# Some Recent Applications of the Star-CCM+ CFD Code at the NASA Armstrong Flight Research Center

Trong Bui, Ph.D.

Aerodynamics and Propulsion Branch
NASA Armstrong Flight Research Center

trong.bui-1@nasa.gov
(661) 276-2645

# CFD Uses at NASA Armstrong

- CFD serves as our very own wind tunnel in the computer.
  - With CFD, we could virtually “fly” our aircraft for hundreds of flights before the real airplane ever takes off
  - The aerodynamics of any new aircraft configurations could be safely investigated in the computer before any hardware and/or flight crews are flown and be put at risk
- We support the flight research projects at NASA Armstrong in:
  - Detailed flow physics analysis
  - Full aircraft aerodynamics analysis
  - Aerodynamic flight loads analysis
  - Reviewing external CFD work and provide inputs for Armstrong projects
- Our CFD analysis products are used in:
  - New flight research project advocacy work and Center Innovation Fund (CIF) projects
  - System requirements reviews
  - Preliminary design reviews
  - Critical design reviews
  - Flight readiness reviews
  - Tech briefs
  - Flight planning
  - Mishap investigation boards
  - External collaborations

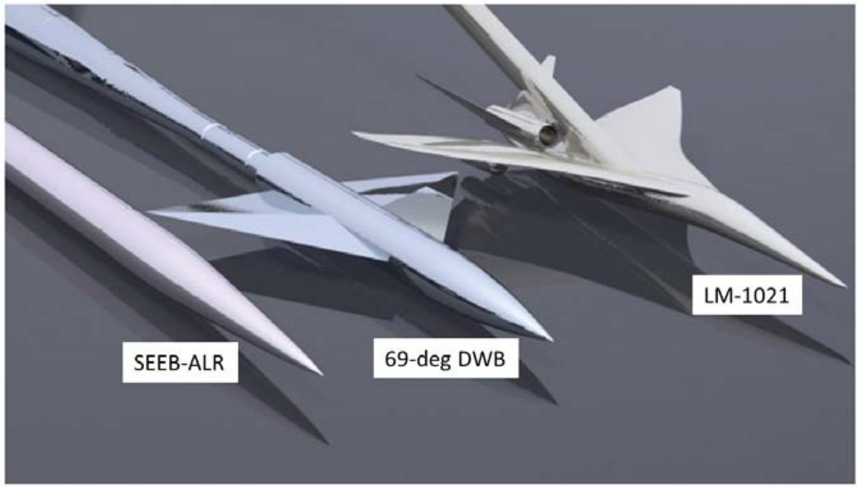
# Our CFD Usage Philosophy

- We are an applied CFD group, not code developers:
  - Our principal goal is to provide timely and responsive answers in supporting the flight research efforts at Armstrong
  - Our required turnaround time is often on the order of 1-2 weeks, not 3-6 months nor a year
- We prefer to start with exact, simple analysis whenever possible, then bring in higher-order analysis tools as needed/required. Examples include:
  - Hoerner's empirical handbook quick-look analysis for Ikhana pods aerodynamics
  - Vortex lattice low-order quick-look analysis for GIII Subsonic Aircraft Roughness Glove Experiment (SARGE) Glove aerodynamics for System Requirements Review
  - 80% of the GIII SARGE Glove and Adaptive Compliant Trailing Edge (ACTE) Flap PDR and CDR analysis matrix were done using Tranair full-potential flow code
  - STAR-CCM+ full Navier-Stokes analysis was then only conducted for (1) validation of Tranair analysis and (2) complex flow cases such as strong shock/boundary layer interactions, high angles of attack, high flap deflections, and stalls
- We document the results of our work in NASA reports, conference papers, and journal papers. We also archive our work to support planning for future work as well as mishap/accident investigations etc…
- Our principal goal is to have routine, quick-turnaround CFD simulations of full aircraft configurations in flight, with possibly store separation as well as unsteady aeroelastic effects

