Comparison of Experimental Surface and Flow Field Measurements to Computational Results of the Juncture Flow Model

Nettie H. Roozeboom

NASA Ames Research Center

Henry C. Lee

Science and Technology Corp.

Laura J. Simurda, Gregory G. Zilliac, and Thomas H. Pulliam

NASA Ames Research Center

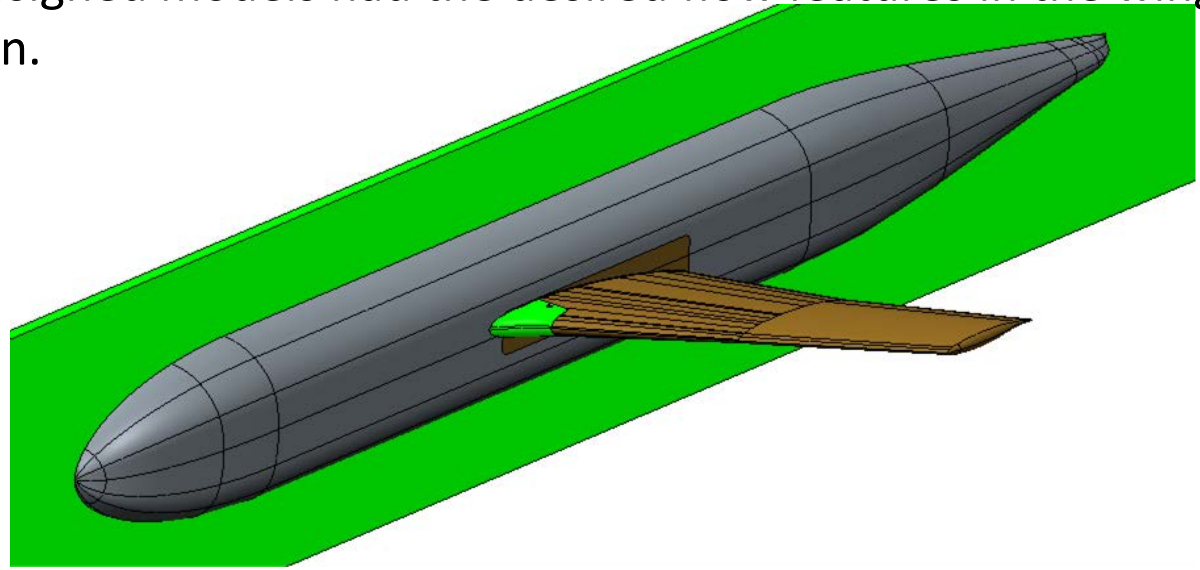
Transformative Aeronautics Concepts Program

Transformational Tools and Technologies (T^3)

Juncture Flow Effort

Sponsored by NASA's Transformative Aeronautics Concepts Program's Transformational Tools and Technologies (T^3) effort

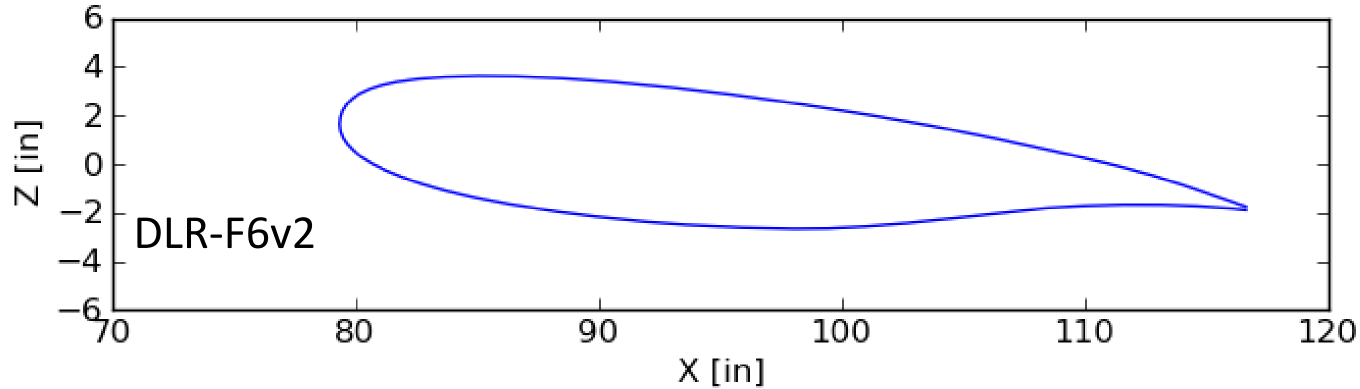
- Substantial effort to investigate the origin of separation bubbles found in wing-body juncture zones.
- Multi-year effort including several large-scale wind tunnel tests
- 2 years of designing model using Computational Fluid Dynamics (CFD)
- Primary goal of early experiments was to gather data demonstrating the CFD-designed models had the desired flow features in the wing-body junction.



Juncture Flow Effort

- CFD used to design candidate geometries

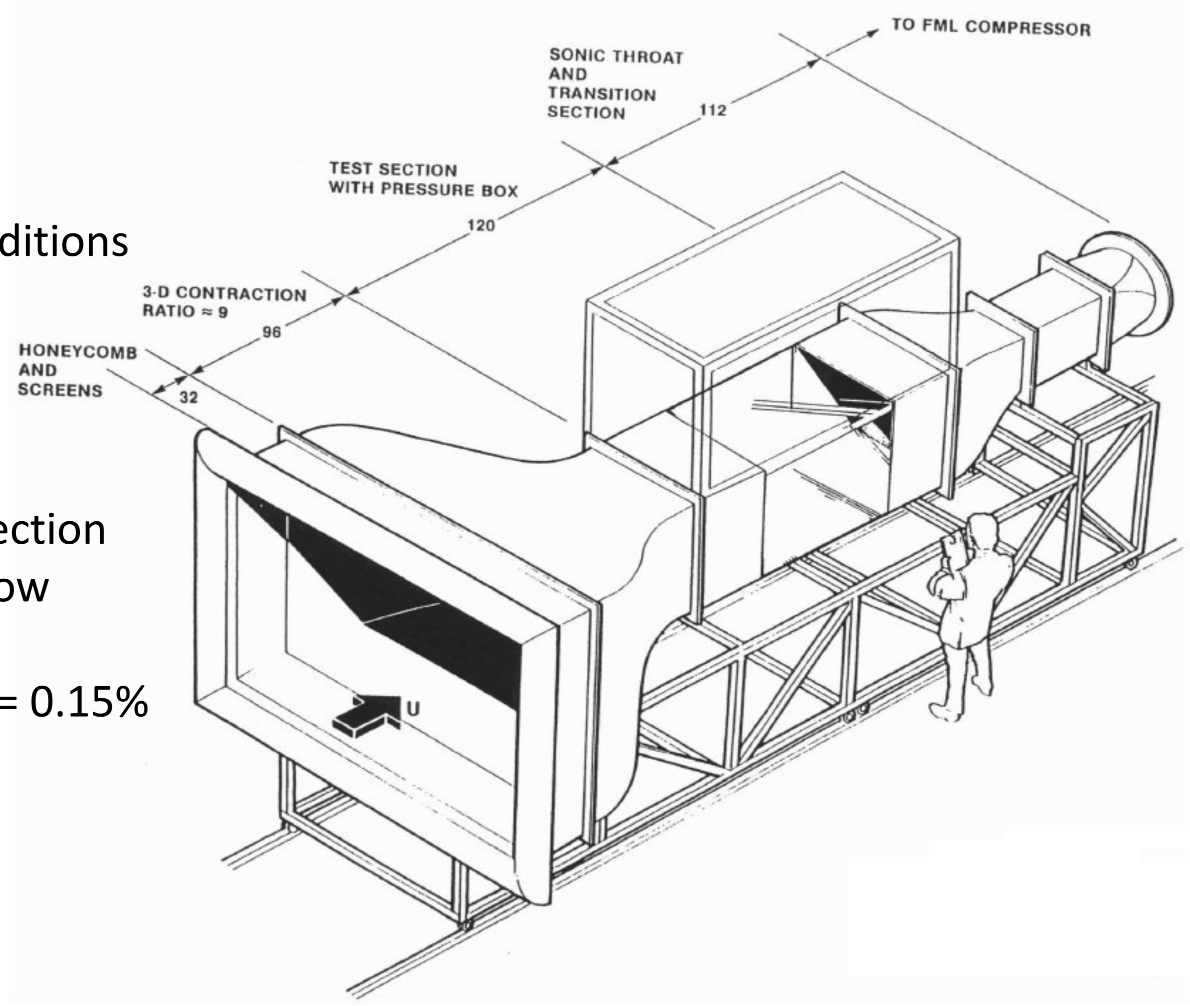
  (no separation–incipient separation–fully separated)
- **Risk reduction tests** ran to guide JF committee plan future tests.
  - Low-cost / quick turn around to give first look at experimental data
  - Highlighted differences between computation and experimental results.
- JF committee interested in testing a wall-mounted model – gather results to quantify the effect of tunnel wall b.l.
- Needed to understand influence of tunnel wall b.l. on JF region.



Re = 0.62 Million
0 deg AOA
5% trips
With Horn

Fluid Mechanics Lab Test Cell 2 (TC2)

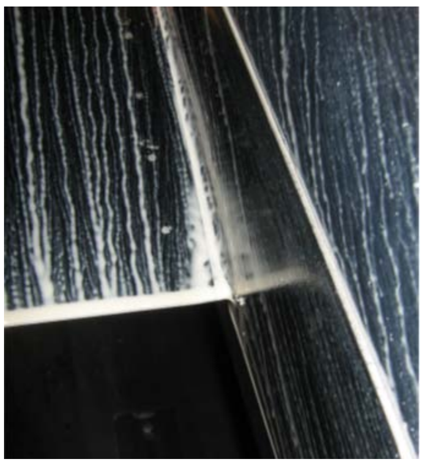
- JFM Tunnel Conditions
  - Tunnel Velocity = ~145 ft/s
  - Re = 0.620 Million
  - Matched VA Tech tunnel conditions
- Tunnel Specifications
  - Honeycomb, 3 screens
  - 9:1 Contraction Ratio
  - 48”W x 32” H x 120” L test section
  - Sonic throat controls mass flow
- Free-stream turbulence intensity = 0.15%
- Core angularity less than 0.3˚



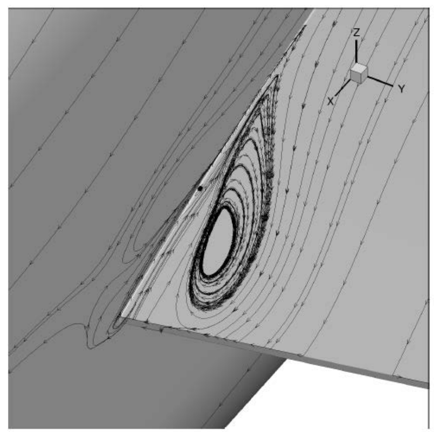
Address Challenges

- Delivered data to the JF community
  - Correlation between CFD and Experimental Fluid Dynamics (EFD) obtained in TC2 were not as strong as the correlation between the results from CFD and VA Tech results.
  - Bubble sizes were different.

