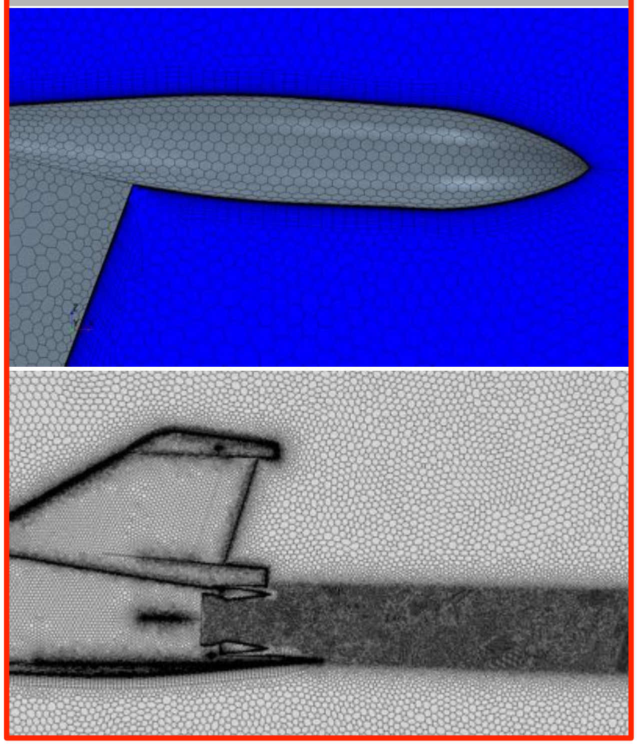
Methodology: Solver

Launch Ascent and Vehicle Aerodynamics (LAVA) framework[4]

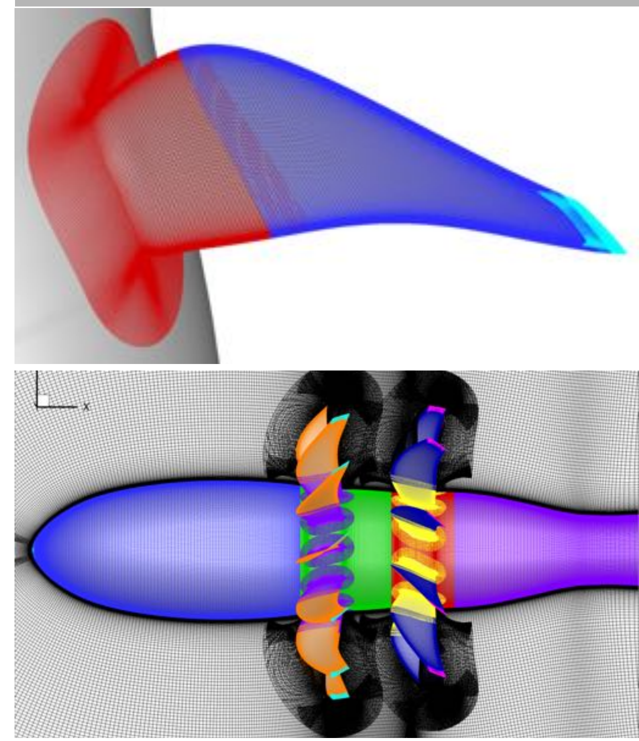
Structured Cartesian with AMR



Unstructured Arbitrary Polyhedral



Structured Curvilinear

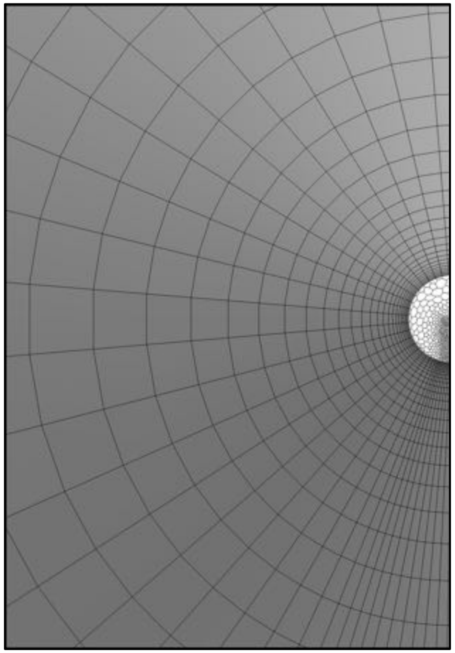


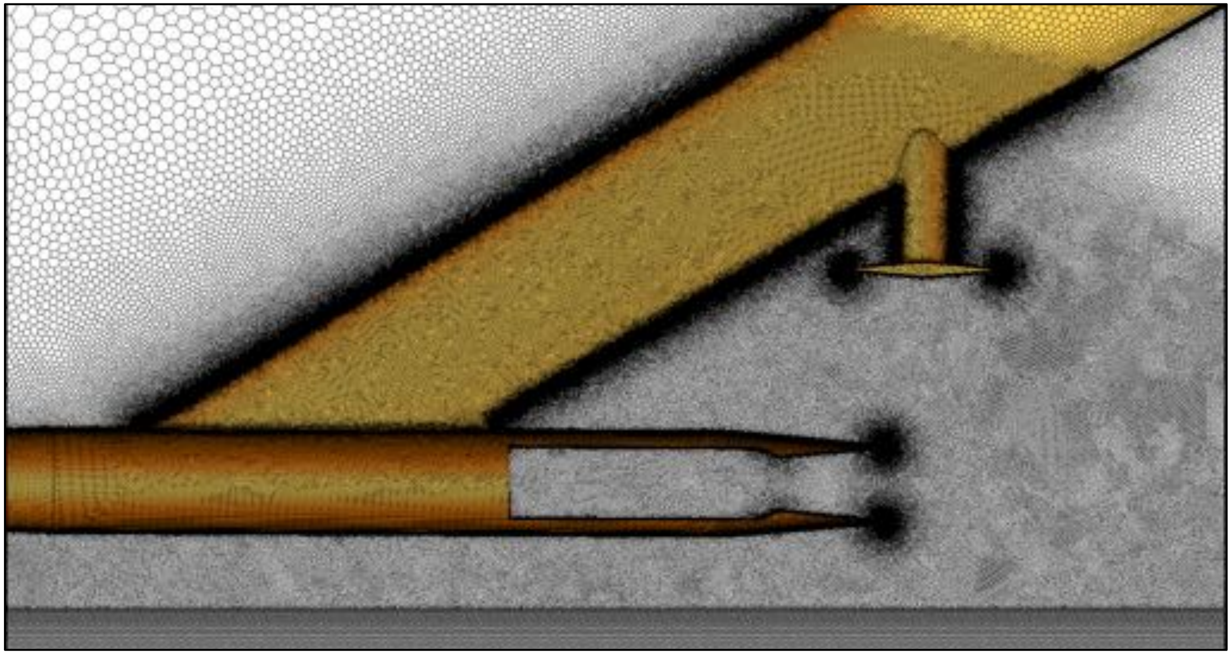
- Unstructured Polyhedral Solver (2nd Order)
  - Steady Reynolds-Averaged Navier-Stokes (RANS)
  - Spalart-Allmaras turbulence model

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

Methodology: Mesh Generation

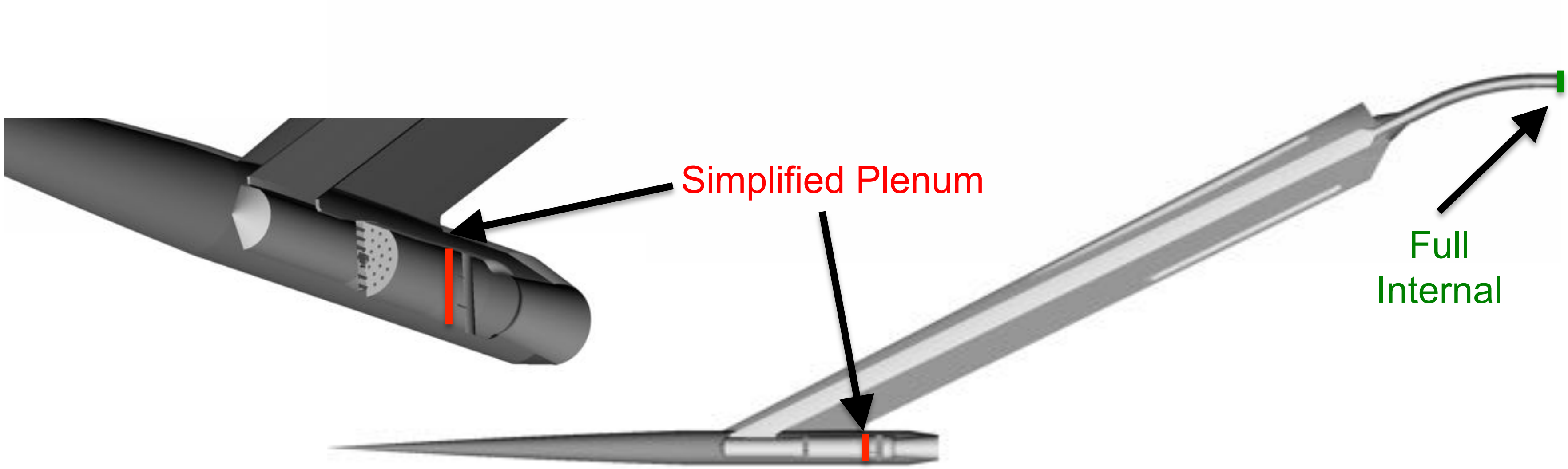
- The mesh was generated using a near body "core cylinder" mesh and Mach aligned extruded prismatic layers
- A Mach aligned refinement region exists between the model and the extruded layer boundary
- Angle of attack is treated by rotating the model relative to the free stream flow vector





Methodology: Boundary Condition

- Three 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)
- Plenum BC was used for the pre-test external design study and for a fast sweep of post-test CFD
- Internal flow BCs were used for the pre-test internal flow path design study and for detailed post-test CFD



Outline

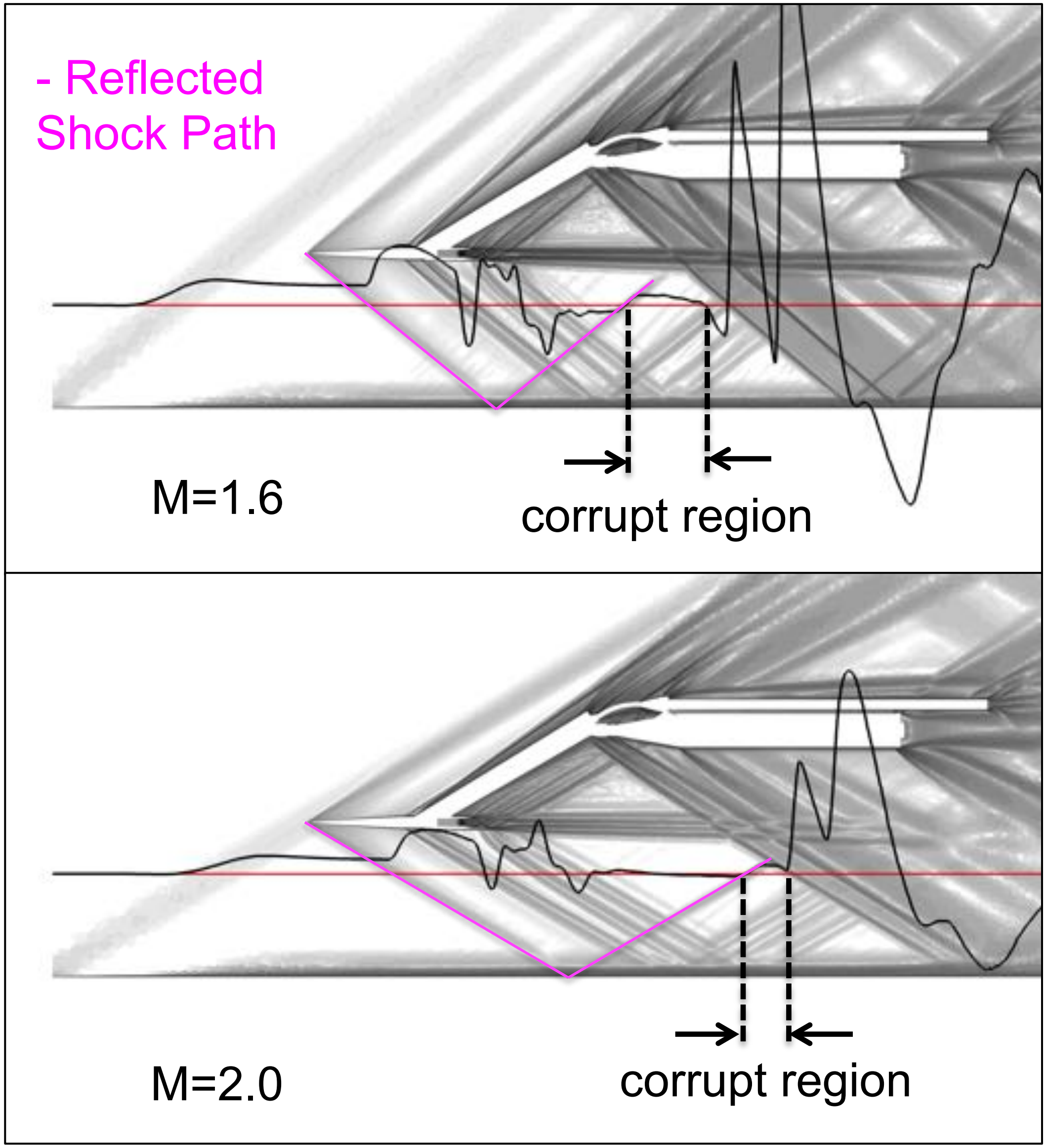
- Background
- Methodology
- Pre-Test CFD
- Post-Test CFD
  - Current Methods
  - Detailed Study
  - Recent Developments
- Conclusions