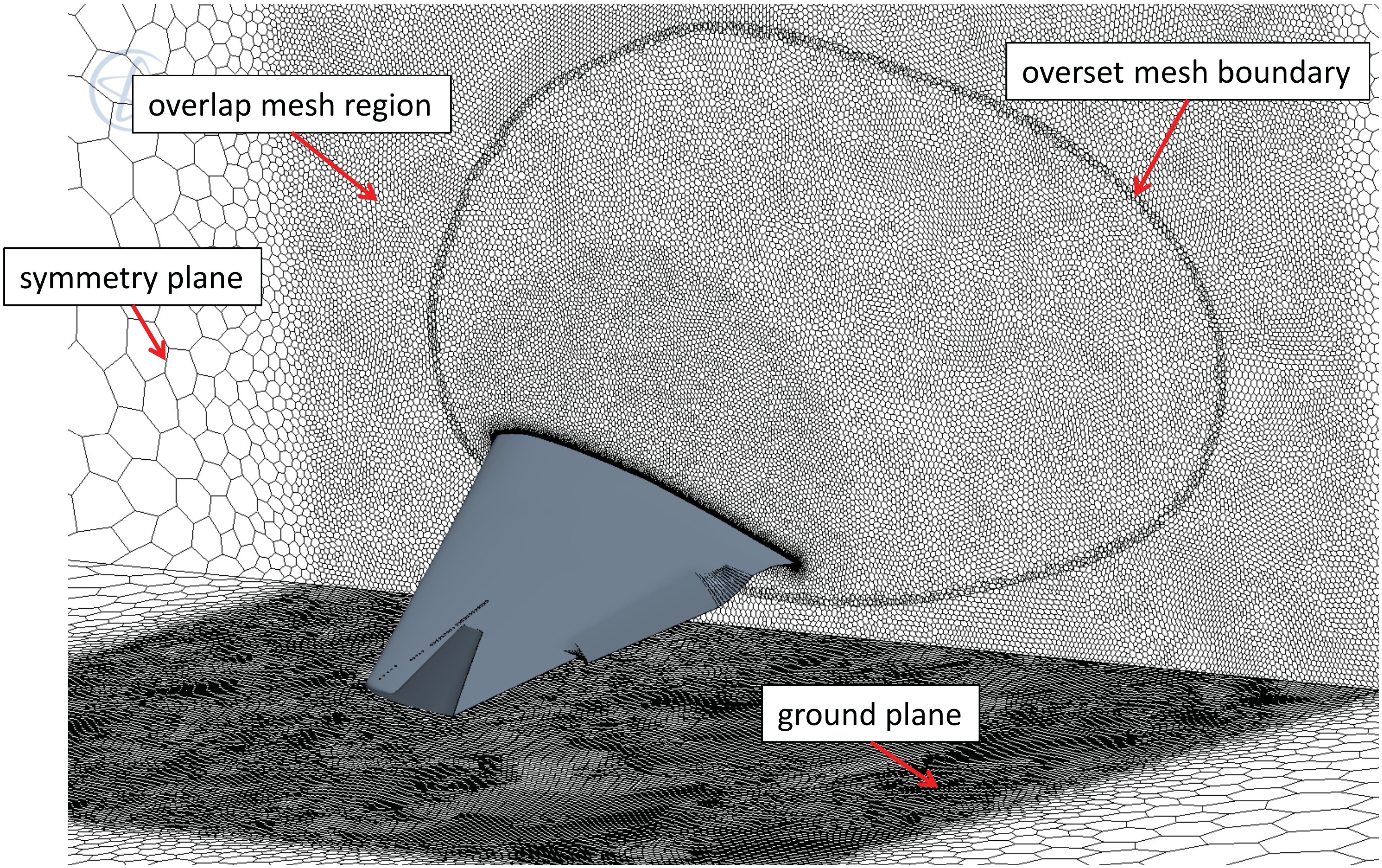
# Recent Applications

- GIII/ACTE

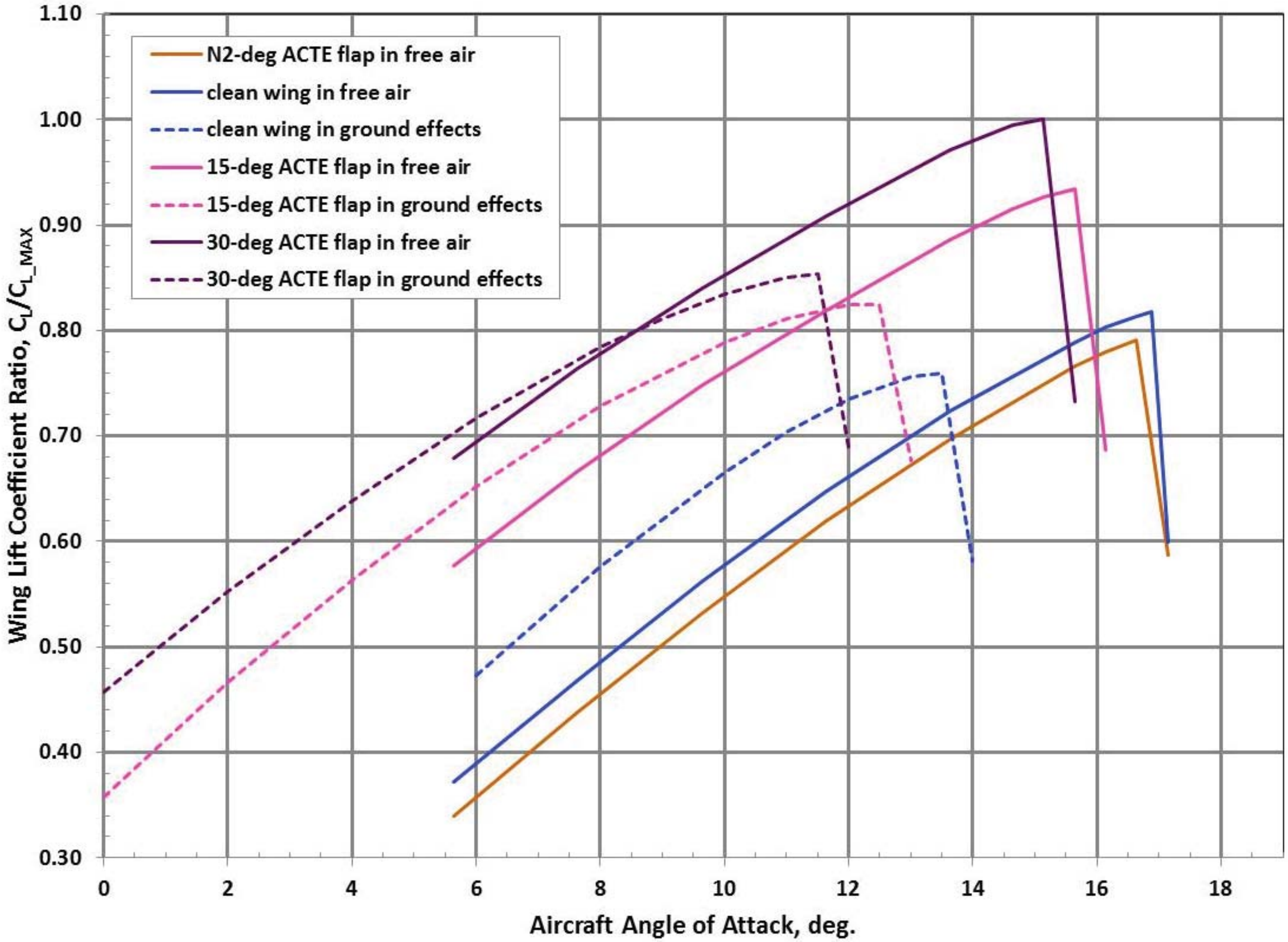

- Sonic Boom Validation