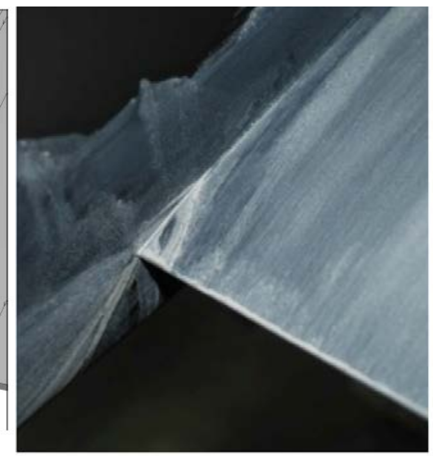
TC2



Overflow



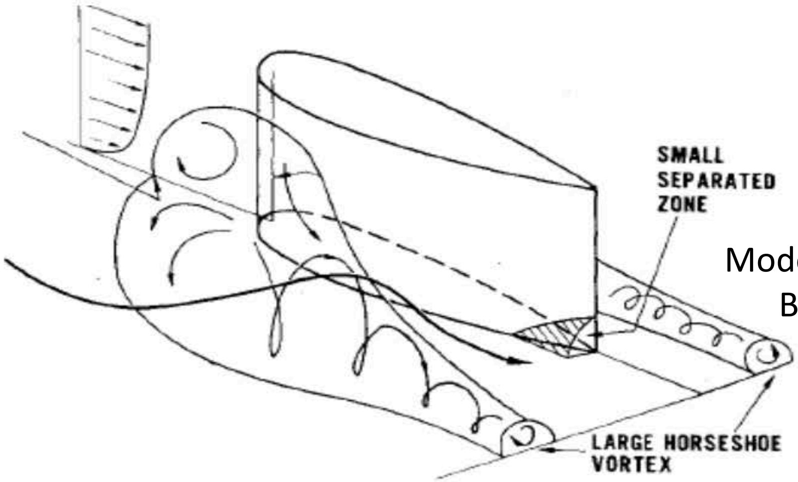
VA Tech



- Started exploring some of the differences between the computation and experiment.
  - Sting-mounted vs. wall mounted
  - Do not see the same size separation between VA Tech results or CFD

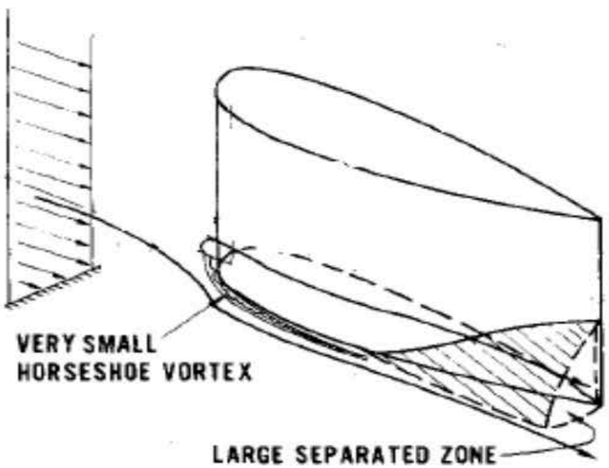
Address Challenges

- Wing-body junction challenging to compute accurately.
  - Turbulent boundary layers merge and form a horseshoe vortex (HSV)
  - Off surface flow is highly three dimensional
  - Trailing edge junction is difficult to compute and measure
- Stronger influence of incoming wall boundary layer with a wall-mounted model



(a) thick boundary layer

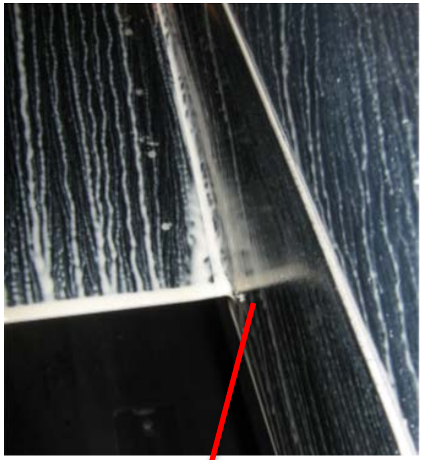
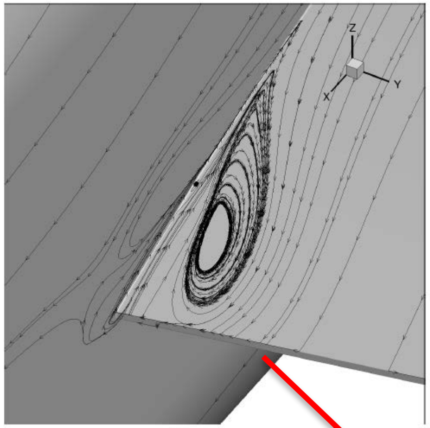
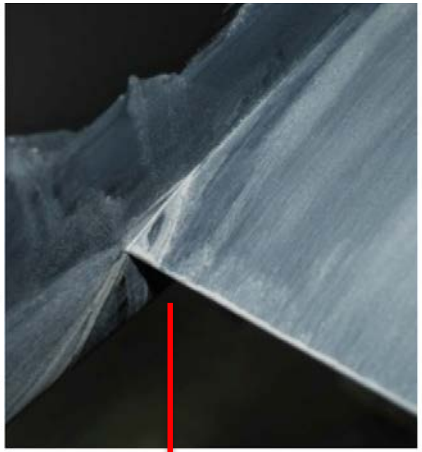
Model proposed by Barber *et al.*

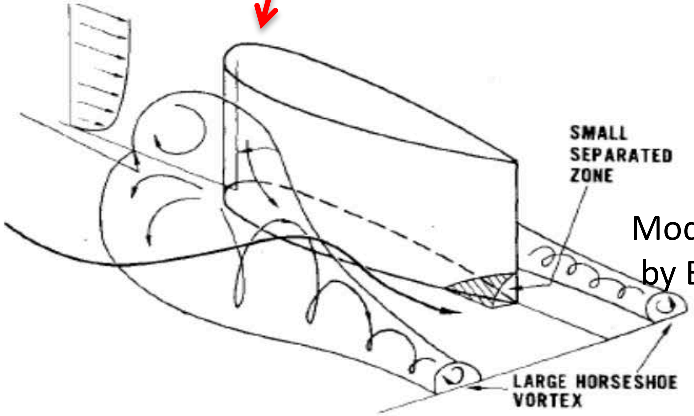


(b) thin boundary layer

- Gand *et al*, Barber *et al*, Simpson: to correctly compute JF, both HSV and wing b.l. have to be captured accurately.

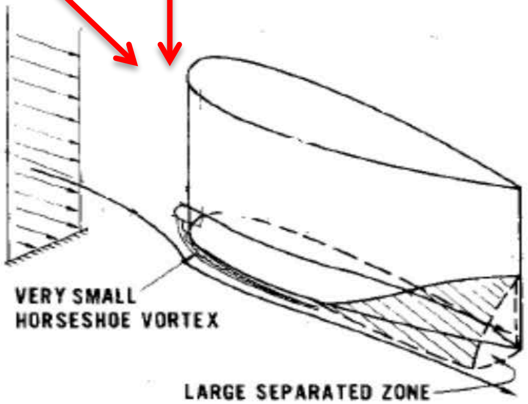
Address Challenges

TC2	Overflow	VA Tech
		



(a) thick boundary layer

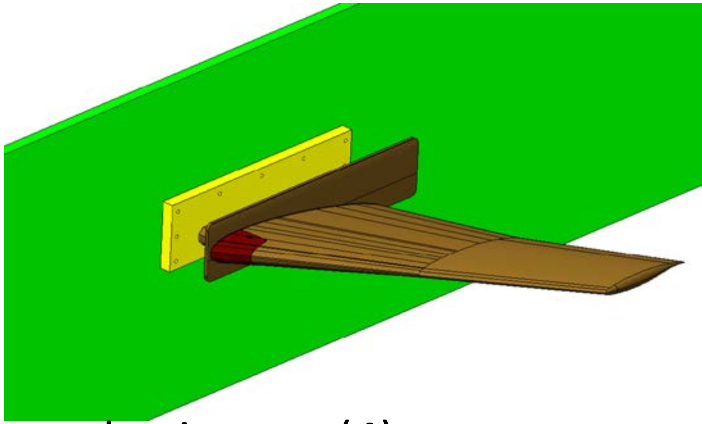
Model proposed by Barber *et al.*



(b) thin boundary layer

Why do we not get separation on wall-mounted model?