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

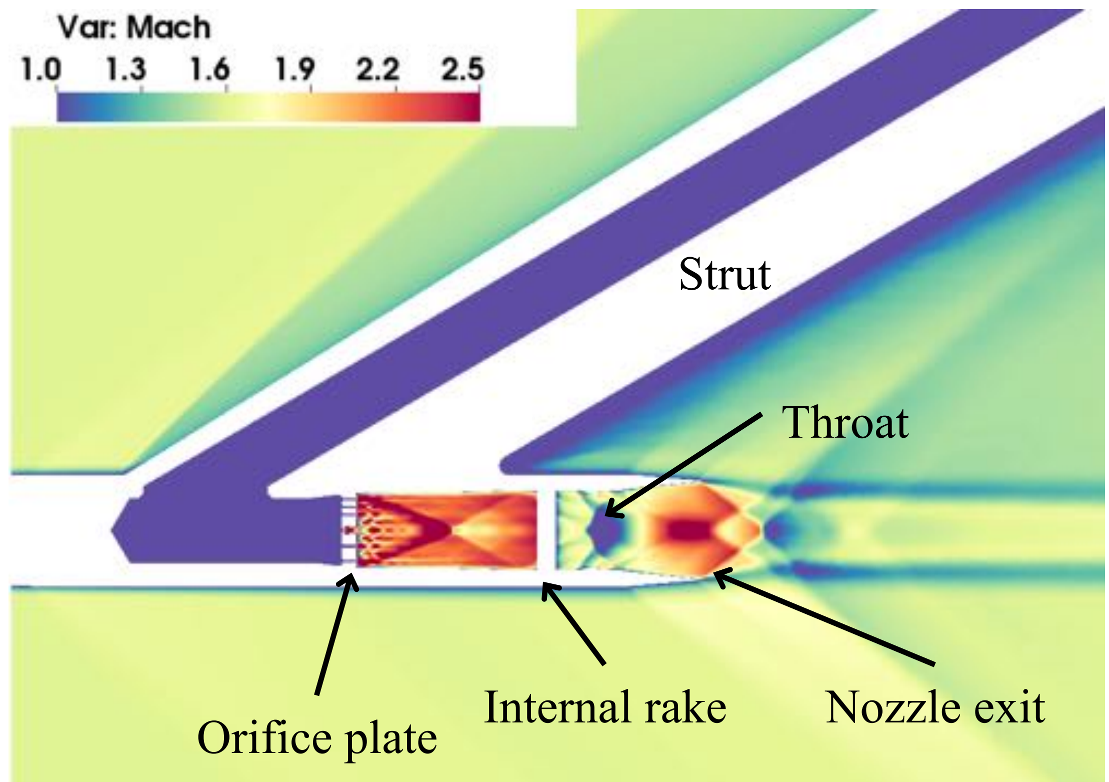
- Original nozzle design was found to have supersonic flow between the orifice plate and the internal rake, additionally, the nozzle throat was not chocked

- 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)

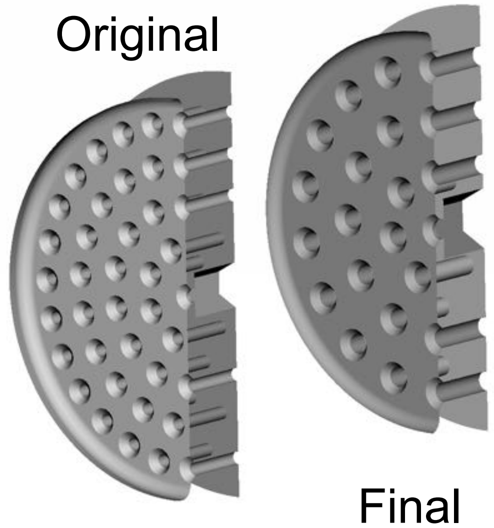
2. Extraneous shocks would propagate from the nozzle into the plume

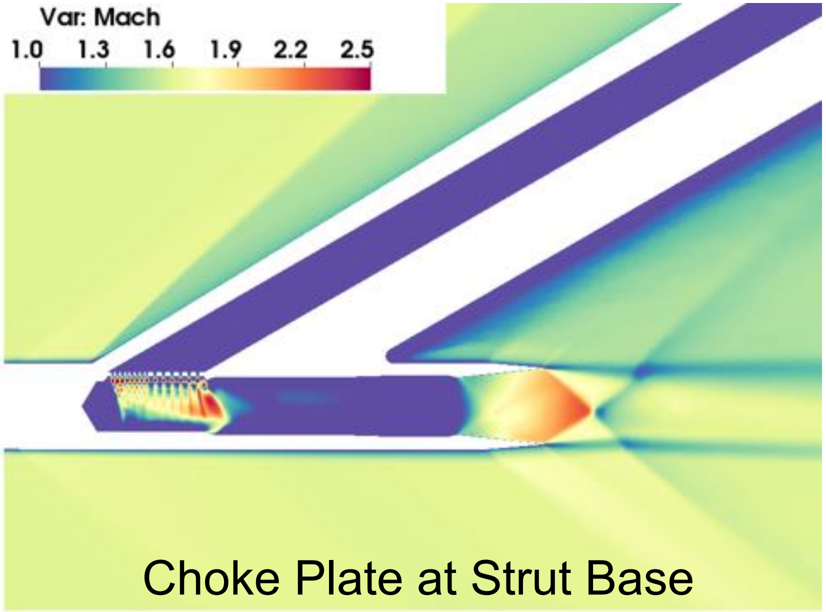
3. Lack of choking at the throat would lead to a less uniform plume that would be more sensitive to the flow conditions



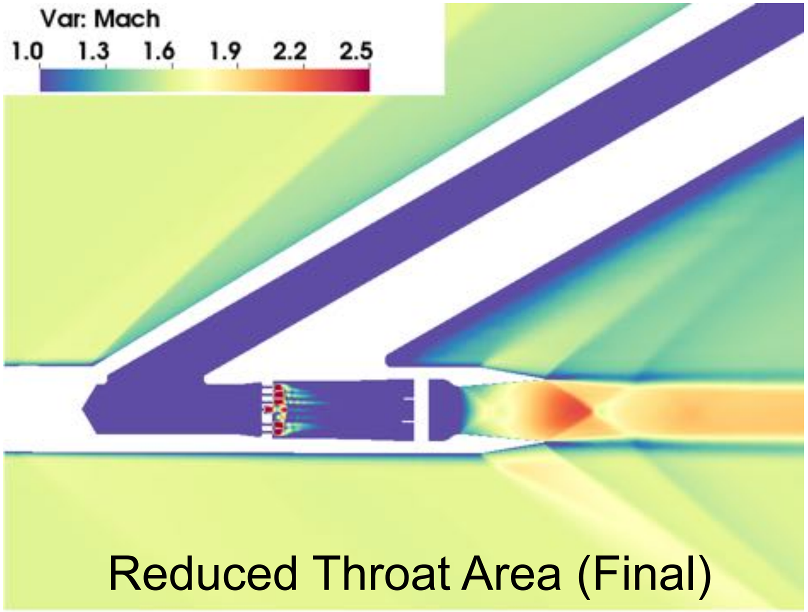
Internal Flow Path Design

- ➢ 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.





Choke Plate at Strut Base



Reduced Throat Area (Final)

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

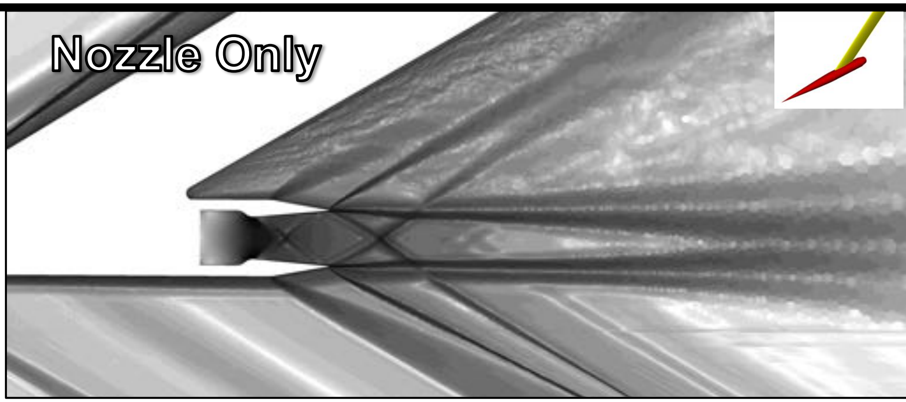
Outline

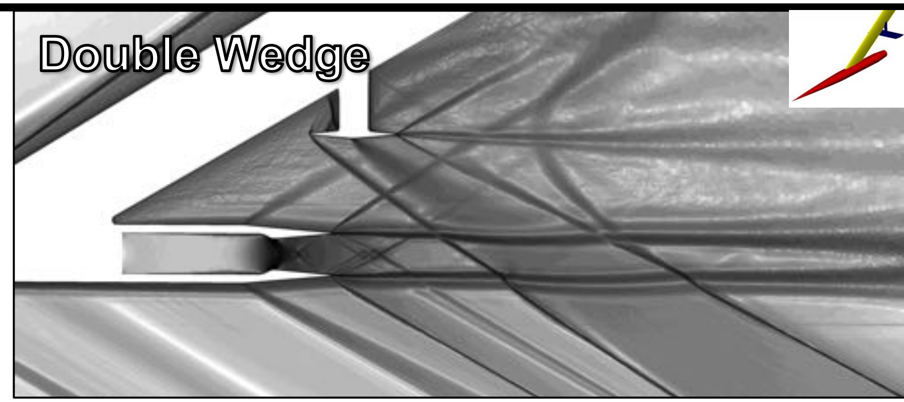
- Background
- Methodology
- Pre-Test CFD
- Post-Test CFD
  - Current Methods
  - Detailed Study
  - Recent Developments
- Conclusions

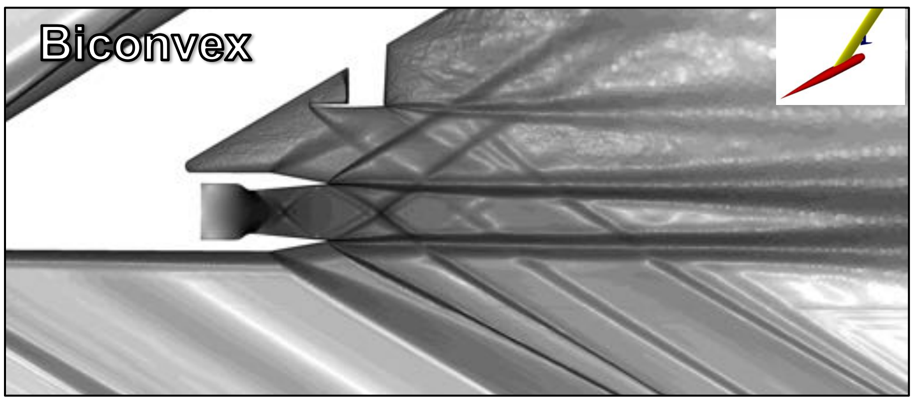
Post-Test CFD Current Methods Objectives