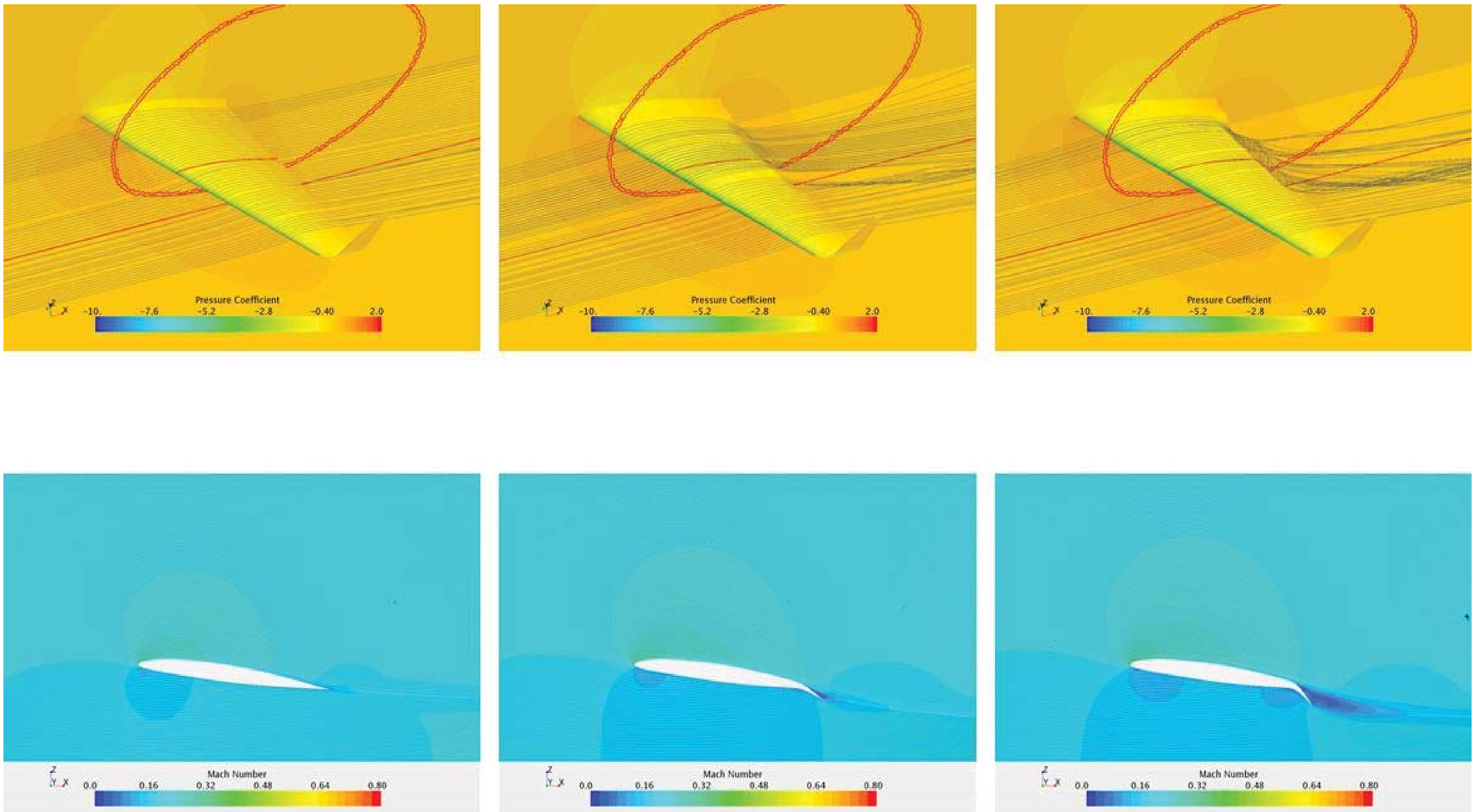
# ACTE Wing Stall in Ground Effects



# Star-CCM+ Lift Coefficient Stall Results

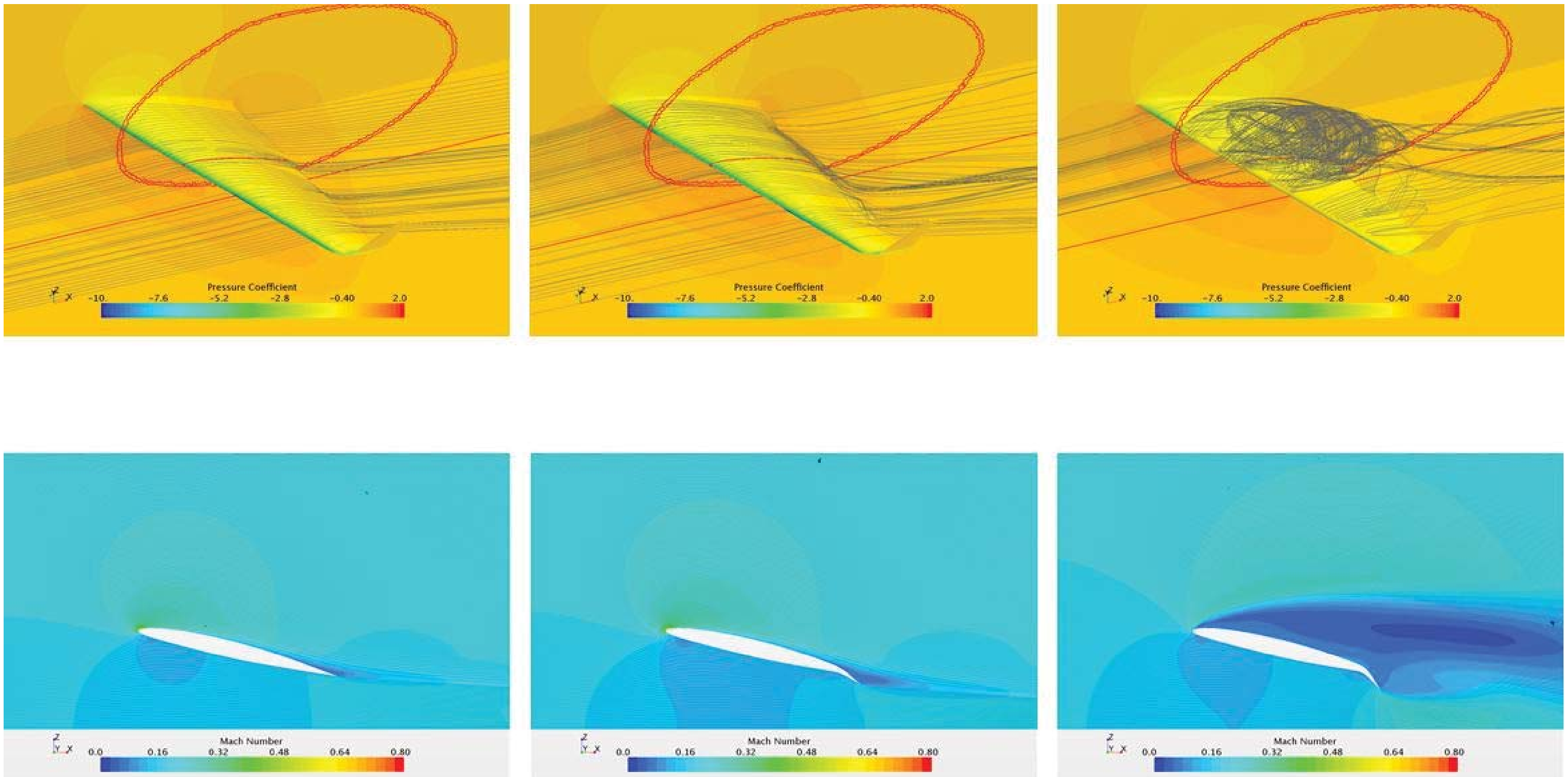