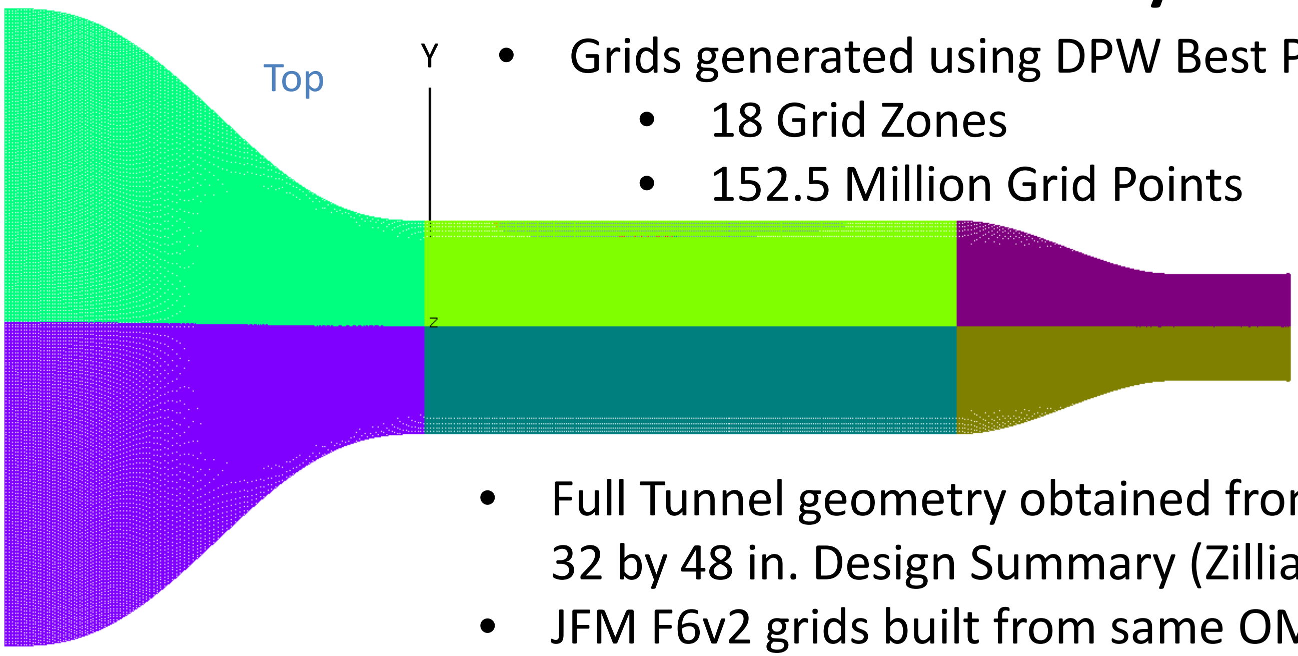
Experimental Setup



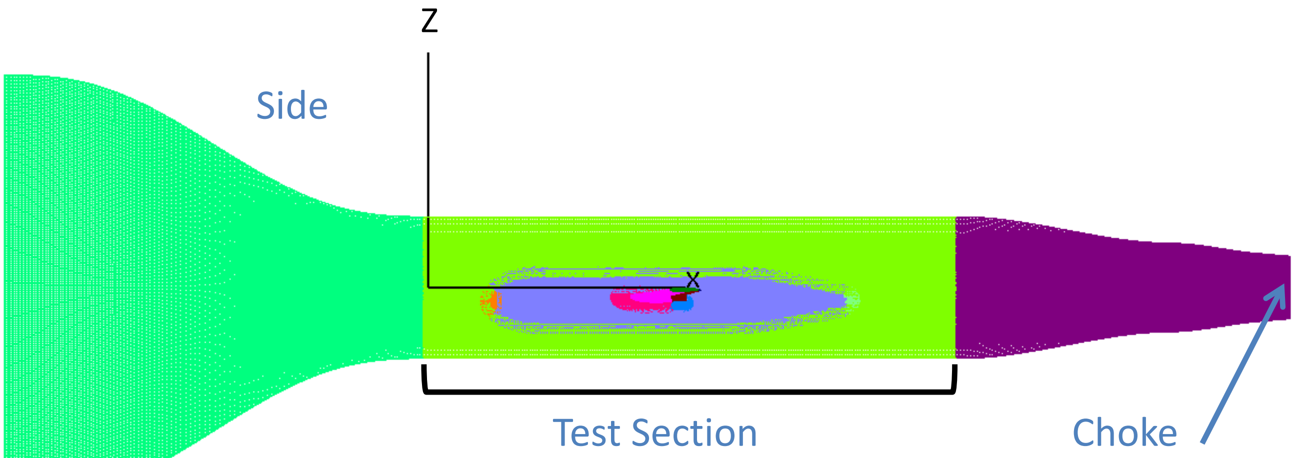
- 3%-scaled semi-span
- Uniform junction and repeatable install
  - Modeling board fuselage / aluminum wing
  - Mounting plates for (0˚, 2˚, 4˚, 6˚, and 8˚)
- $25K to manufacture fuselage, 2 wings, and leading edge inserts (4)
- Tests investigated both surface and off-surface flow features in and around the wing-body junction
- Boundary layer surveys, oil flow vis, and skin friction measurements **(EFD)**

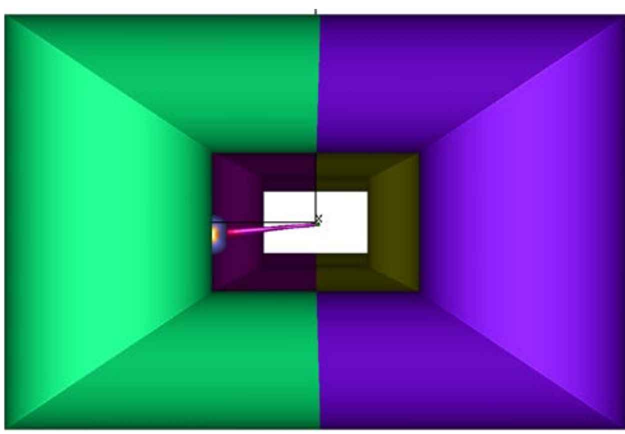
CFD Geometry

- Grids generated using DPW Best Practices
  - 18 Grid Zones
  - 152.5 Million Grid Points



- Full Tunnel geometry obtained from the 32 by 48 in. Design Summary (Zilliac) Memo
- JFM F6v2 grids built from same OML used to manufacture the model

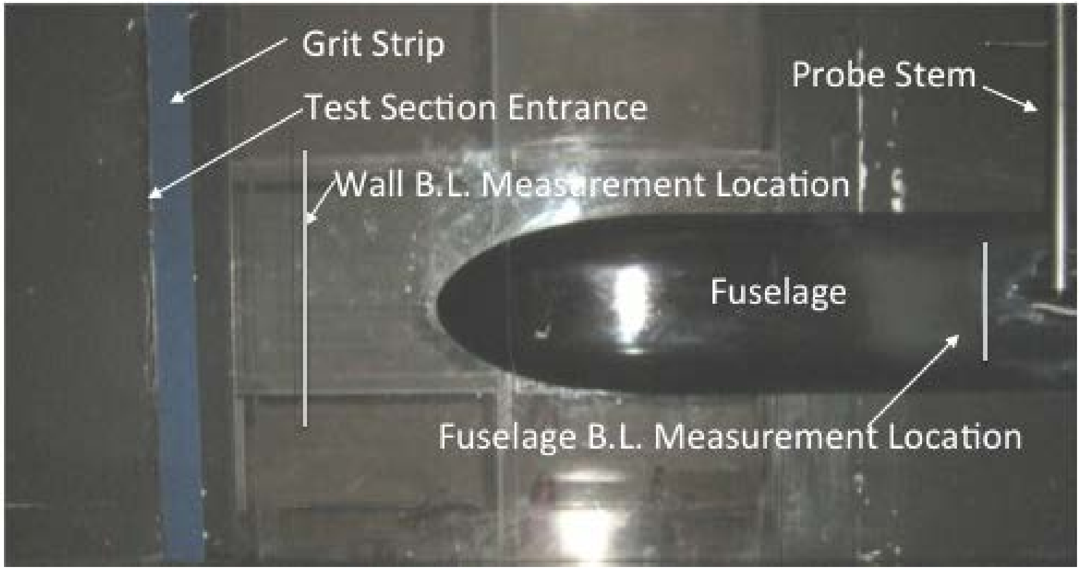


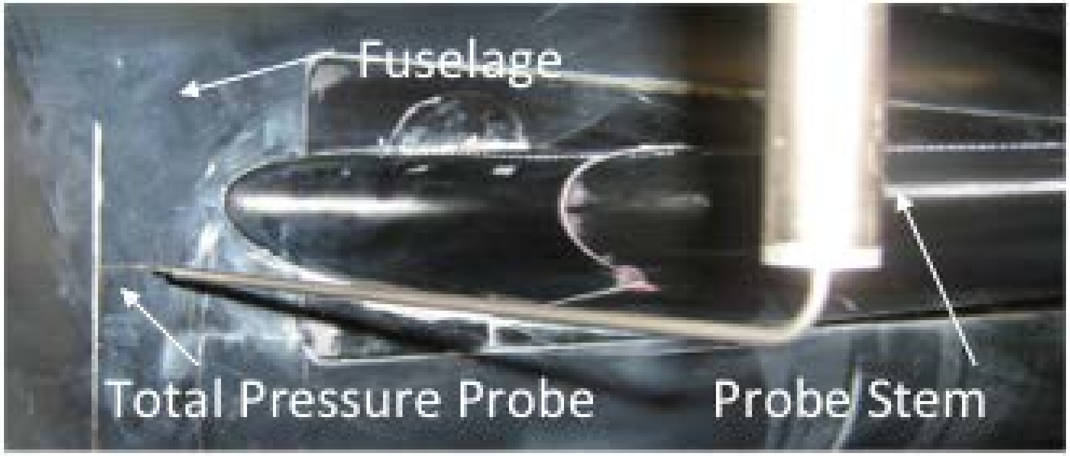


CFD Simulation Setup

- Overflow 2.2k
- Spalart Allmaras (SA) Turbulence Model
  - Rotational correction on
  - Quantitative Constitutive Relations (QCR) on
- Roe upwind, ARC3D diagonalized Beam-Warming scalar pentadiagonal scheme
- Boundary Conditions
  - Inlet: set stagnation Pressure and Temperature
  - Exit: vary back pressure ratio
    - Choke in diffuser (near exit)
    - Speed in tunnel at reference station matches WT