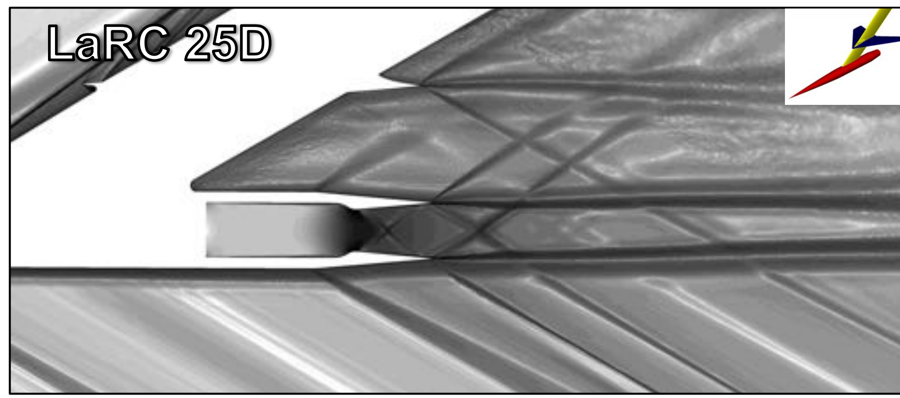
- Compare CFD results generated with a simplified plenum boundary condition to the wind tunnel test data and assess their validity
    - All test configurations
    - Mach number effect
    - Angle of attack effect
    - Model altitude effect
    - Off-track effect
    - Nozzle pressure ratio effect

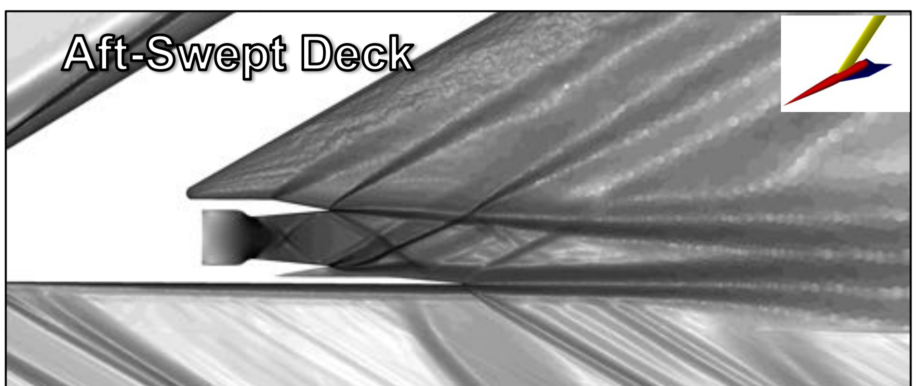
Model Density Gradient Comparison







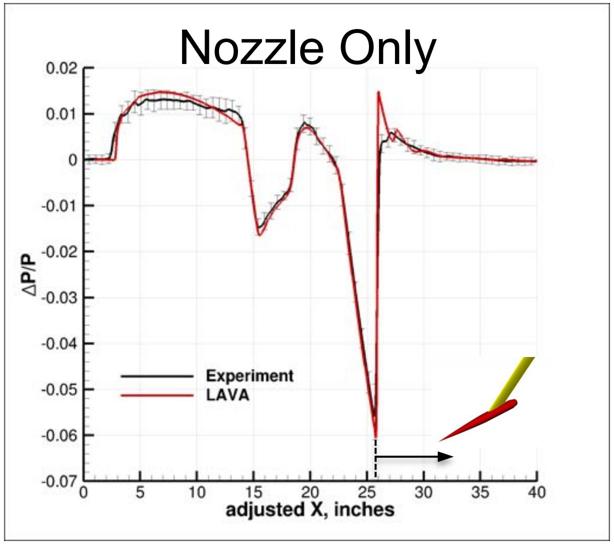


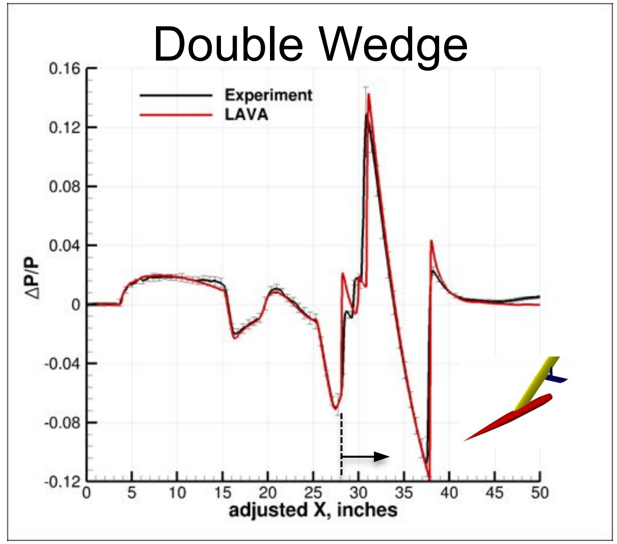


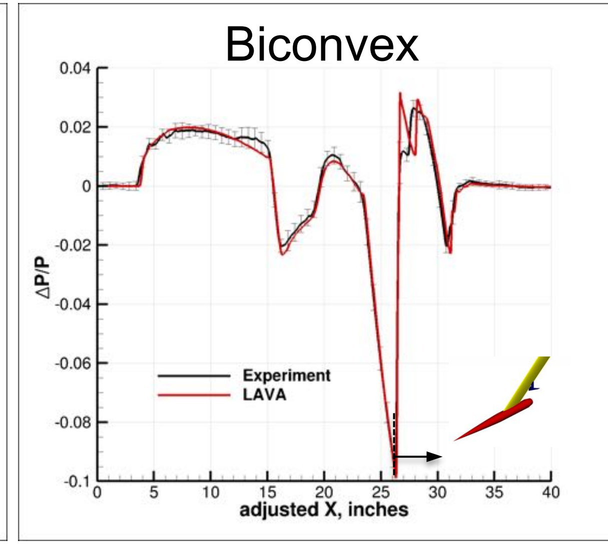
M=2.0 and NPR=8
(Double Wedge
NPR=10)

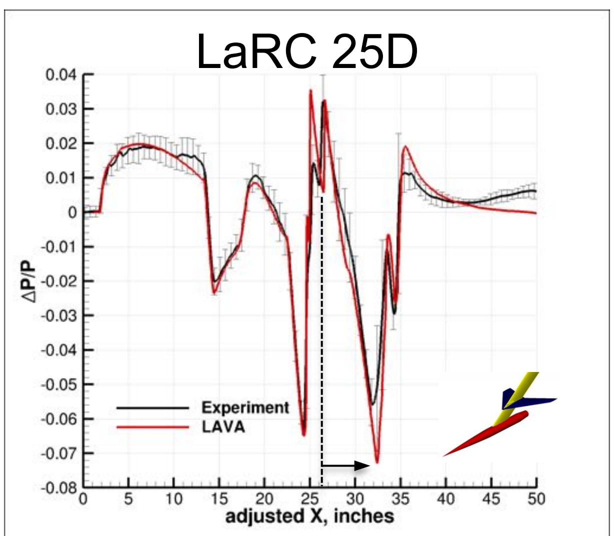
Model Signature Comparison

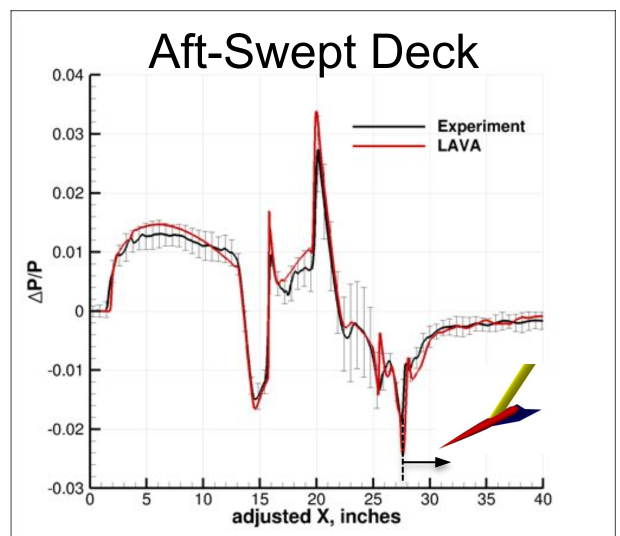
M=2.0, H=8” (Aft-Swept Deck H=15”), and NPR=8 (Double Wedge NPR=10)