# Flow Comparison at 6-Deg AoA




CLEAN 15-DEG ACTE 30-DEG ACTE

# Flow Comparison at 12-Deg AoA

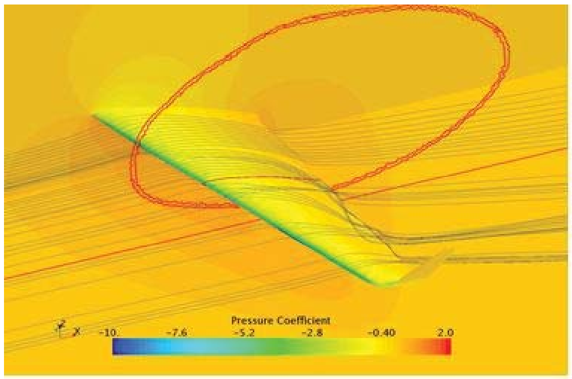


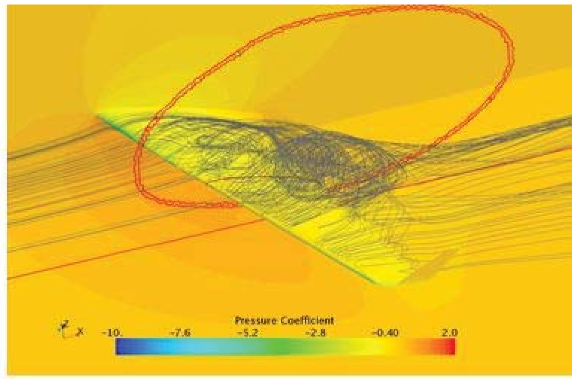