Boundary Layer Surveys

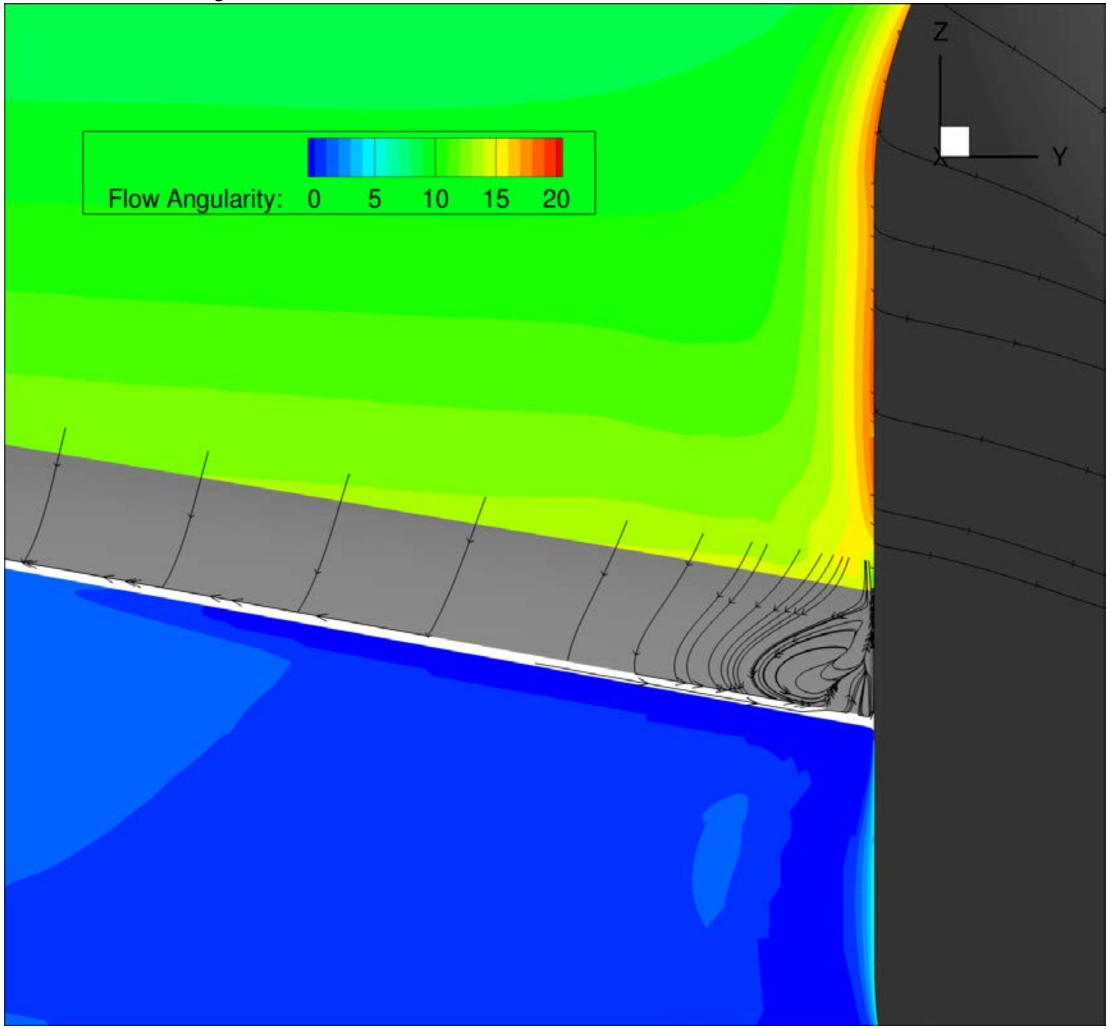




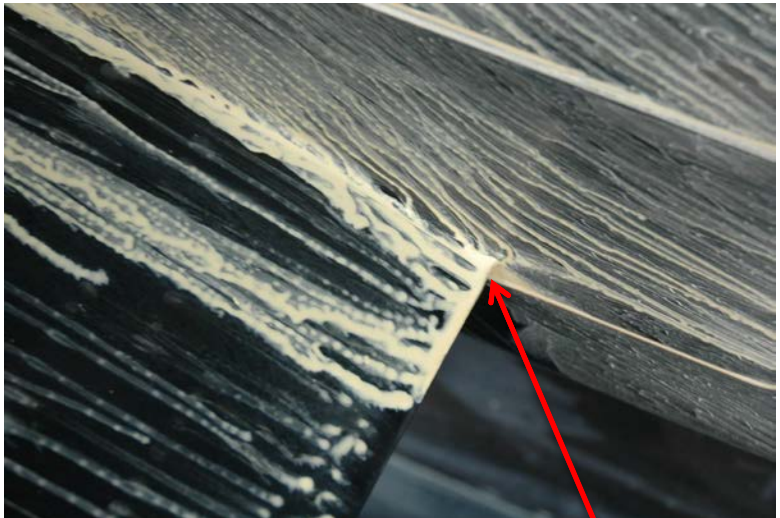
- United Sensors conical 0.025” diameter total pressure probe
- Probe help by a 0.75” diameter probe stem extension attached to a three axis traverse system.
- Time-averaged data samples taken for 15 second at 1000 Hz.
- High values of flow angularity would compromise velocity measurement.

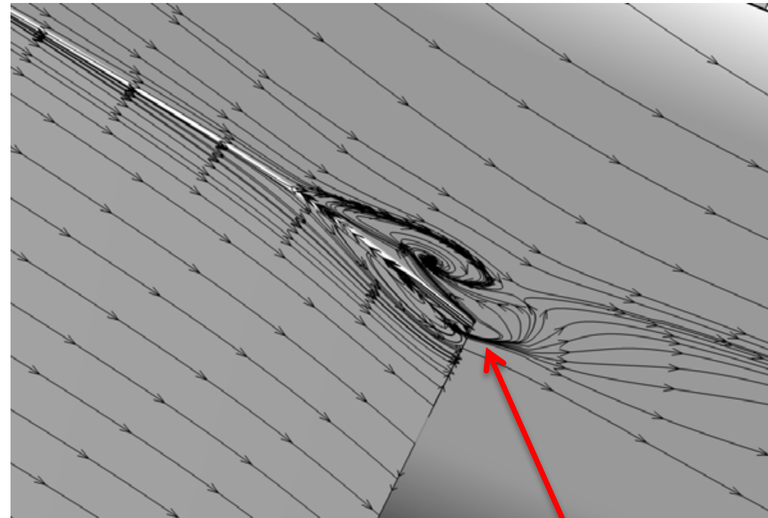
Flow Angularity

- Used CFD to investigate flow angularity.
- Probe measurement may need to be adjusted.
- For 0.025" diameter conical probe, total pressure will be within 1% of actual total pressure for flow angles up to 15-20˚
- Vast majority of data not impacted but measurements downstream of trailing edge, measured velocity will be less than actual velocity.