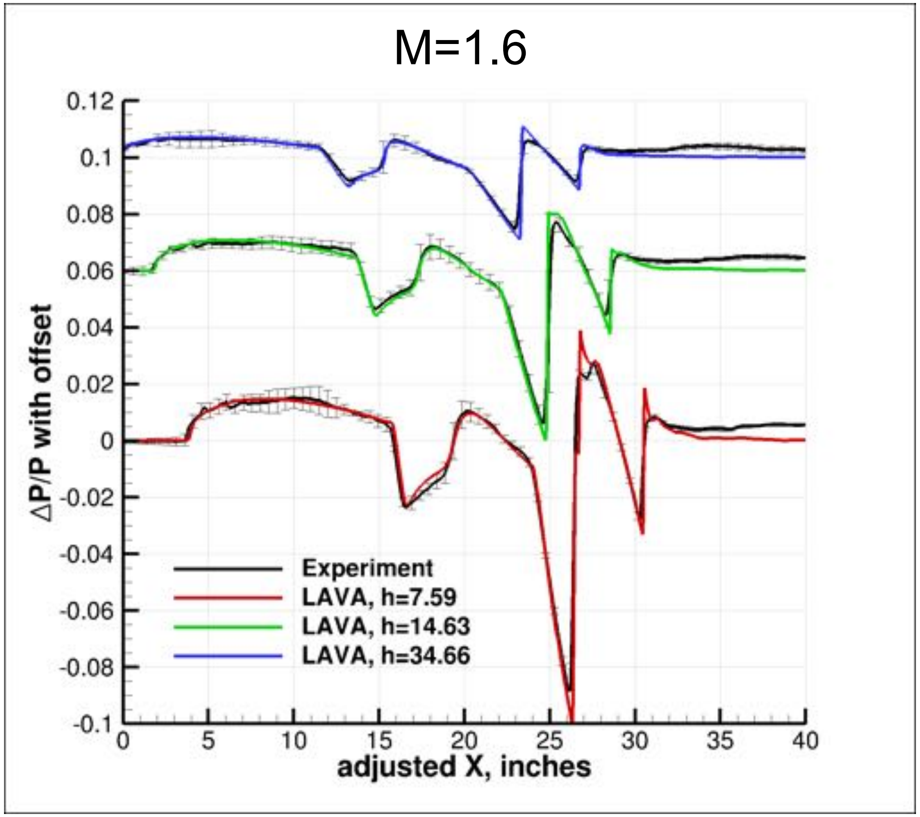


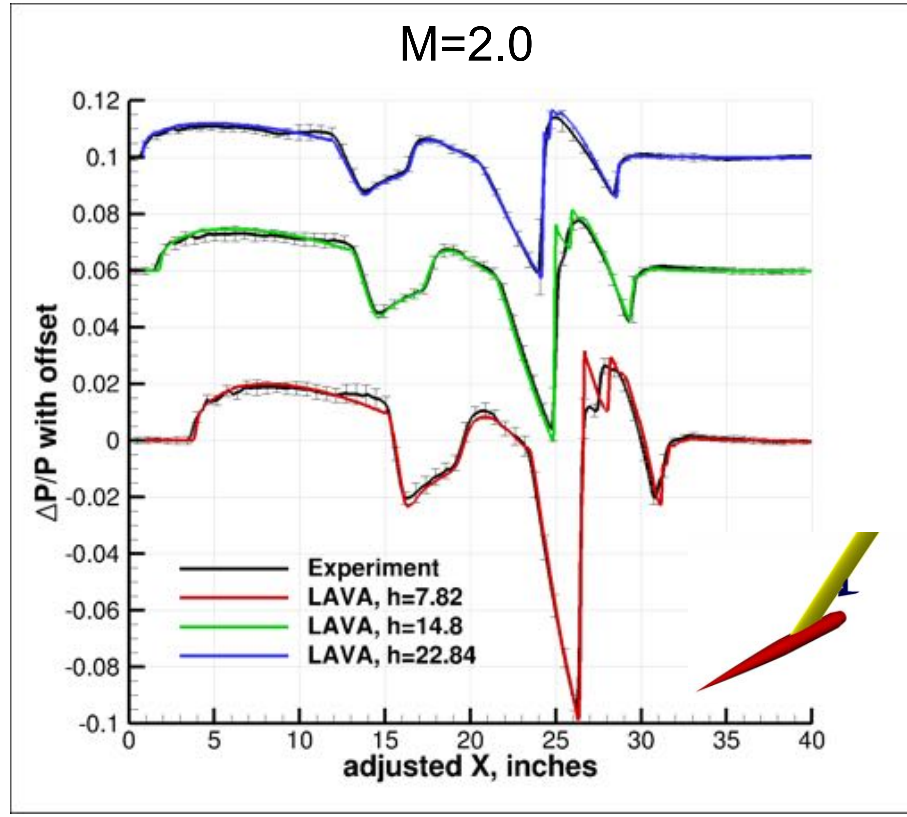




Mach Number Comparison

Biconvex Configuration at NPR=8

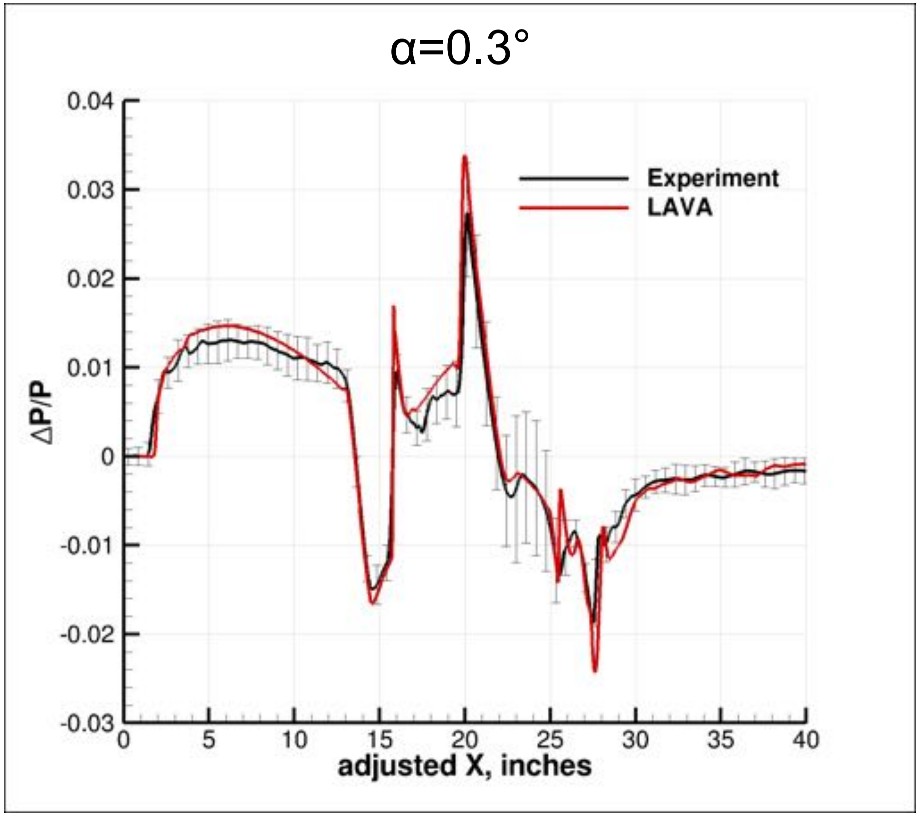


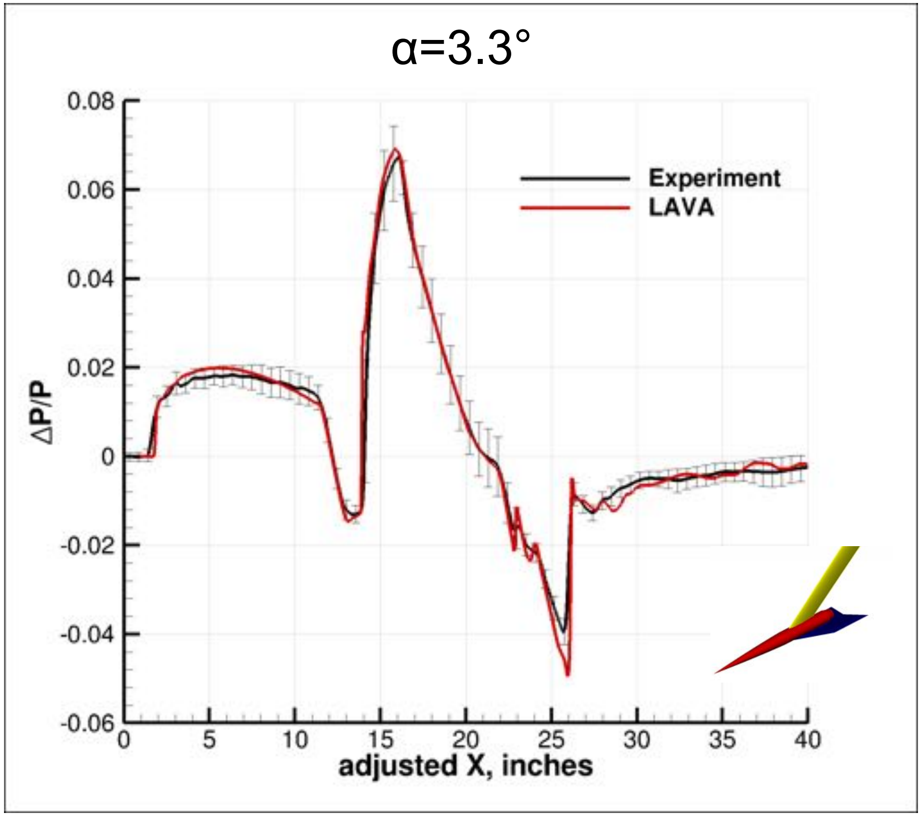


- ➢ The Mach number does not have a negative impact on the comparison between the experiment and the CFD

Angle of Attack Comparison

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



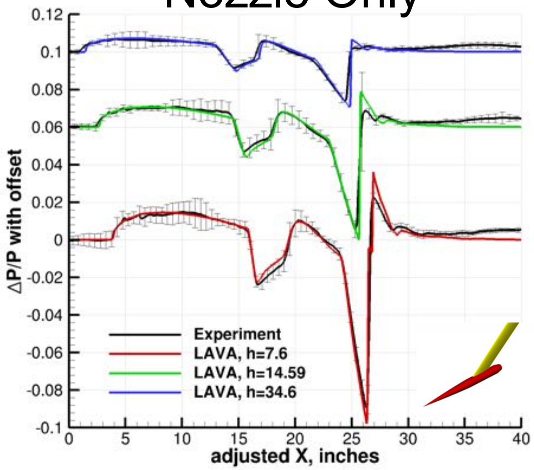


- Changing the angle of attack does not negatively affect the comparison between the experiment and the CFD