CLEAN

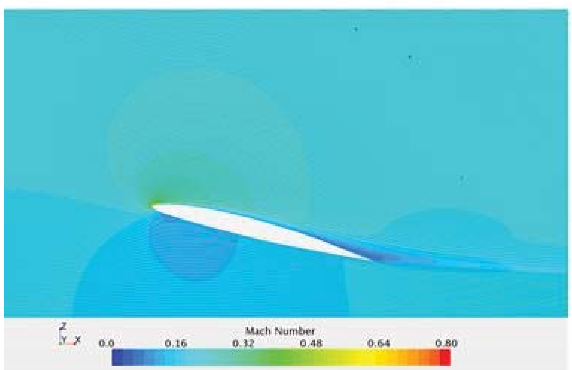
15-DEG ACTE

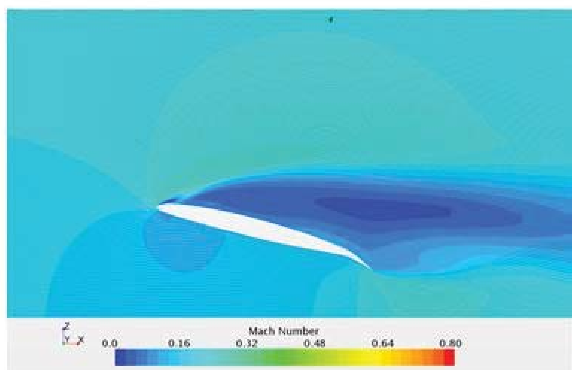
30-DEG ACTE

# Flow Comparison at 13-Deg AoA