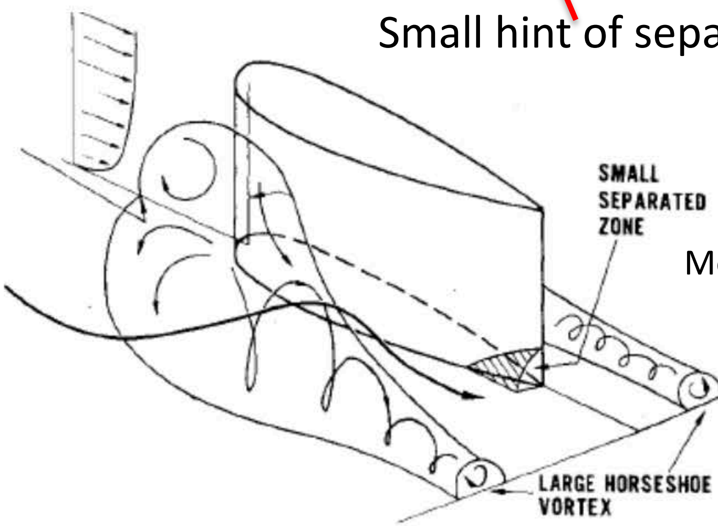
Results – Oil Flow





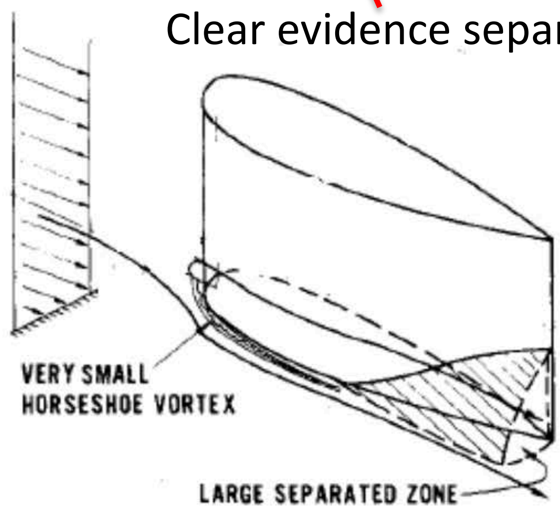
Small hint of separation

Clear evidence separation



(a) thick boundary layer

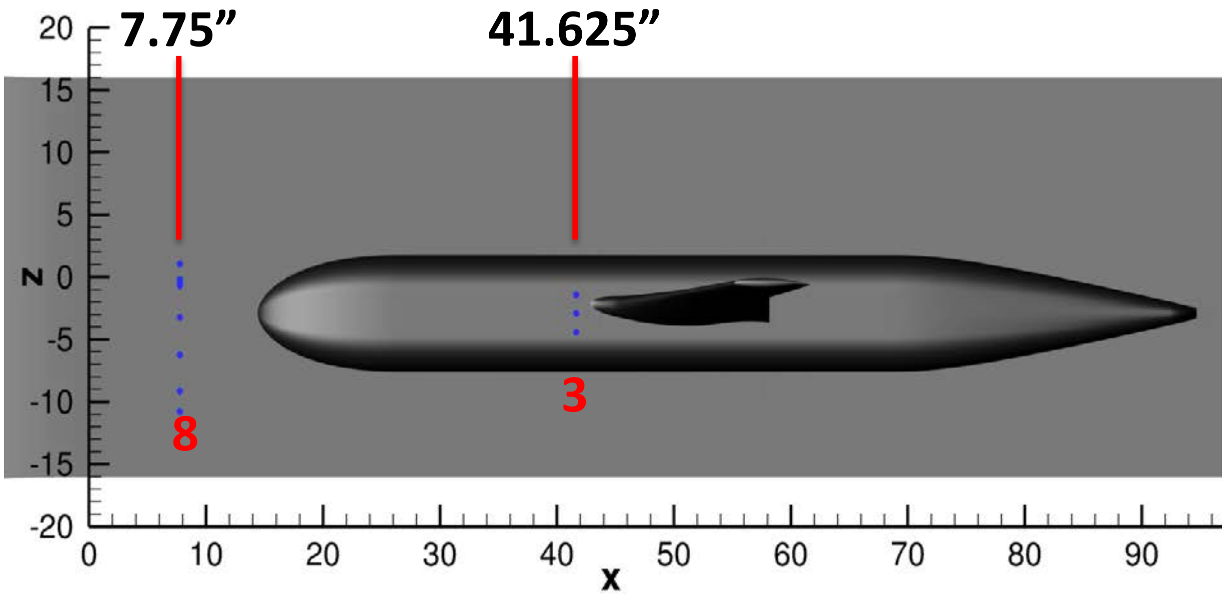
Model proposed by
Barber *et al.*



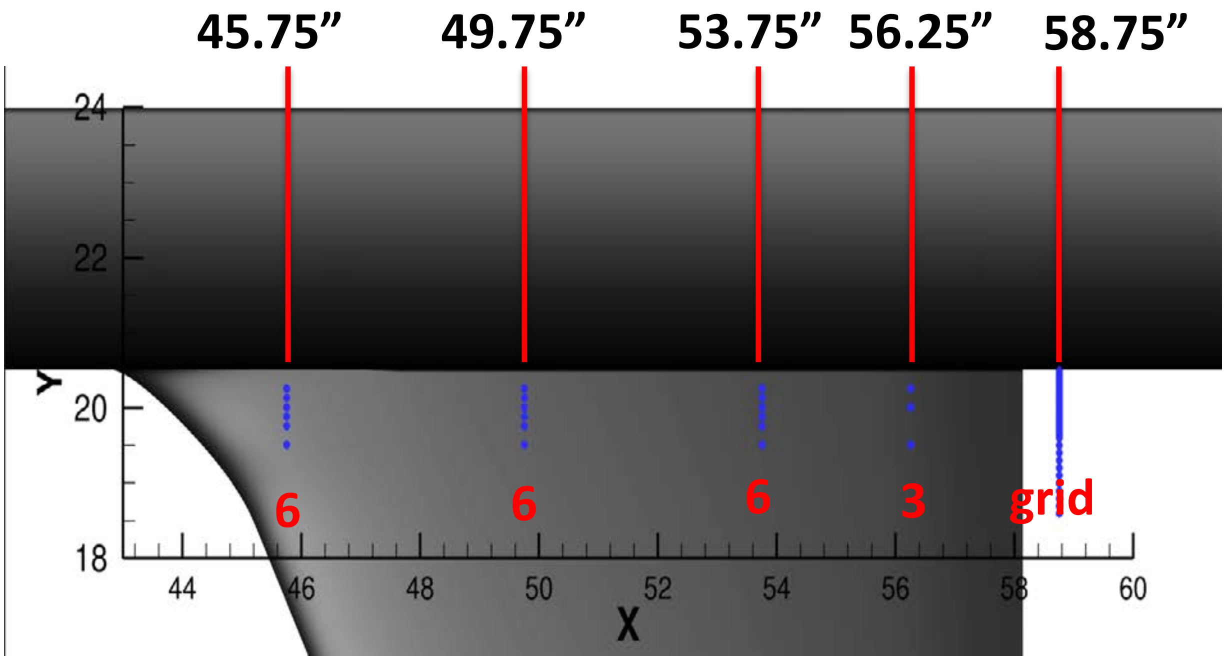
(b) thin boundary layer

- Both horseshoe vortex and wing boundary layer have to be captured accurately.
- Slight difference between EFD / CFD highlight modeling or measurement deficiencies.

Boundary Layer Surveys

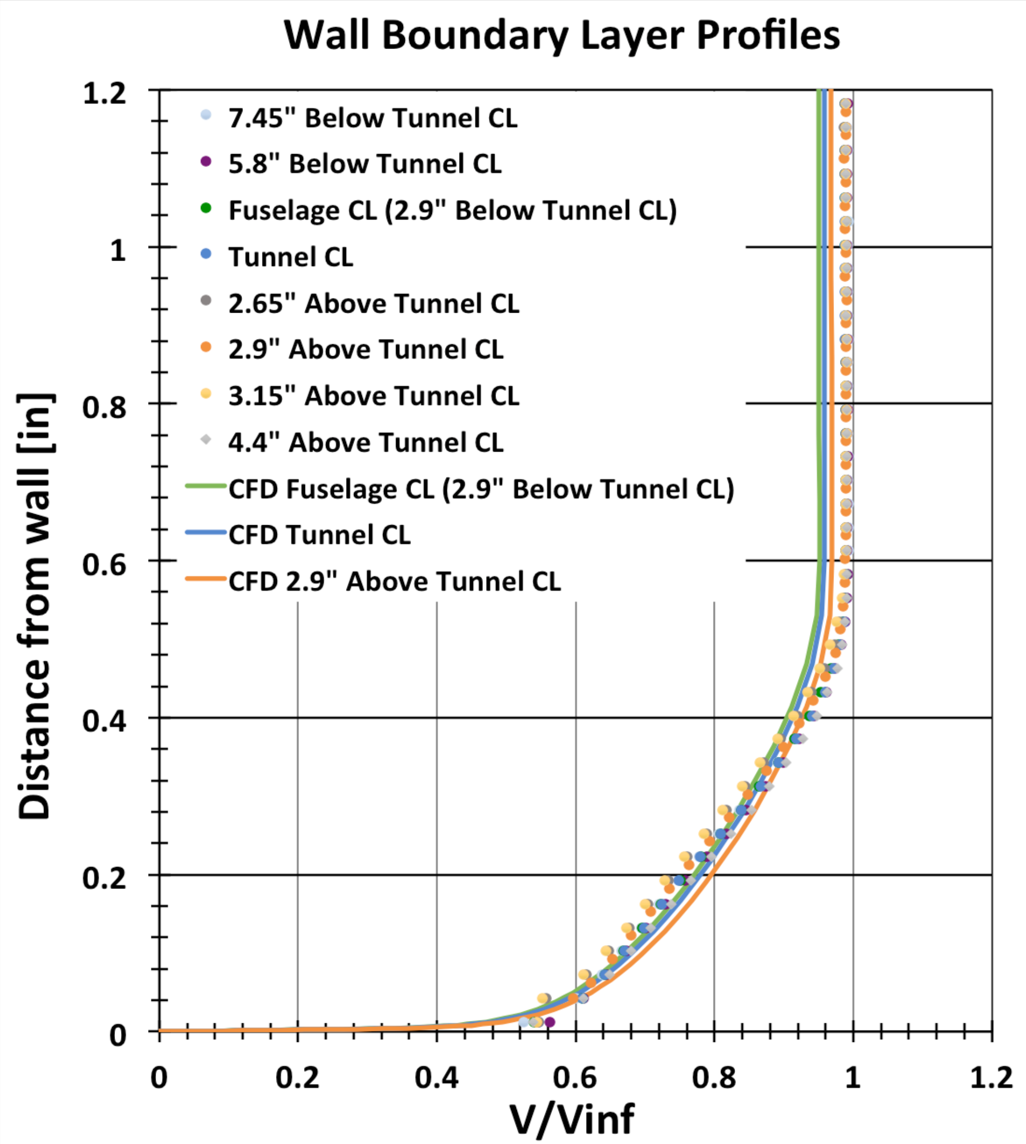


Traversed probe away from the wall / fuselage wall (Y- direction)



Traversed probe away from the wing surface (Z+ direction)

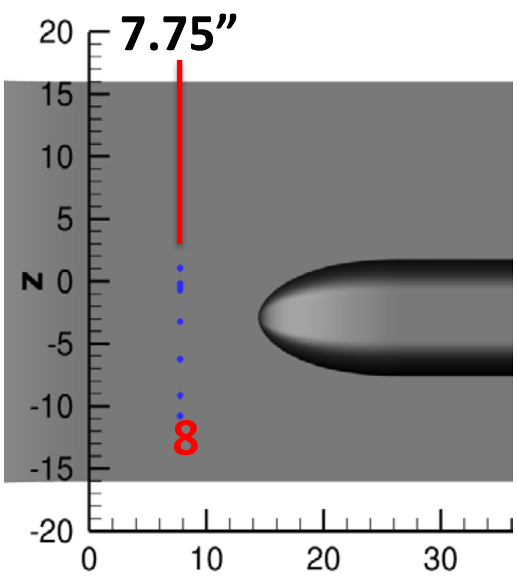
Results – Wall Boundary Layer



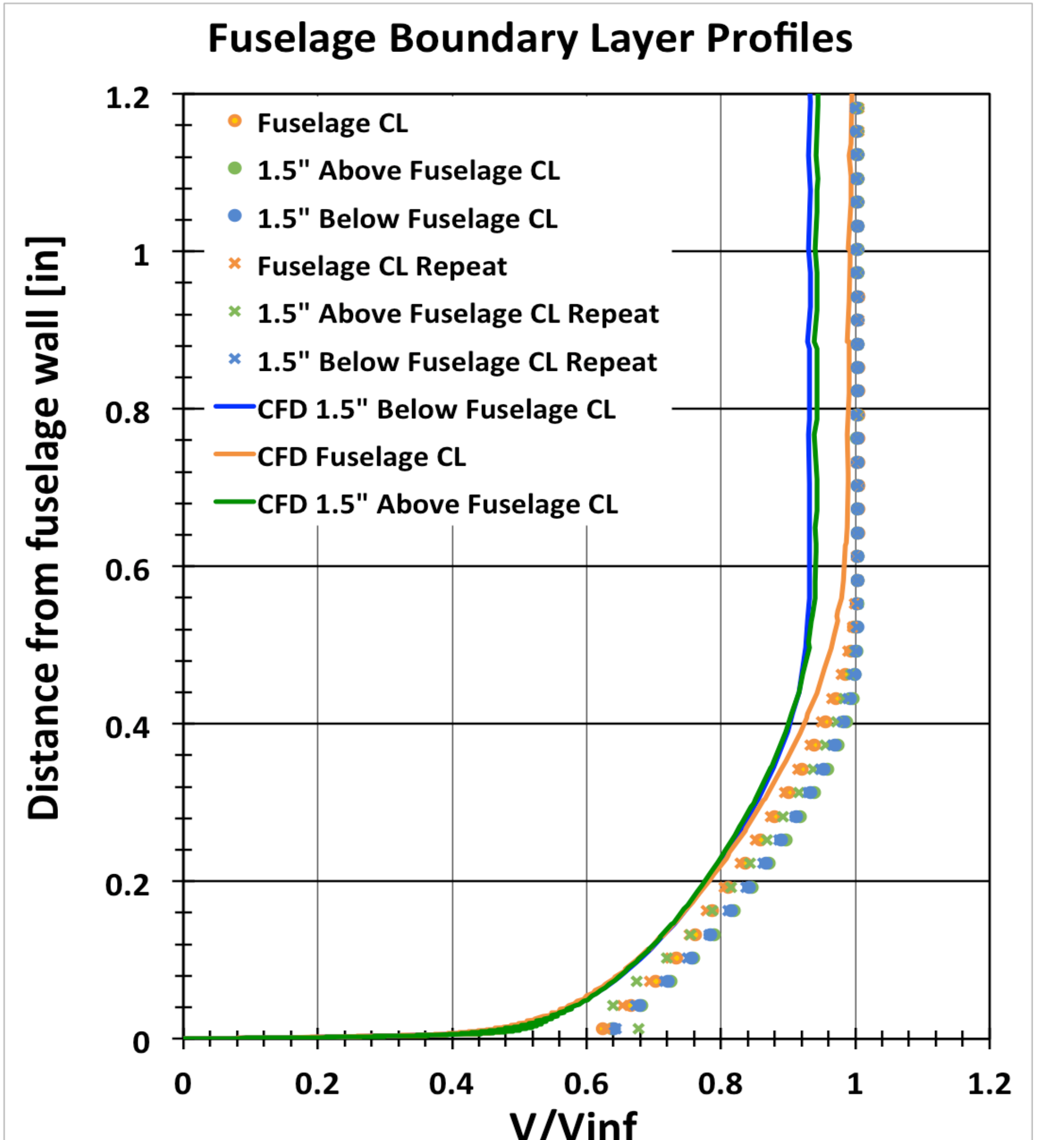
JFM F6v2 with horn, 0˚ AoA, Re = 0.620 Million, X = 7.75”