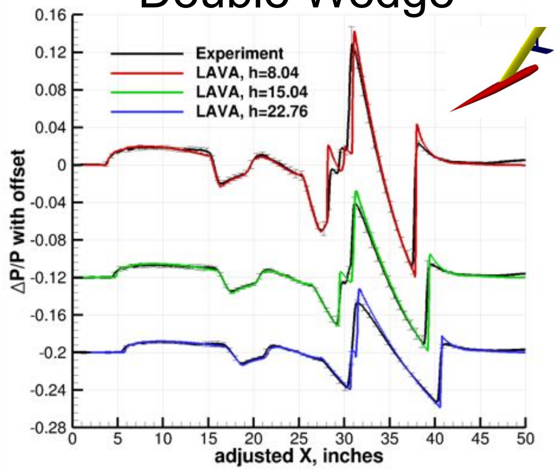
Altitude Comparisons

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

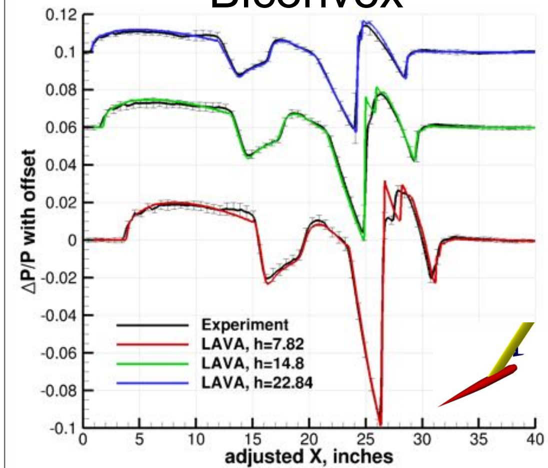
Nozzle Only



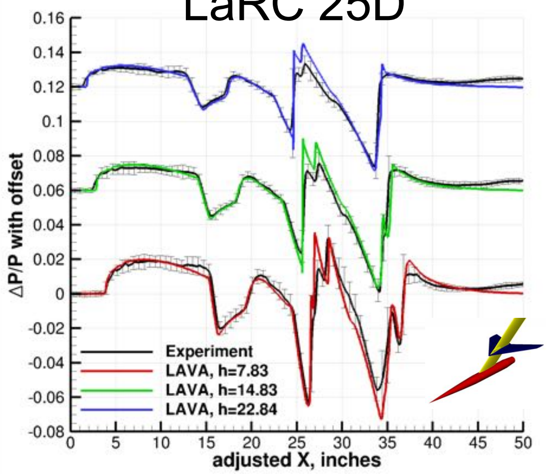
Double Wedge



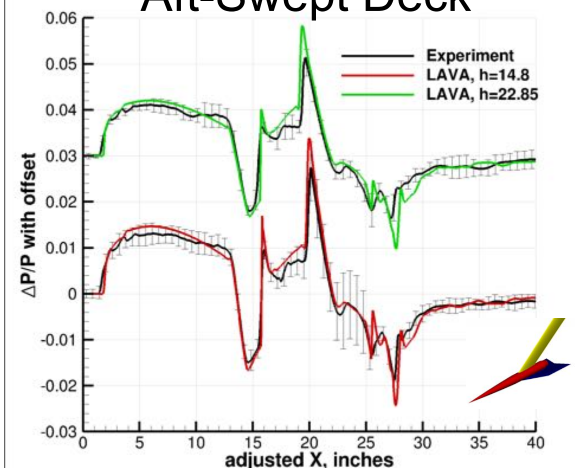
Biconvex



LaRC 25D



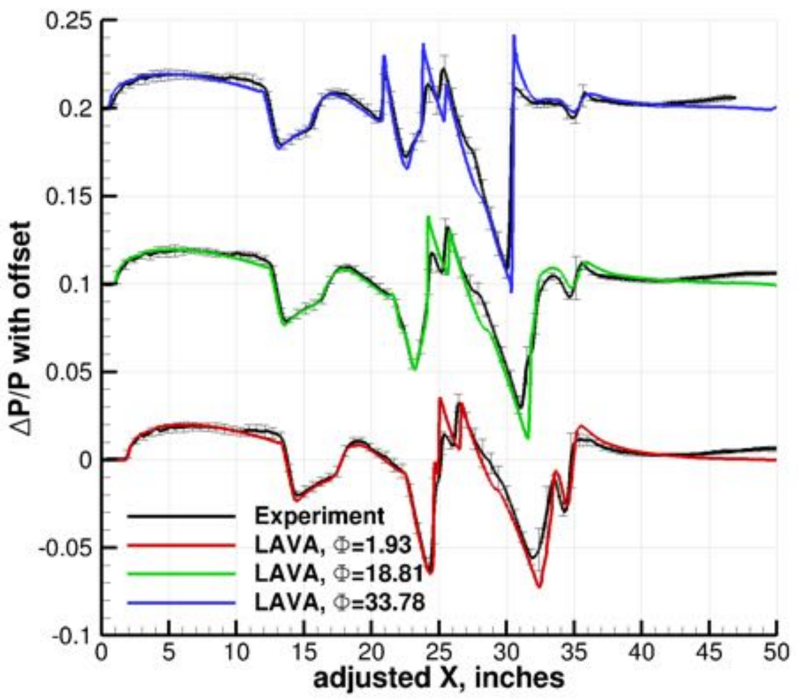
Aft-Swept Deck



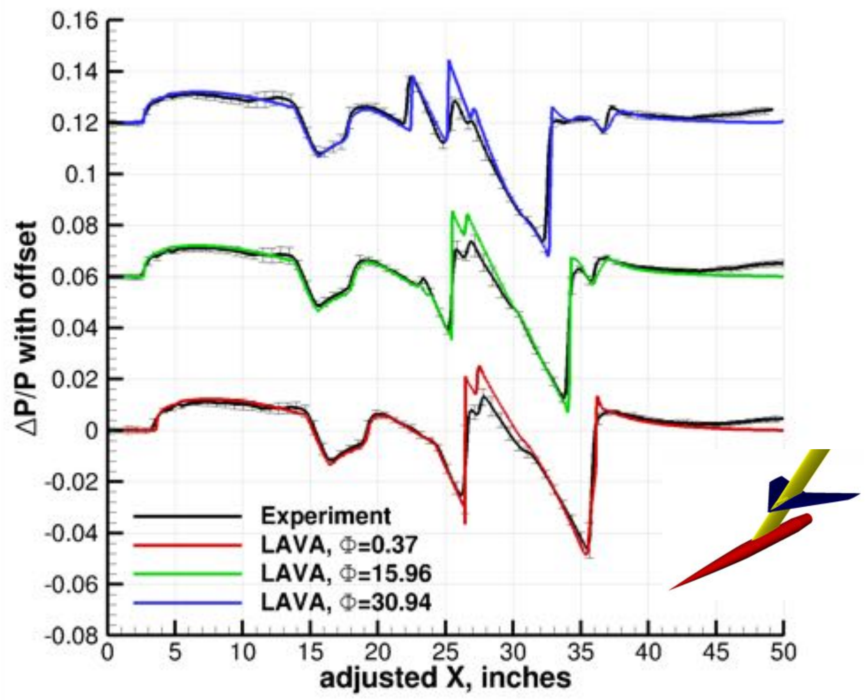
Off-Track Comparisons

LaRC 25D at M=2.0 and NPR=8

H=8’’

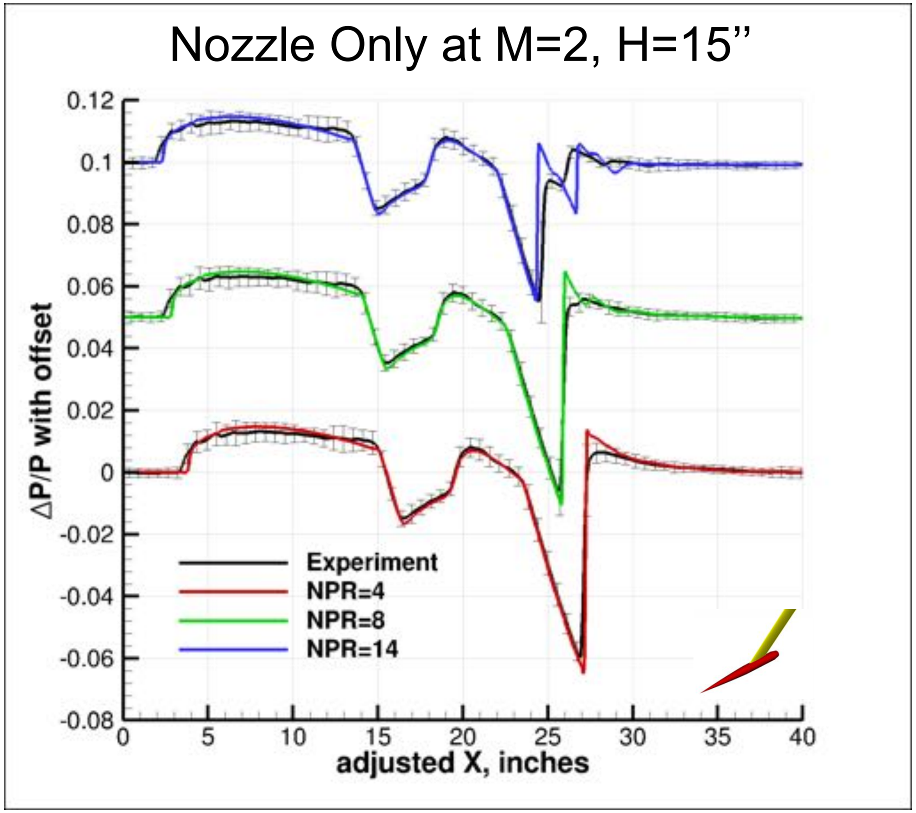


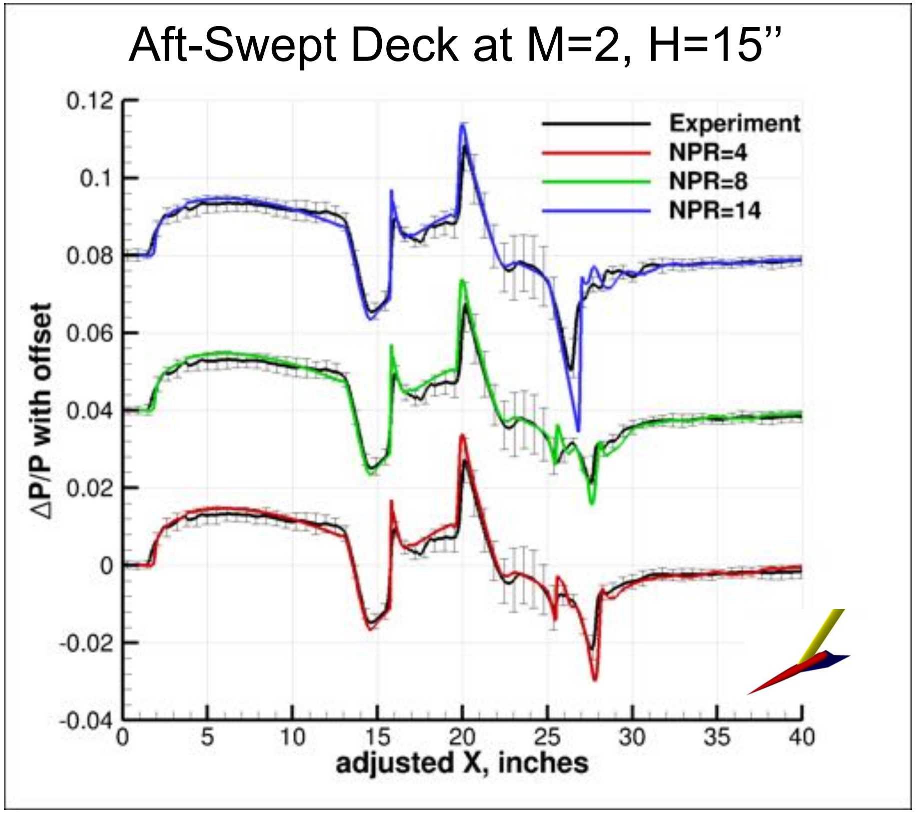
H=23’’



- 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

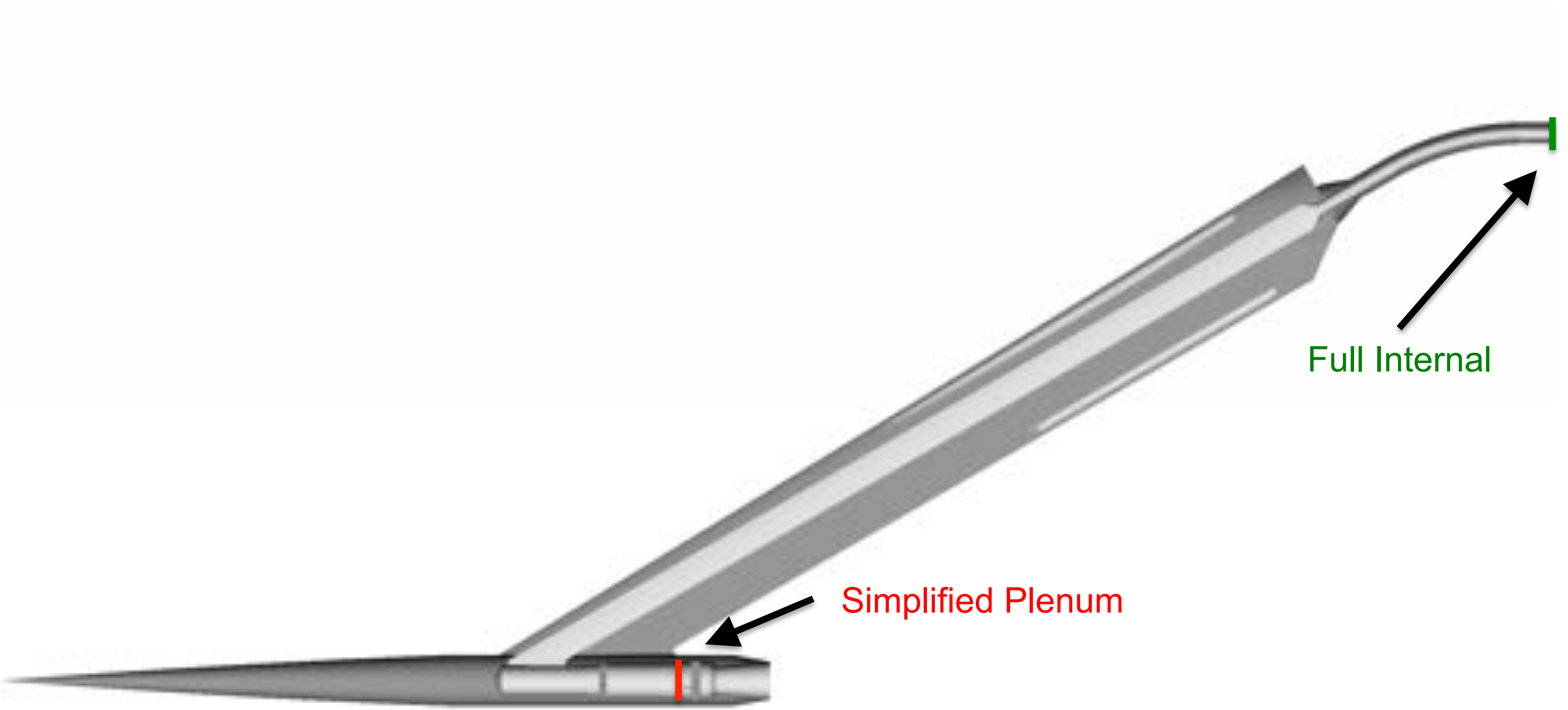
- The results generated using the plenum boundary condition compared well to the experimental data
  - Changing the conditions of the simulations (Mach and angle of attack) as well as the location of where the data was extracted (model altitude and off-track angle) did not negatively affect the comparison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