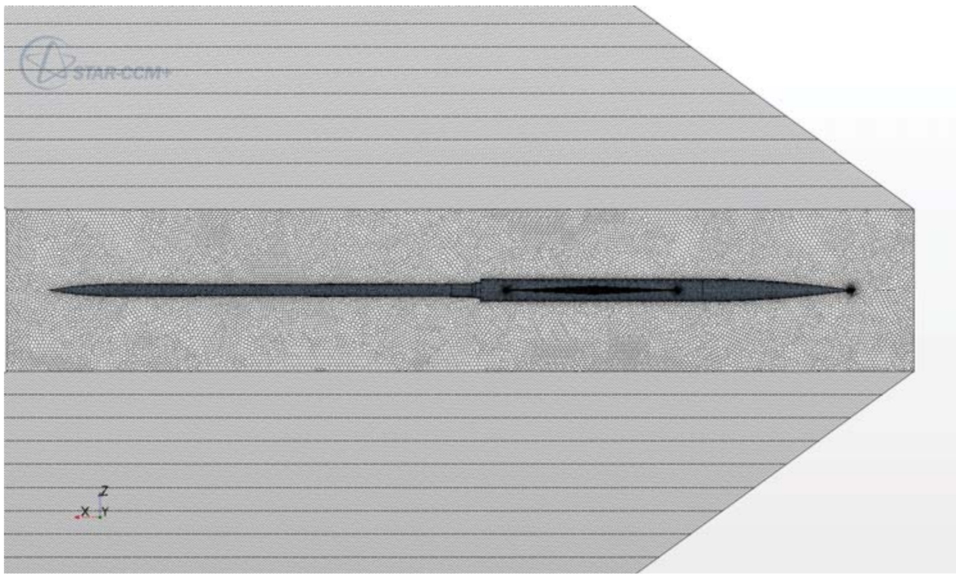



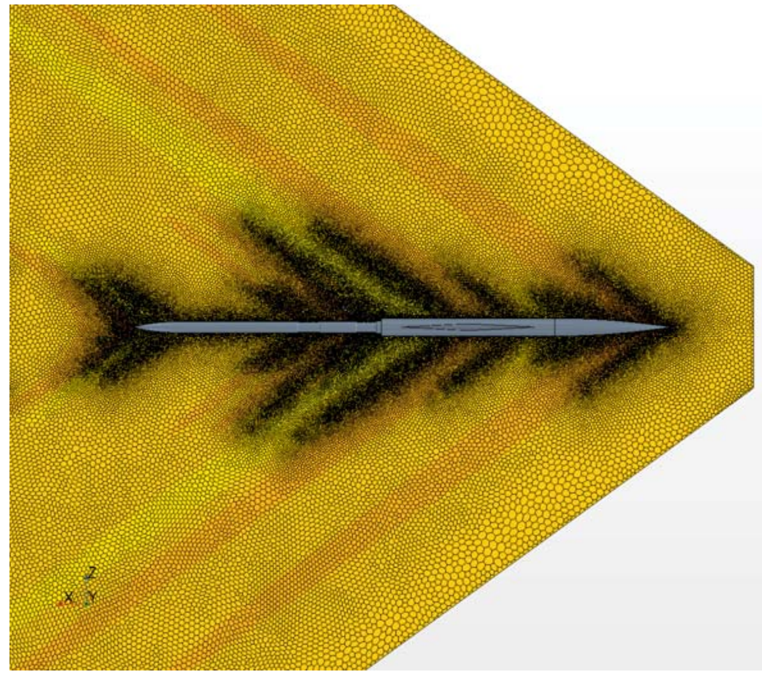




CLEAN

15-DEG ACTE

# Star-CCM+ Sonic Boom Analysis Approaches