OBSERVATIONS

- Overall height comparable
- Overall shape is slightly different
- Minor differences in roughness / steps in WT, reducing experiments velocity
- CFD is seeing a stronger influence from model upstream of wing LE



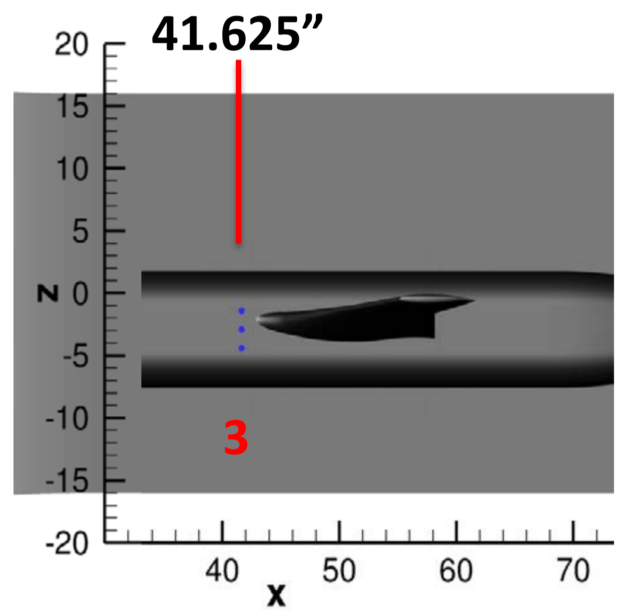
Results – Fuselage Boundary Layer



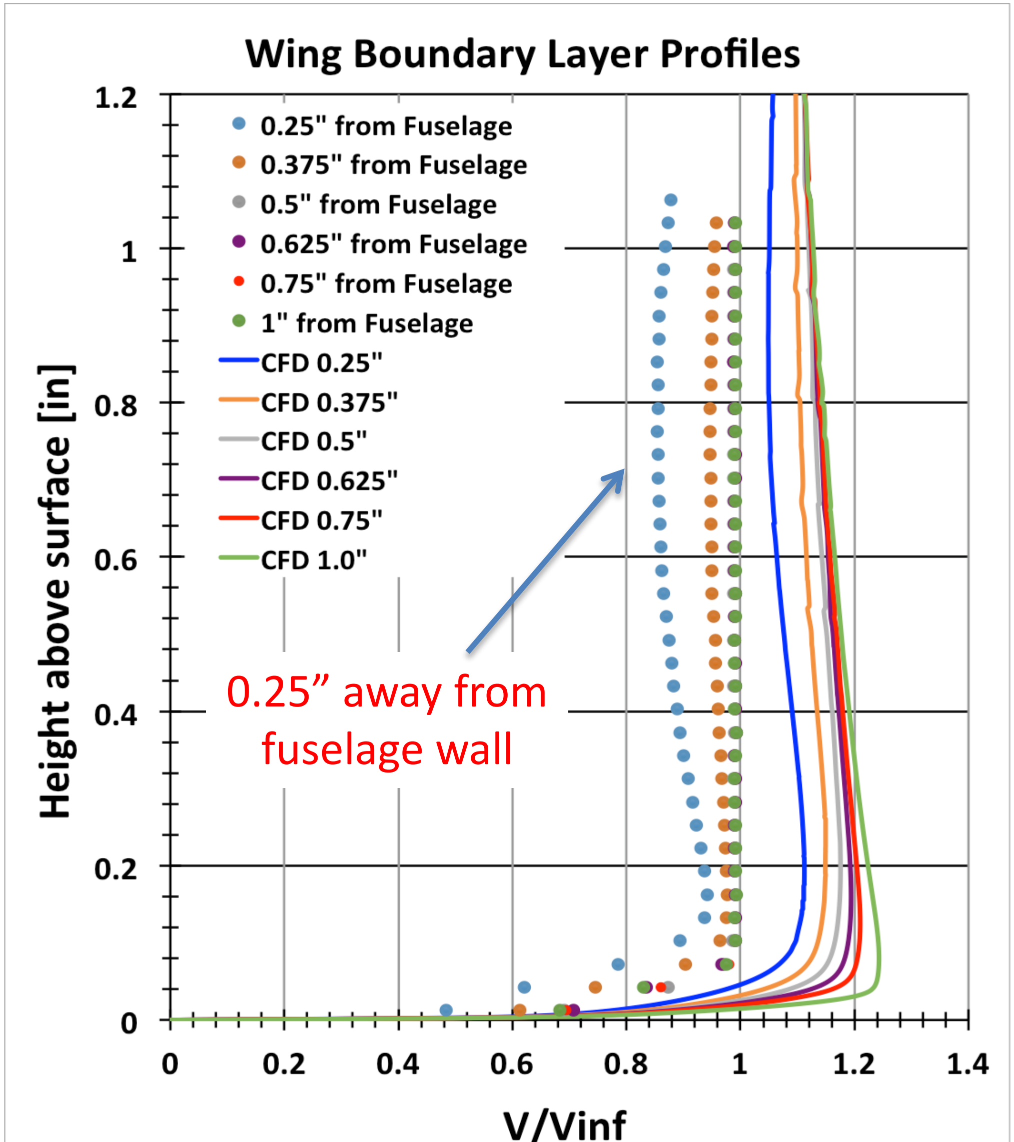
JFM F6v2 with horn, 0˚ AoA, Re = 0.620 Million, X = 41.625”

OBSERVATIONS

- Overall height comparable
- Overall shape is slightly different
- CFD is seeing a stronger influence from model upstream of wing LE



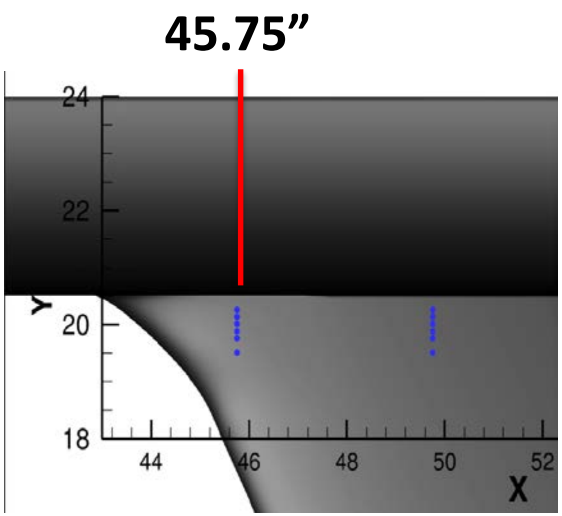
Results – Wing Boundary Layer



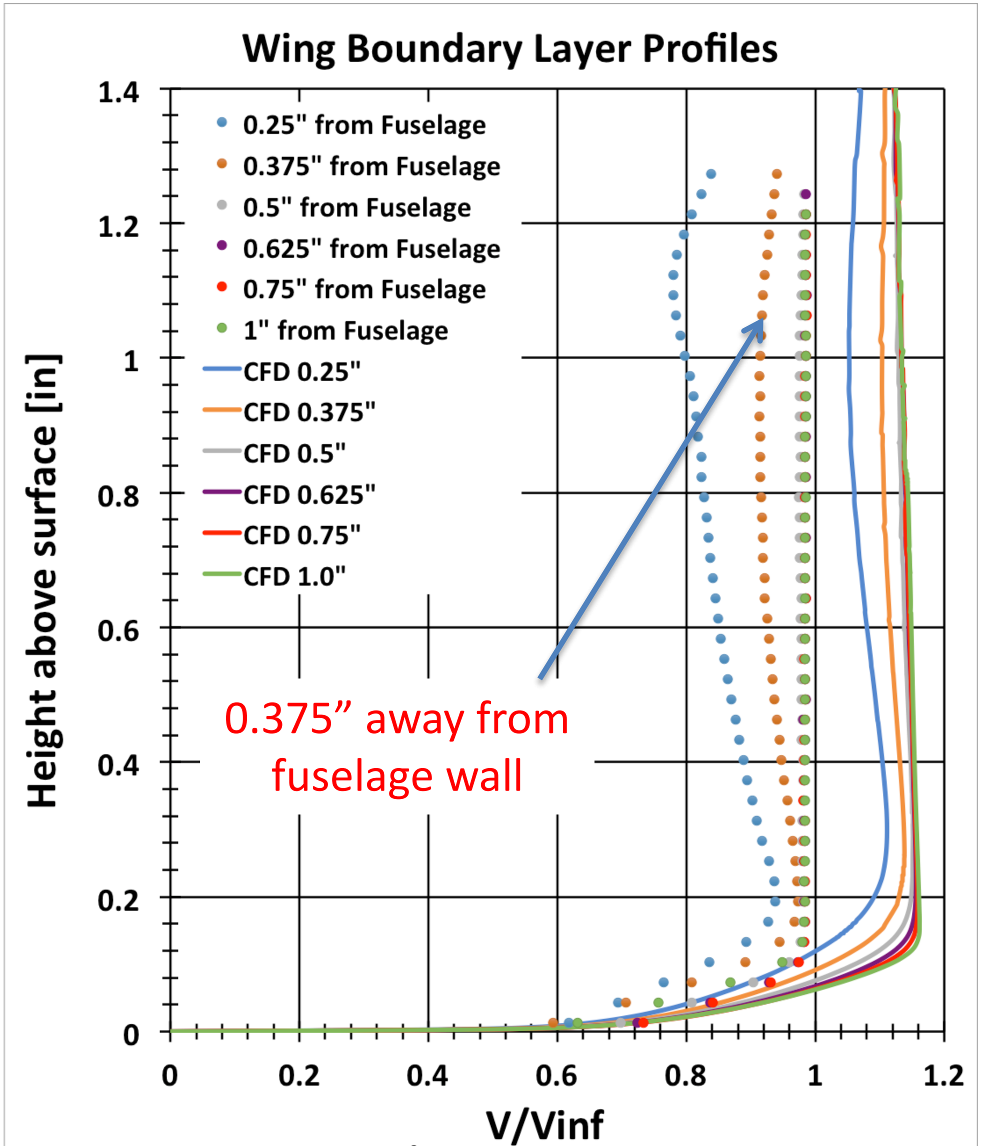
JFM F6v2 with horn, 0˚ AoA, Re = 0.620 Million, X = 45.75"

OBSERVATIONS

- EFD seeing larger presence of fuselage b.l.
- CFD thinner fuselage b.l.
- CFD shows increased velocities in the wing junction