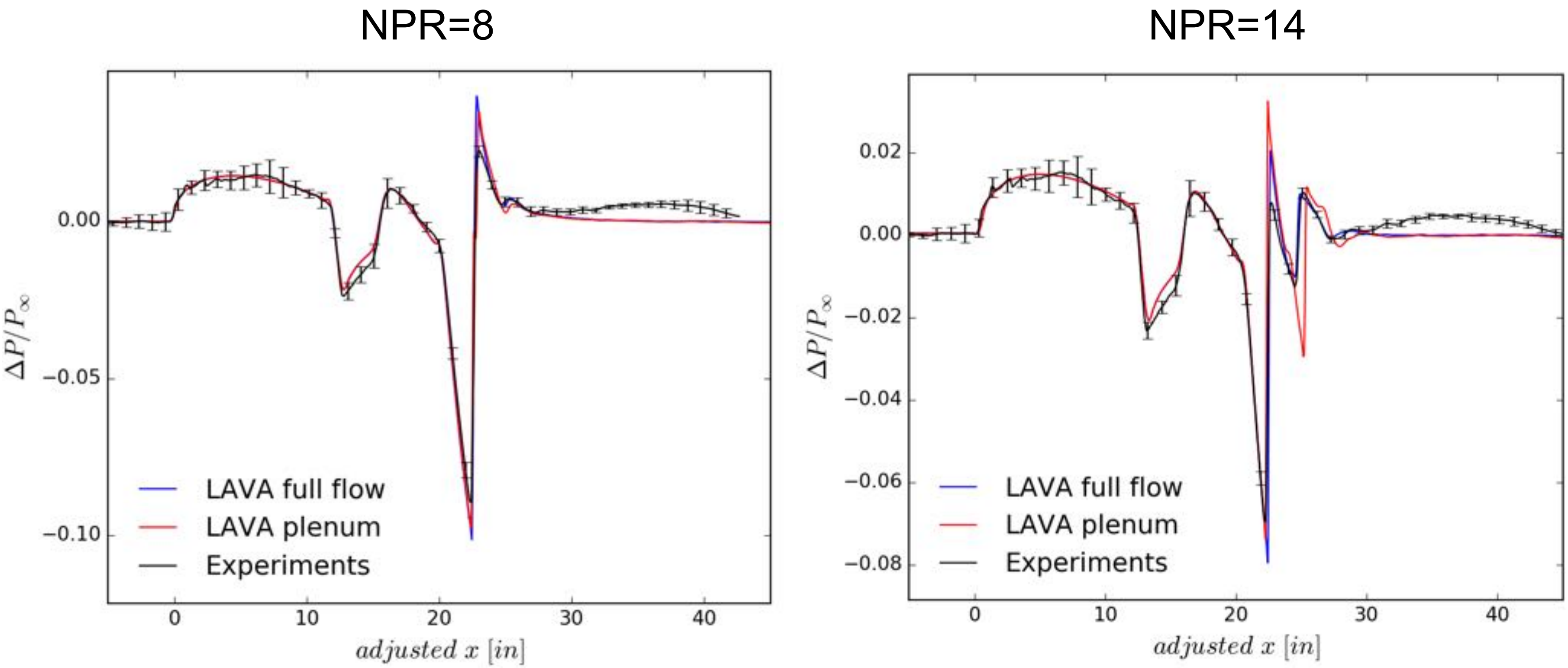
Detailed CFD Study with Internal Flow Path

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



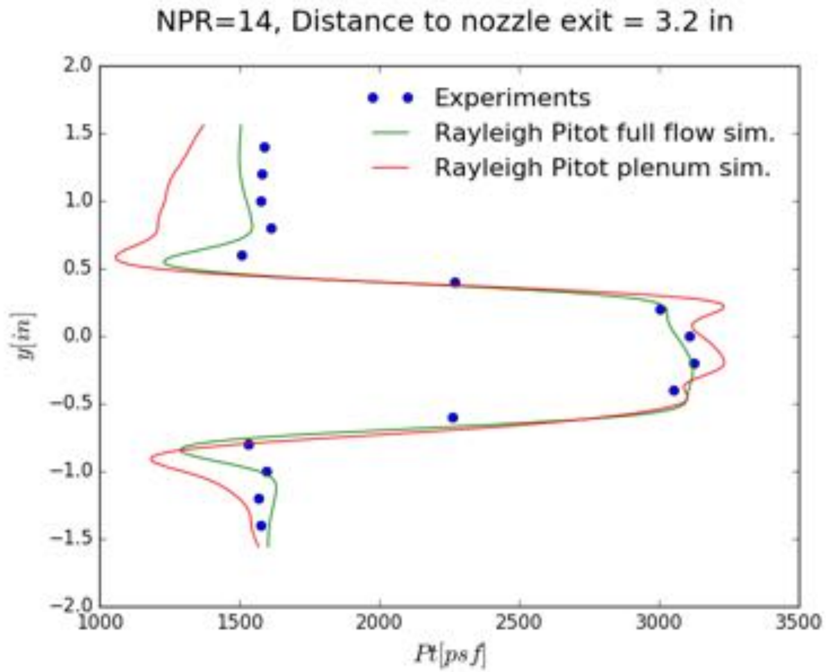
Pressure Signature Comparisons

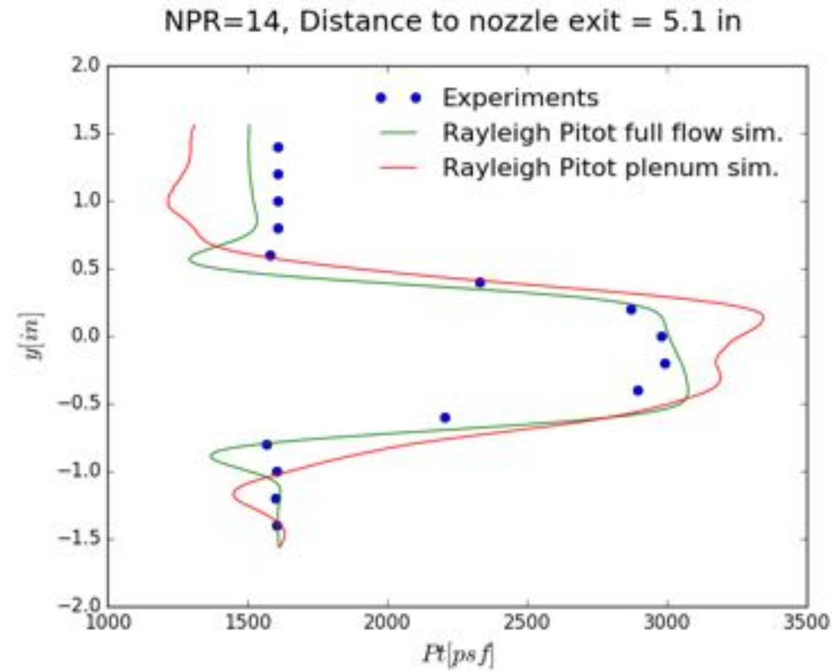
Nozzle Only at M=1.6 and H=15”

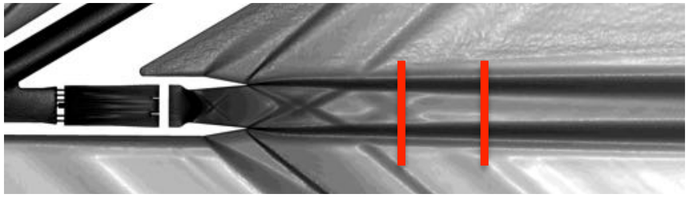


- At NPR=8 the full flow, plenum, and experiment are all in close agreement but at NPR=14 the plenum boundary condition signature has significant differences as compared to the experiment

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



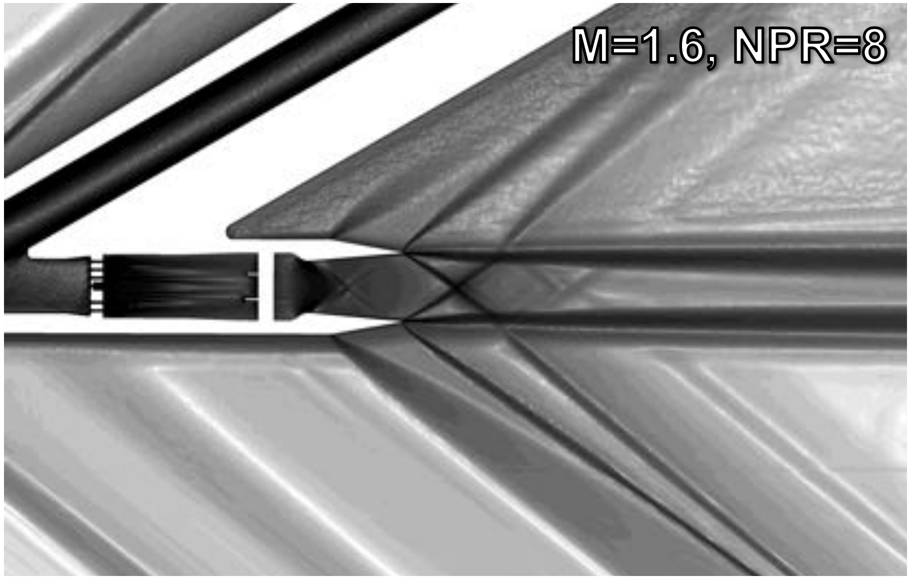


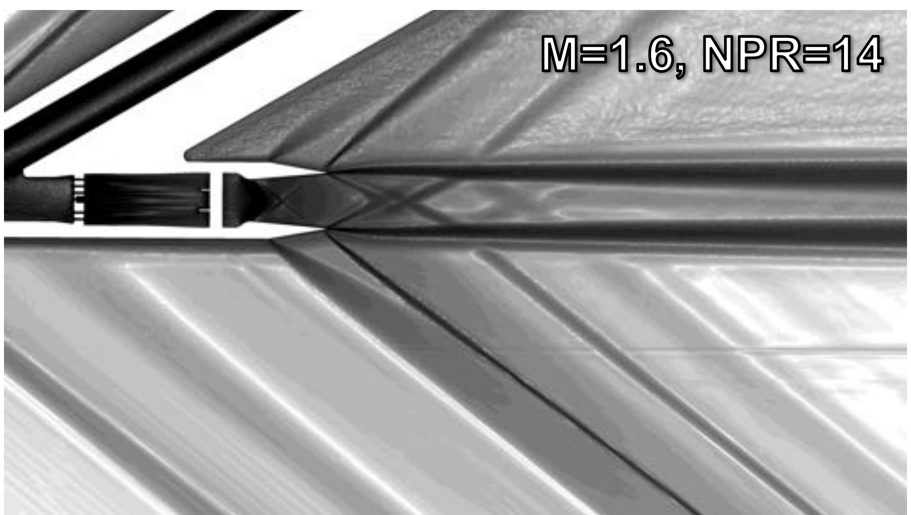


- The probe data are approximated from the simulations by applying the Rayleigh Pitot tube normal shock correction at the location of the probes
- Included corrections for the drooping and AOA of the rake

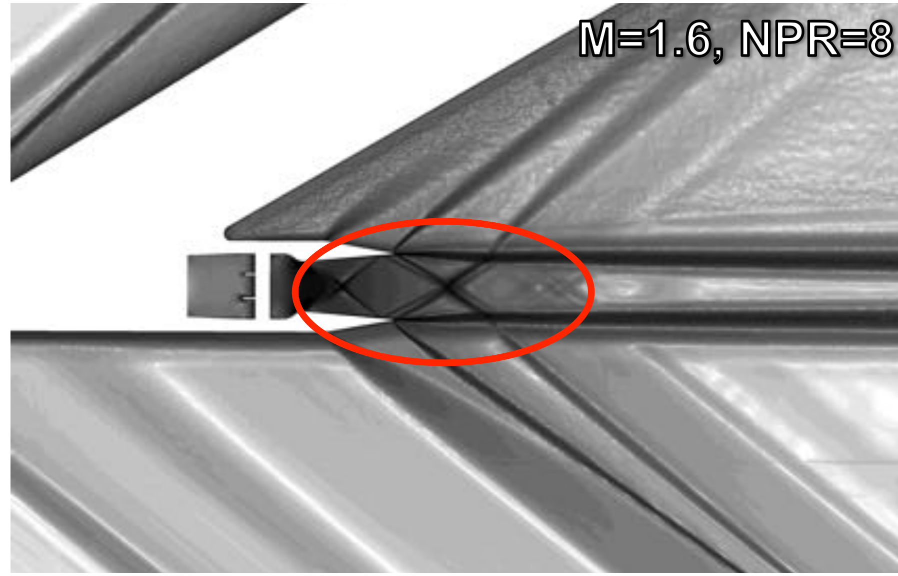
Density Gradient Comparison (Nozzle Only)