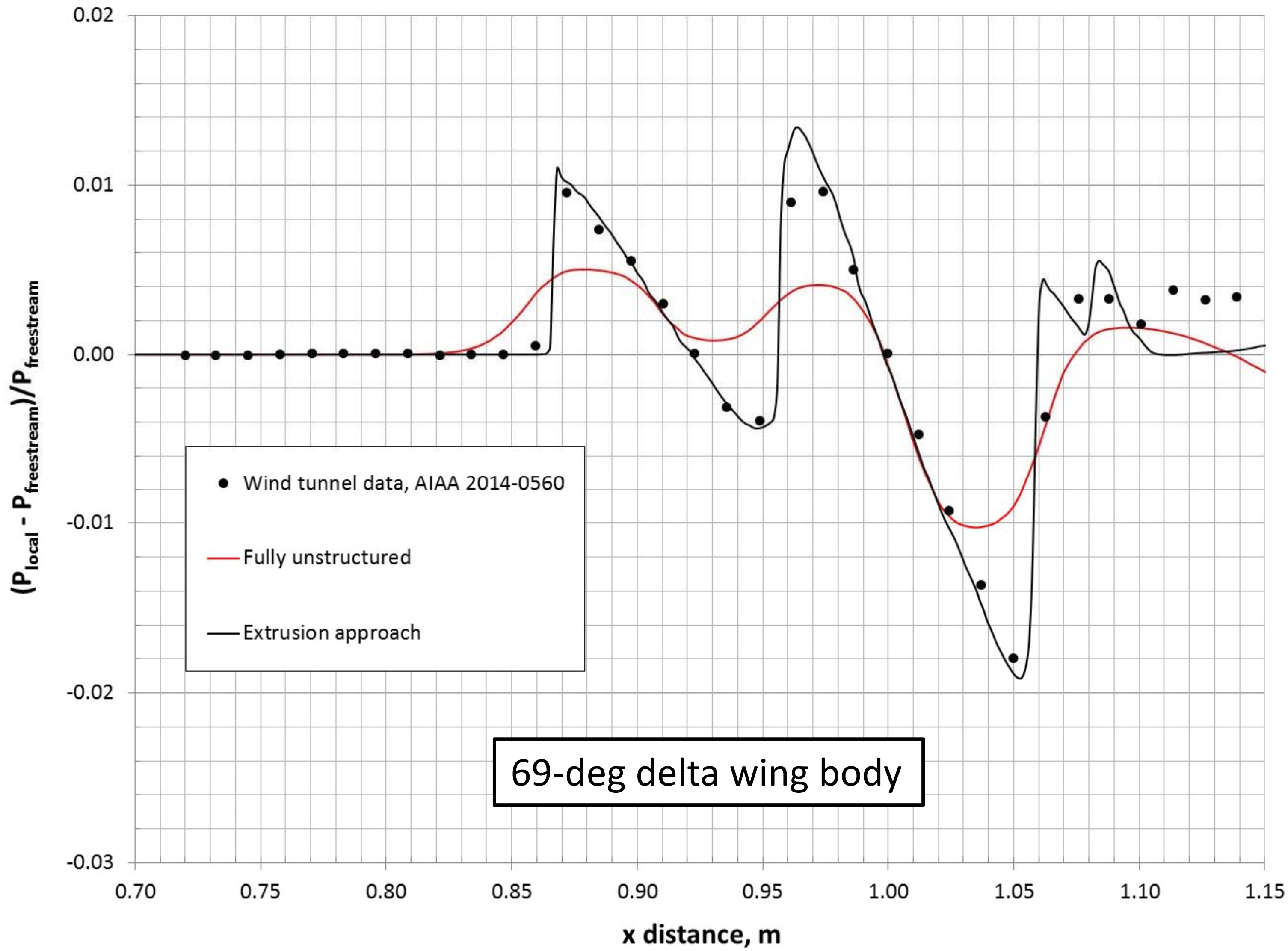


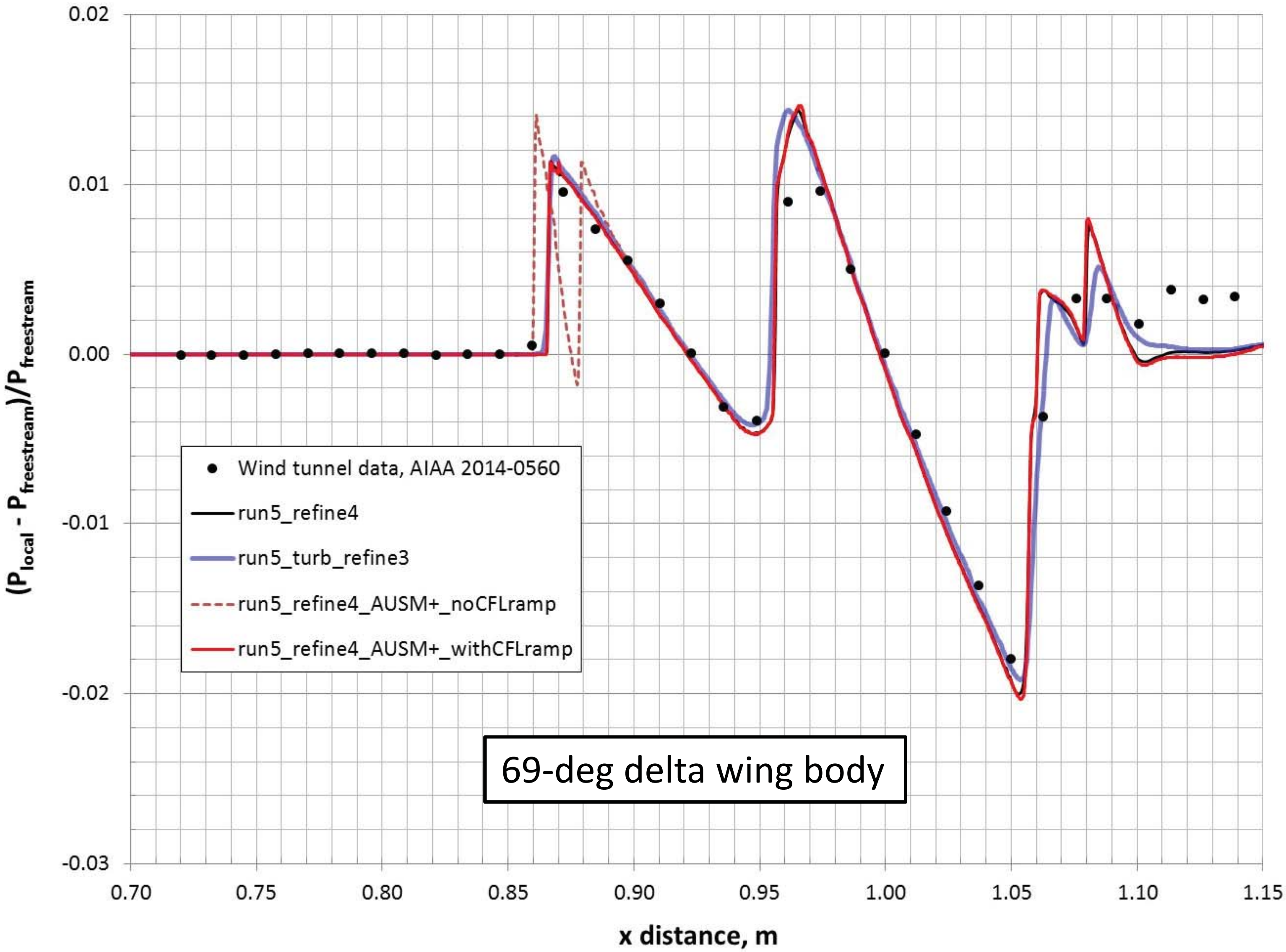
EXTRUSION:
Polygonal unstructured local mesh with Mach-aligned prism layer extrusion