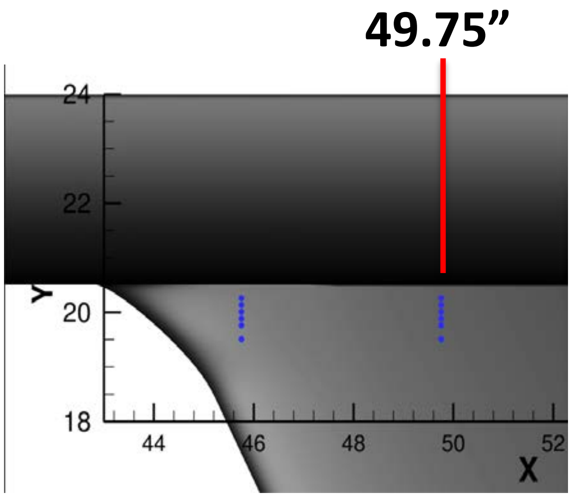
Results – Wing Boundary Layer



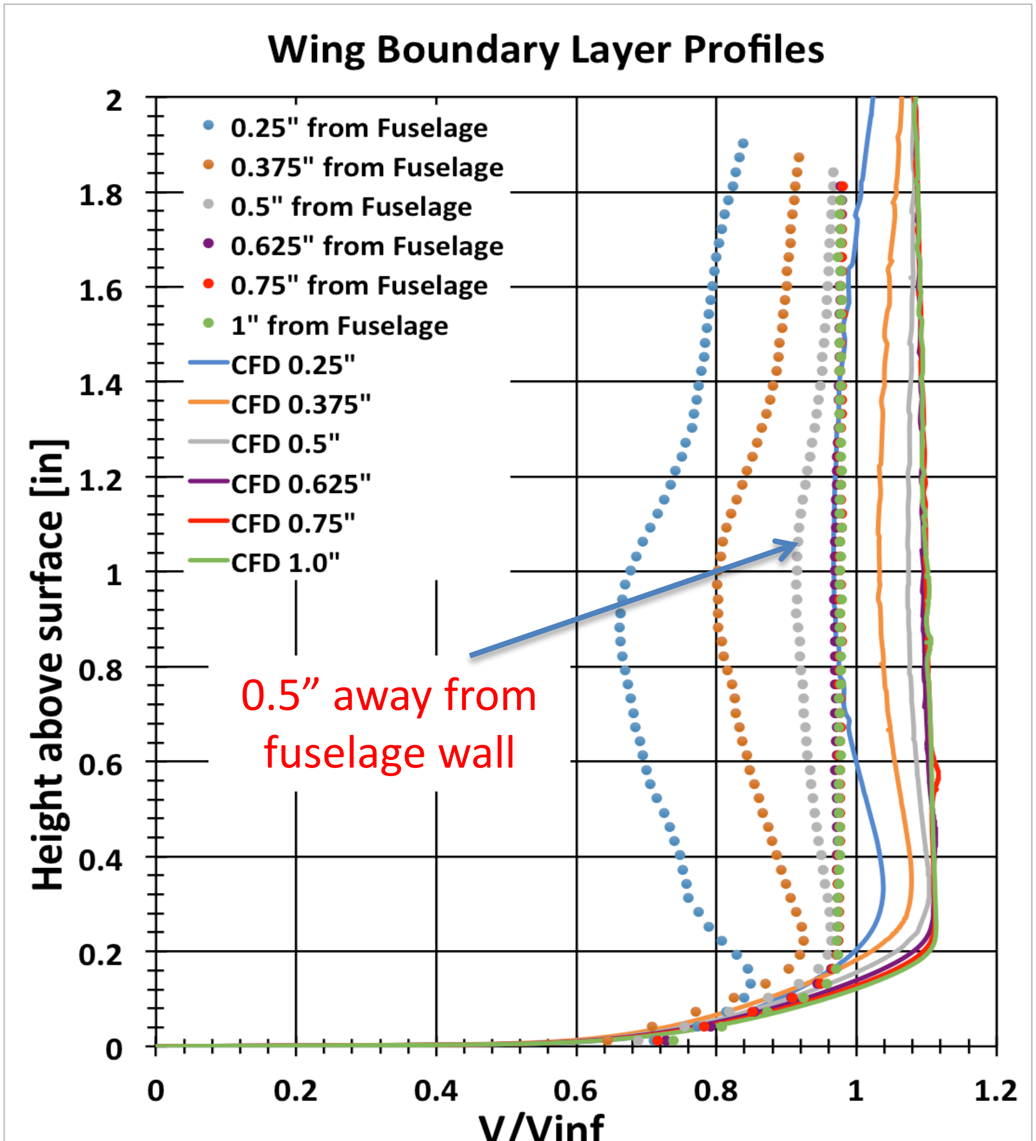
JFM F6v2 with horn, 0˚ AoA, Re = 0.620 Million, X = 49.75”

OBSERVATIONS

- EFD shows a dip in velocity 0.25” away from fuselage wall, 1.1” above wing.
- EFD seeing influence of fuselage b.l. 0.375” away
- CFD shows less influence from fuselage b.l.



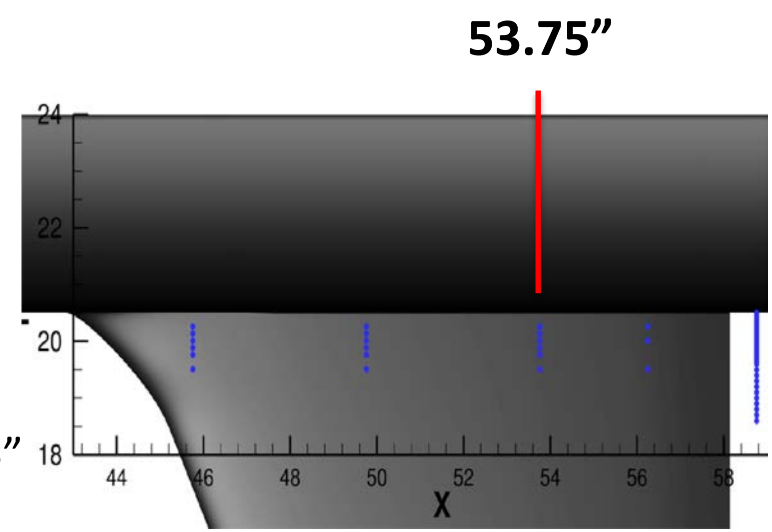
Results – Wing Boundary Layer



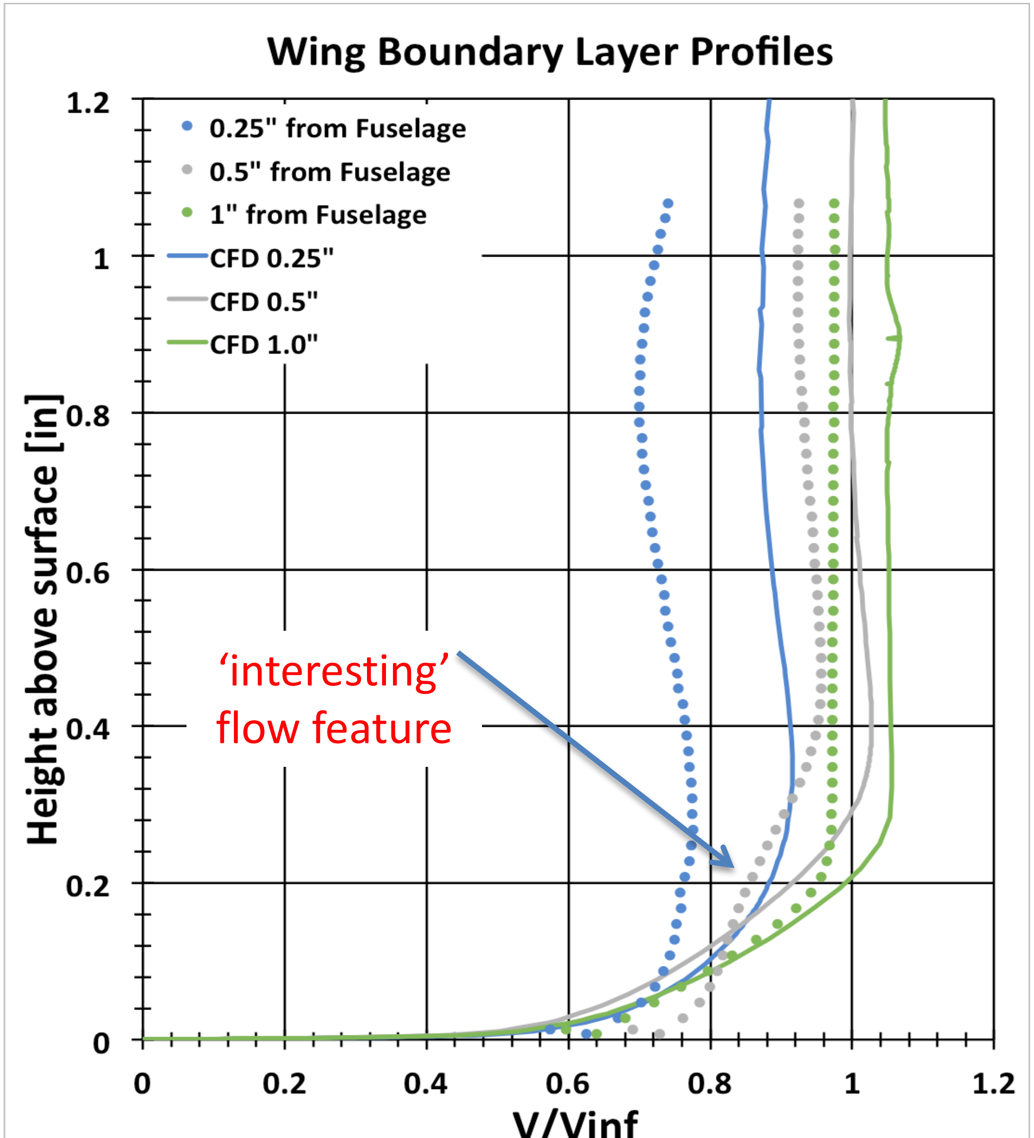
JFM F6v2 with horn, 0˚ AoA, Re = 0.620 Million, X = 53.75”

OBSERVATIONS

- EFD shows a dip in velocity 0.375” away from fuselage wall, 0.9” above wing.
- EFD seeing influence of fuselage b.l. 0.5” away
- CFD weak presence of fuselage b.l.
- CFD shows increase in speed over the wing.