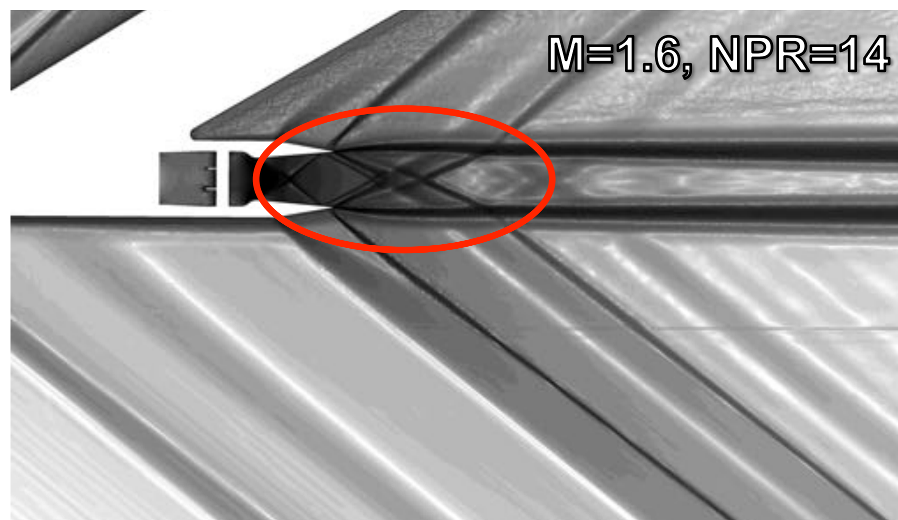
Full Internal Flow





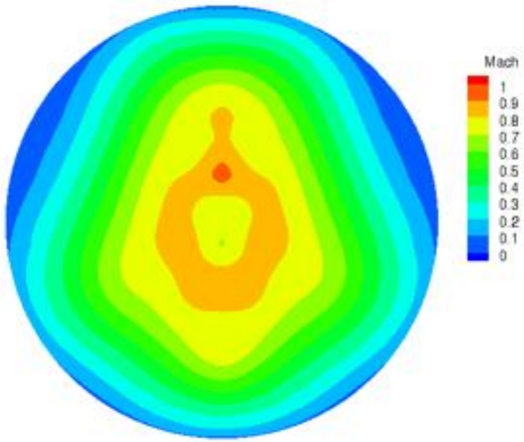
Plenum

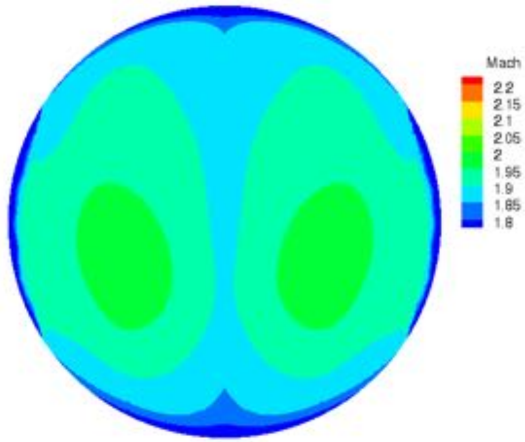


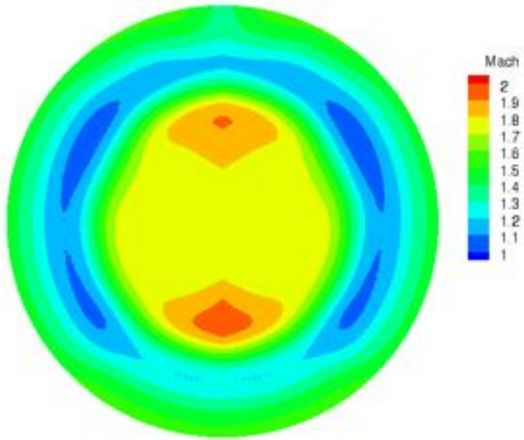


Internal Mach Contour Comparisons (Nozzle and NPR=8)

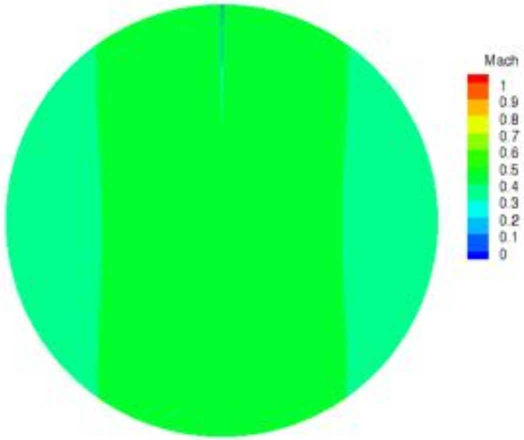
Near Internal Rake | Nozzle Exit | 3 Diameters Downstream

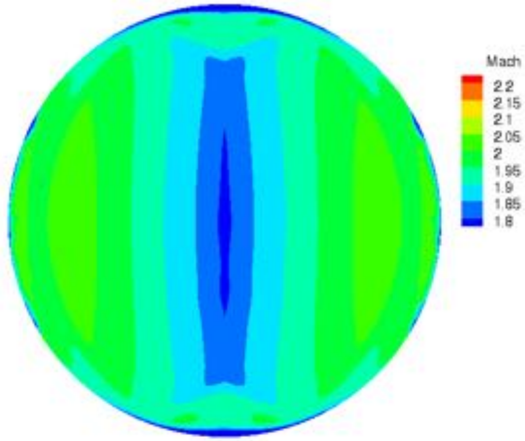


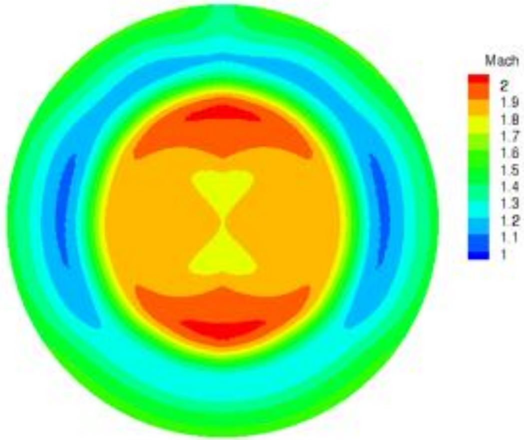




Mach Profiles for Full Internal Flow



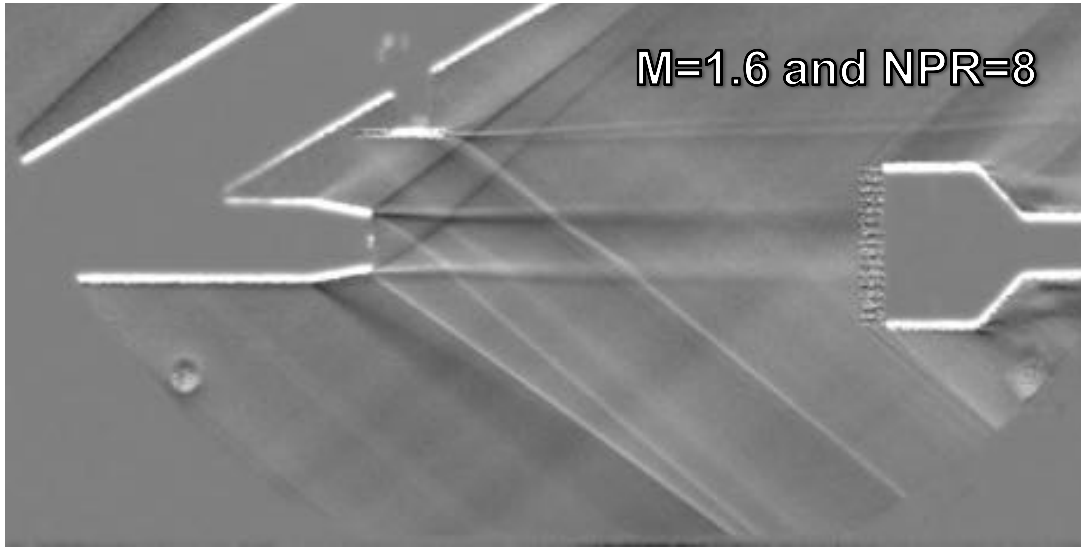




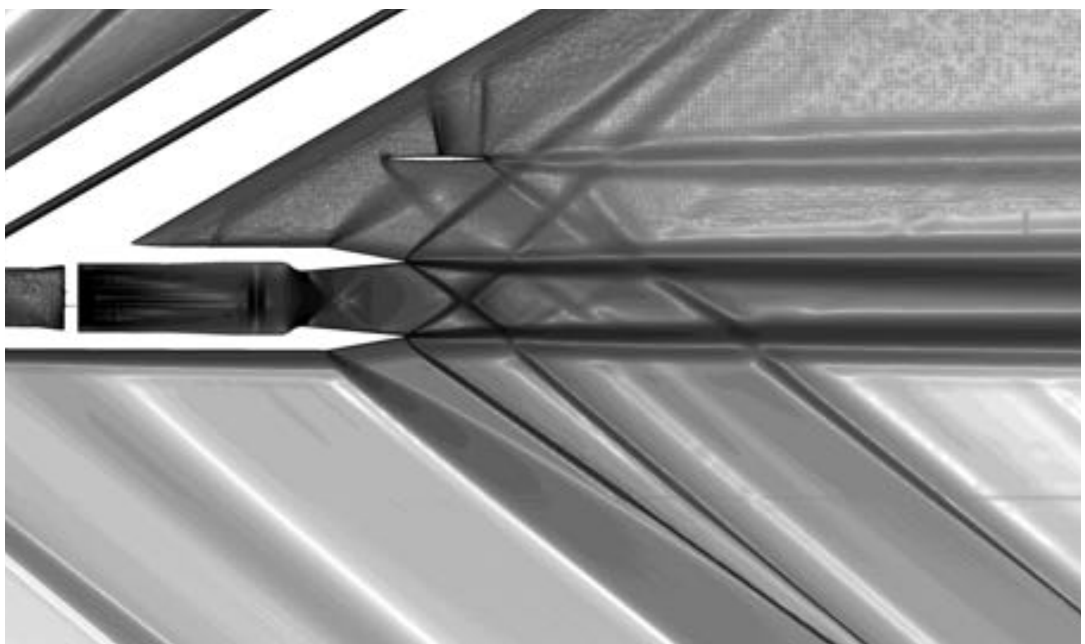
Mach Profiles for Simplified Plenum

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

- Using the plenum BC provides a closer match to the experiment than the simplified plenum BC using the same mesh

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 create an ascii file with the averaged radial profile
  data columns should be in order: radius, static P, static T, velocity normal to the plane of the nozzle Uz, and radial velocity Ur, e.g. in nozzle.dat:

```
# r  P  T  Uz  Ur
0.000000e+00  1.934005e+05  2.443122e+02  2.200106e+02 2.892030e+00
6.892365e-05  1.933989e+05  2.443072e+02  2.200447e+02 2.891260e+00

...
```

In the input file, specify the BC with the file name, nozzle center point and normal vector:

Radial BC specification

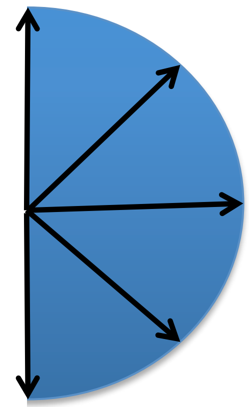
Extract cut lines from the full flow path solution at the desired location of the inlet BC (e.g., 1 ray at every 45°), and create an ascii file with the averaged radial profile

data columns should be in order: radius, static P, static T, velocity normal to the plane of the nozzle Uz, and radial velocity Ur, e.g. in nozzle.dat:

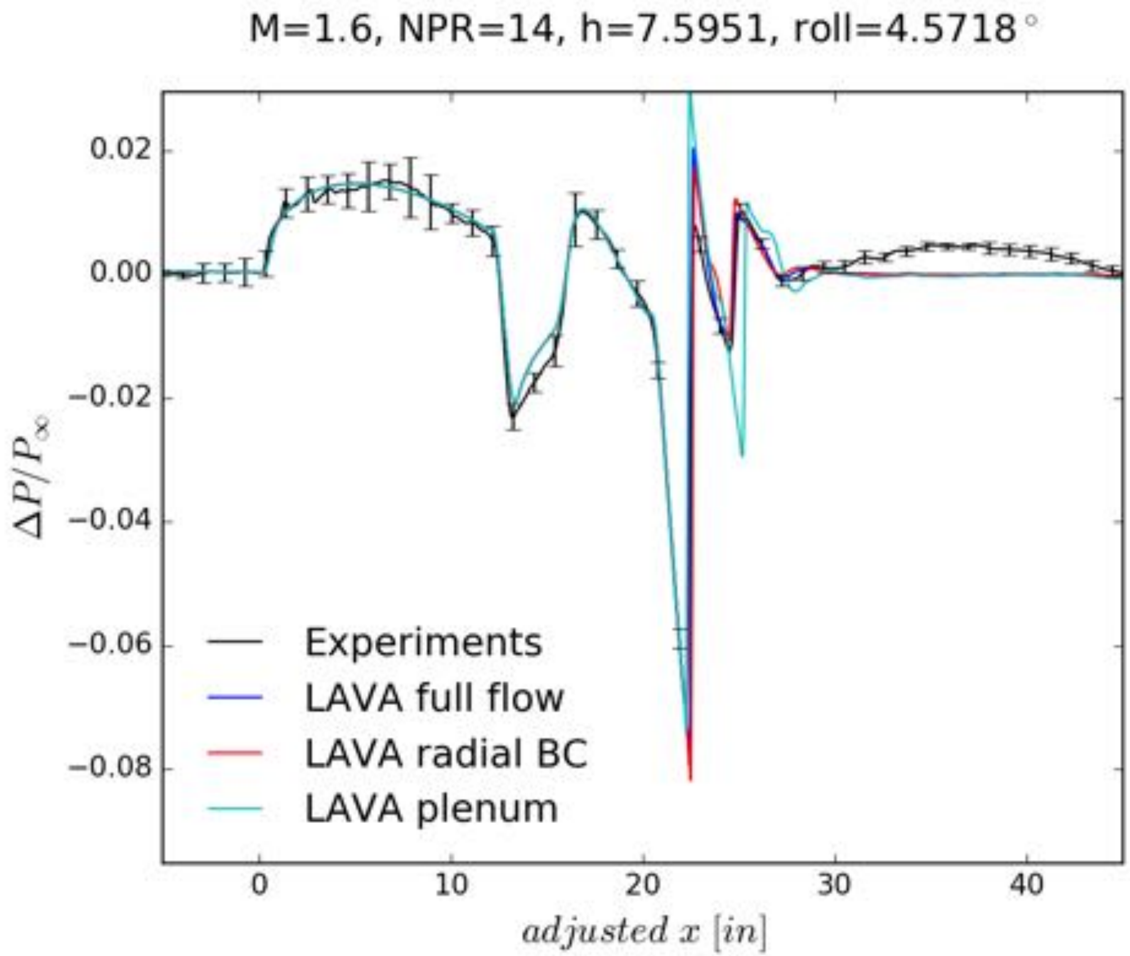
```
# r  P  T  Uz  Ur
0.000000e+00  1.934005e+05  2.443122e+02  2.200106e+02 2.892030e+00
6.892365e-05  1.933989e+05  2.443072e+02  2.200447e+02 2.891260e+00

...
```

In the input file, specify the BC with the file name, nozzle center point and normal vector:

```
BC_1  {type = nozzle
       datafiletype = 1
       filename   = nozzle.dat
       nozzlecenter = [0.5,0,0]
       nozzlenormal = [1,0,0]
       }
```

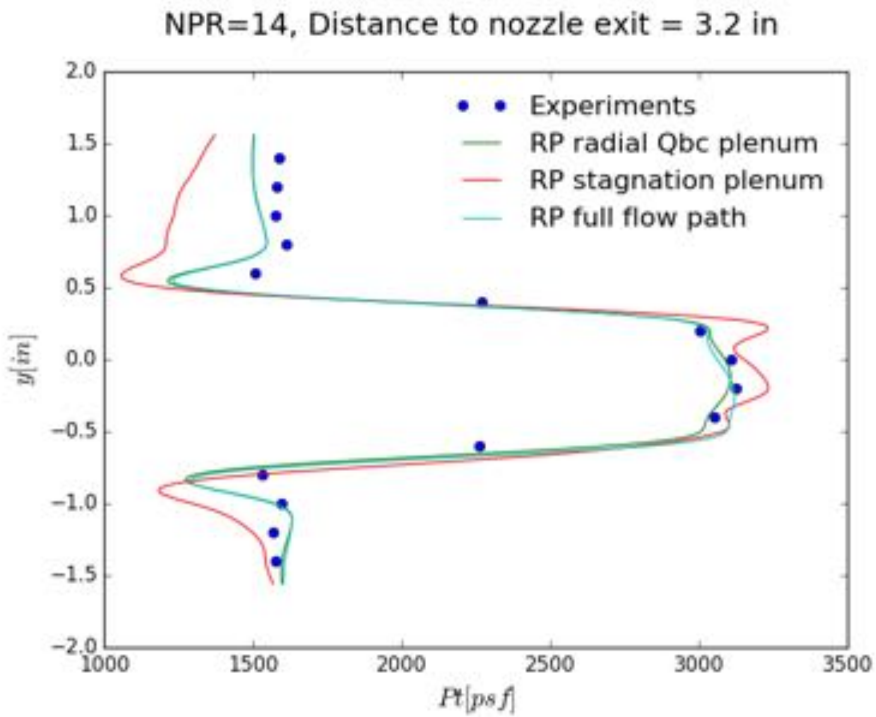


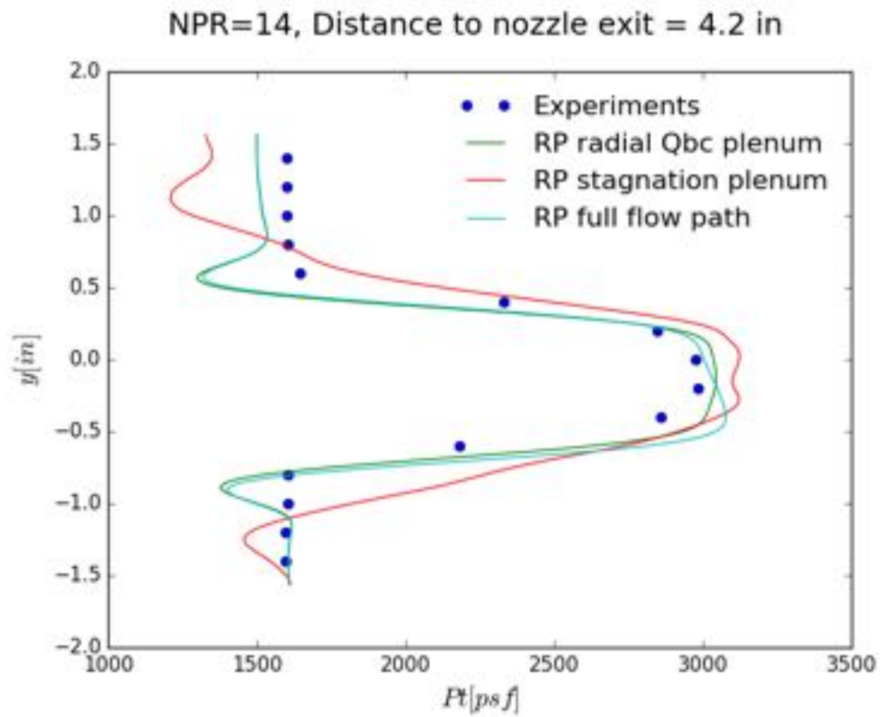
Radial BC Signature Comparison (nozzle only)



- 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)

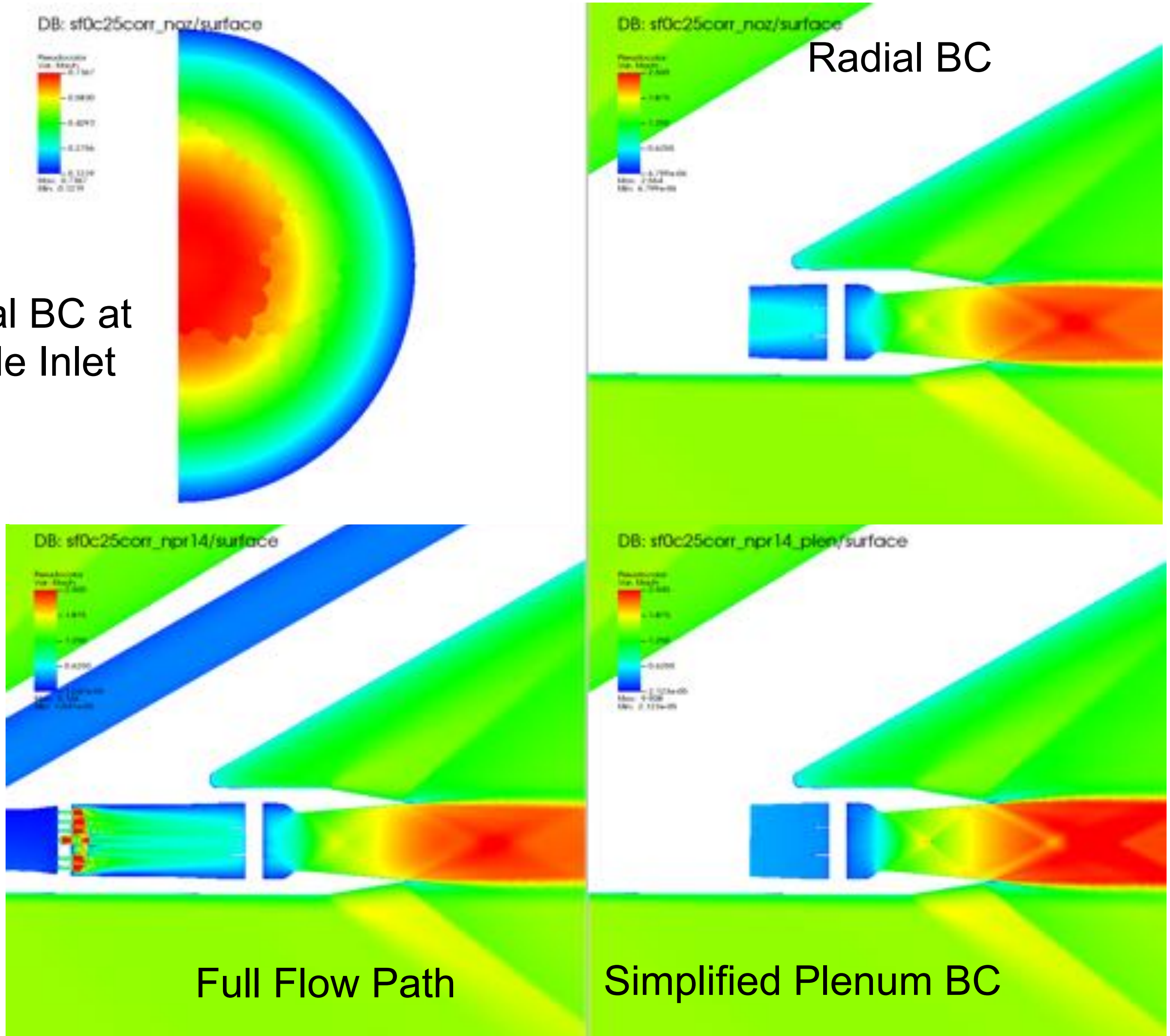




- Satisfying agreement between radial BC and full flow path simulations.

Mach Contour Comparison (nozzle only)

Radial BC at Nozzle Inlet



Summary and Conclusions

- The LAVA unstructured solver was successfully used to improve the experiment
- The results generated using the simplified plenum boundary condition and current meshing practices compared well to the experimental data
- 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

Future Work

- Simulations including the full rake geometry
- Full span solution computations for plume shear wall reflections
- Investigate the effects of the tunnel on the signatures
- Generate solutions using the LAVA curvilinear solver with structured overset meshes
- Knowledge-based grid refinement studies possibly using LAVA structured code

Acknowledgments

- Derek Dalle at Science and Technology Corp. in the Computational Aerosciences Branch at NASA Ames Research Center (ARC)
- 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