Results – Wing Boundary Layer

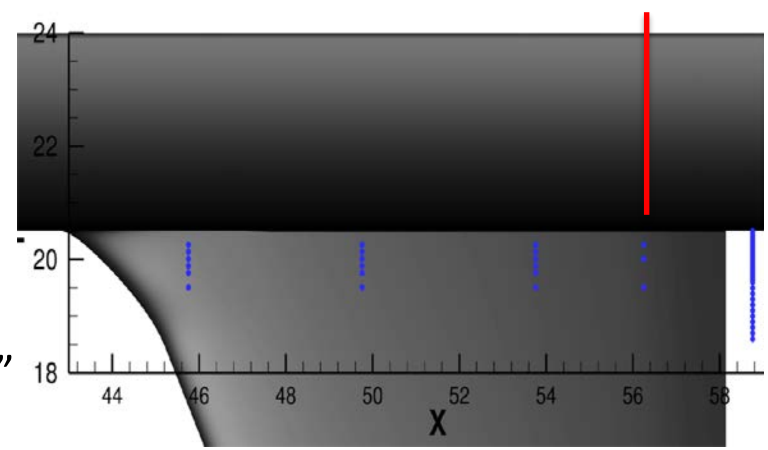


JFM F6v2 with horn, 0˚ AoA, Re = 0.620 Million, X = 56.25”

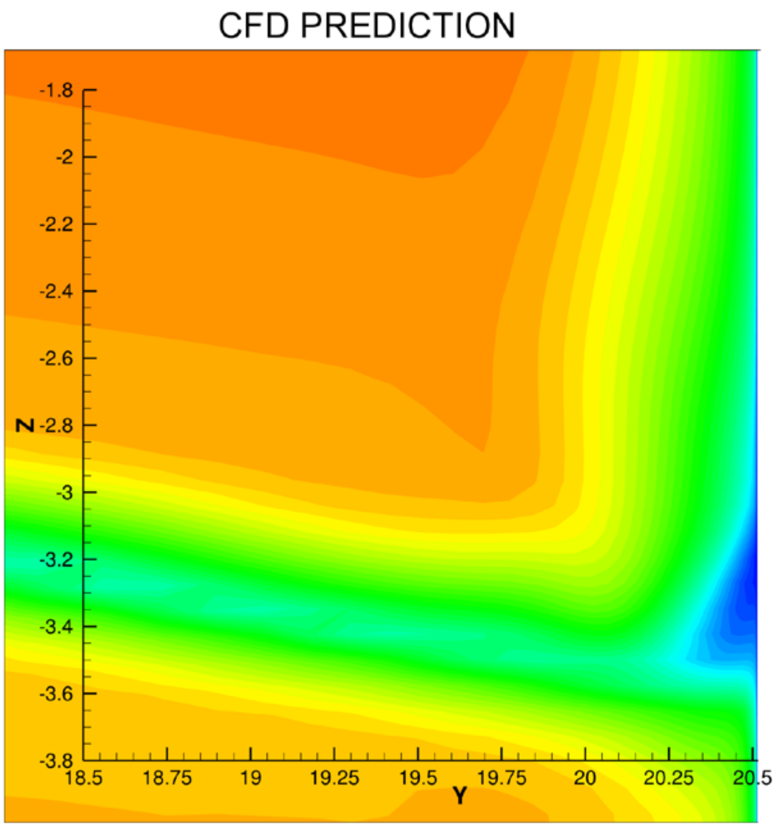
OBSERVATIONS

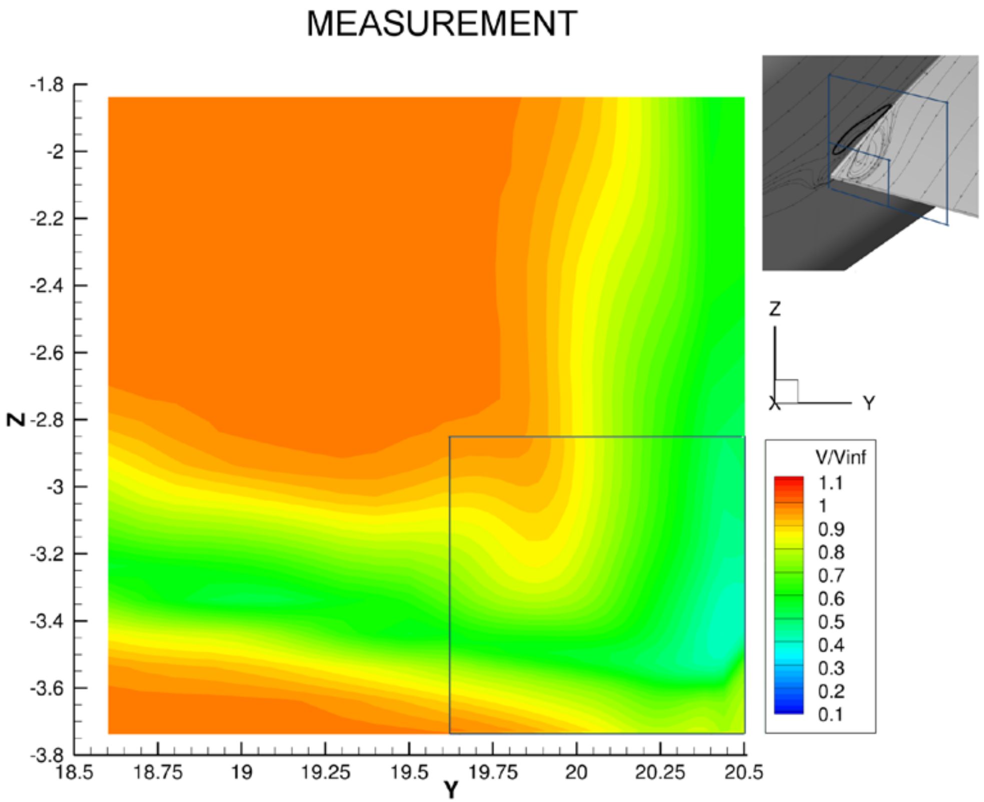
- EFD shows ‘interesting’ survey 0.5” away from fuselage
- CFD weak vortex upstream of separation zone

56.25”



Results – T.E. Boundary Layer





CFD

- Evidence of separation downstream of TE
- Larger influence from wing

EFD

- 2"x2" (x 0.1") & 0.9"x 0.9" (x 0.030")
- Small hint of separation
- Velocities affected by flow angularity

Summary

NASA Ames team was asked to run a risk assessment test on semi-span, wall-mounted JFM model. Results showed inconsistencies between EFD and CFD for wall-mounted model. Committee decided to proceed with JF test using a sting-mounted model.

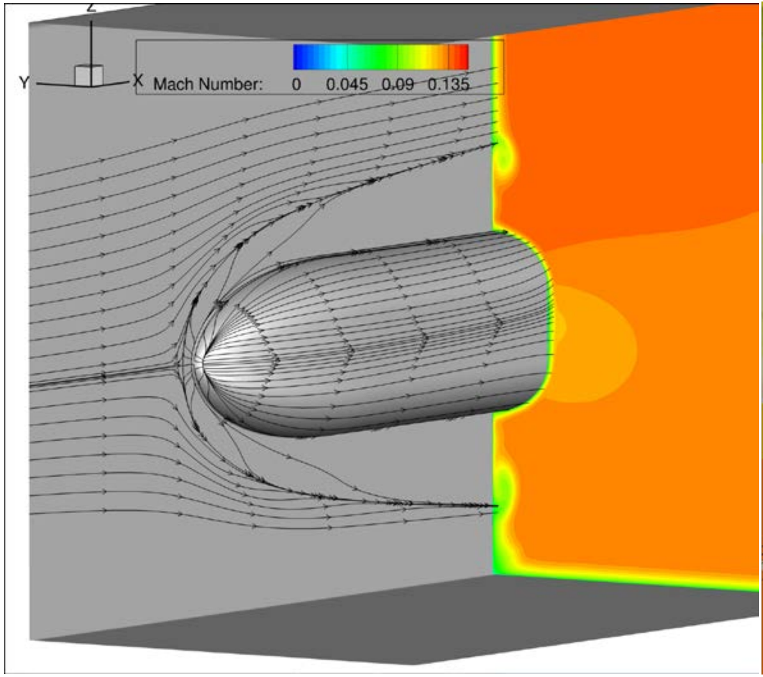
Concluding Remarks from Results

- Significant flow separation zones were not observed in EFD
- CFD show separation on wing/fuselage trailing edge junction
- To correctly simulate the juncture flow, both horseshoe vortex and wing boundary layer must be captured accurately
    a. Thicker b.l. in EFD, lack of separation bubble.
    b. Thinner b.l. in CFD, weaker vortex, larger side-of-body separation
    c. Influence of the fuselage b.l is apparent in EFD but little influence in CFD
- CFD sees more substantial influence on flow field than EFD

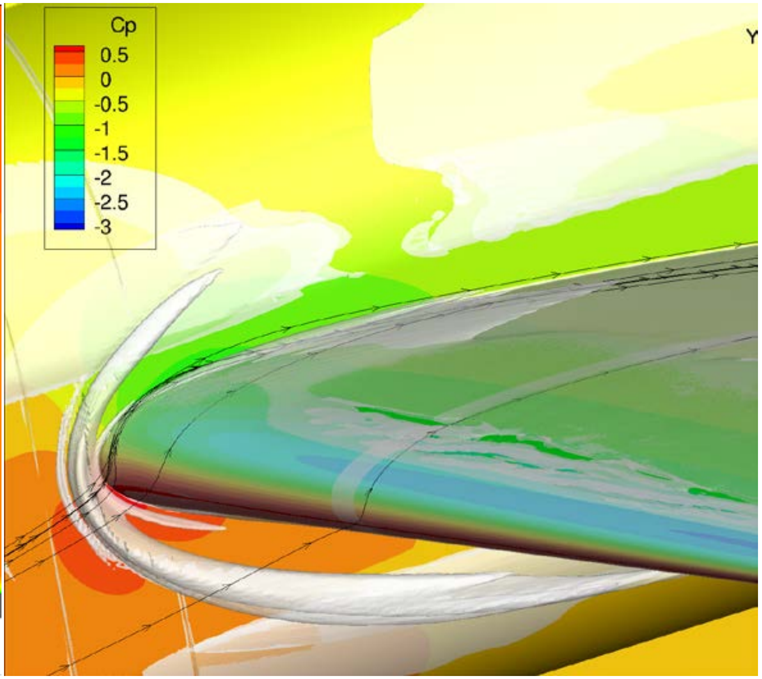
Future Work

- More tunnel runs
  - Need more experimental data to pin down horseshoe vortex on wing and fuselage.
  - What influence does the horseshoe vortex off the nose have on the juncture flow?
  - EFD and CFD are seeing different b.l. on the wing.
  - Turbulence models felt to be inconsistent.

- Flow field is unique and different enough, CFD results aren't perfect ➡ great validation case



Horseshoe Vortex off Nose



Weak Horseshoe Vortex off LE