FULLY UNSTRUCTURED:
Polygonal unstructured mesh everywhere with feature-based mesh adaptation and refinement

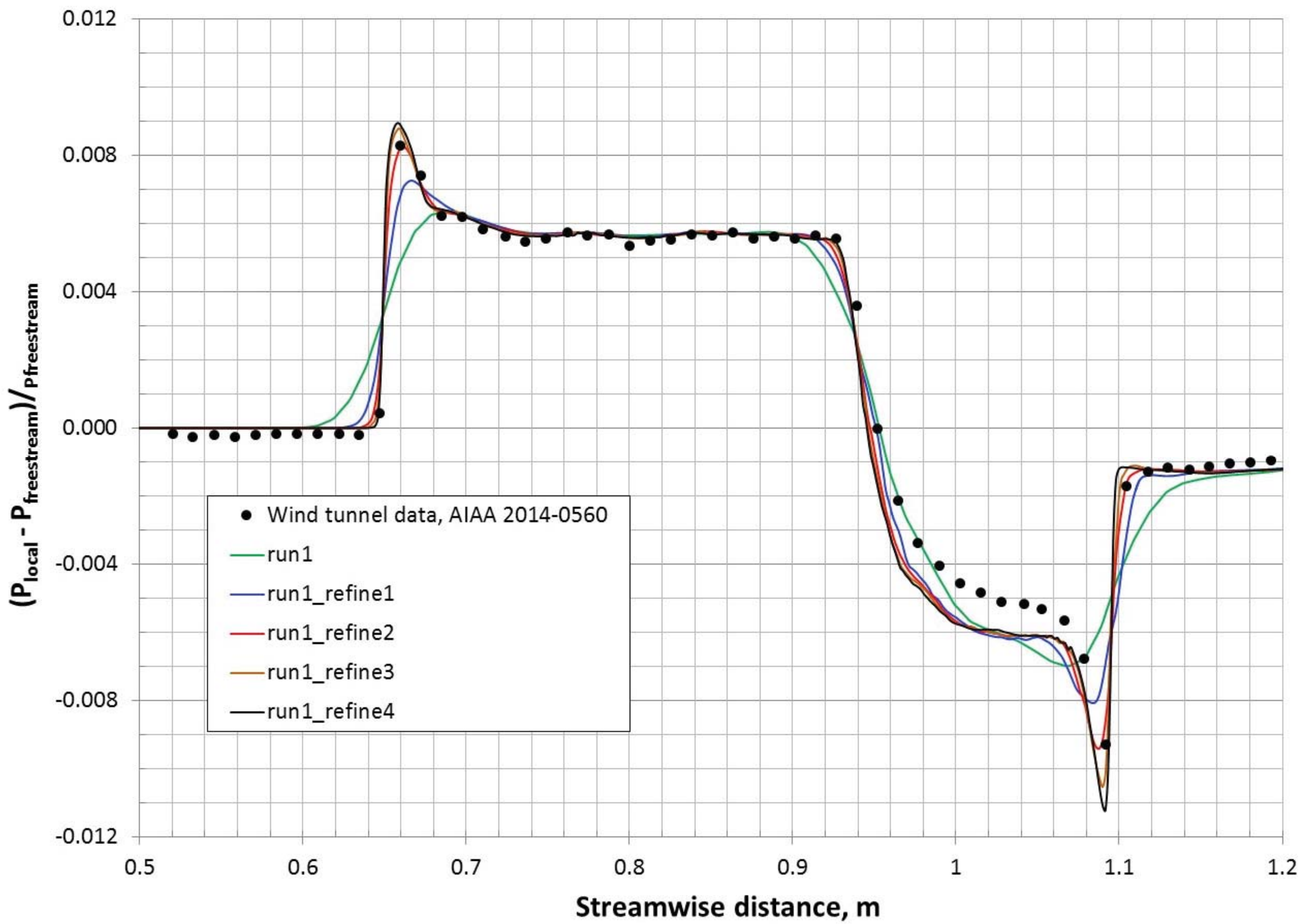
# Extrusion Approach is More Effective



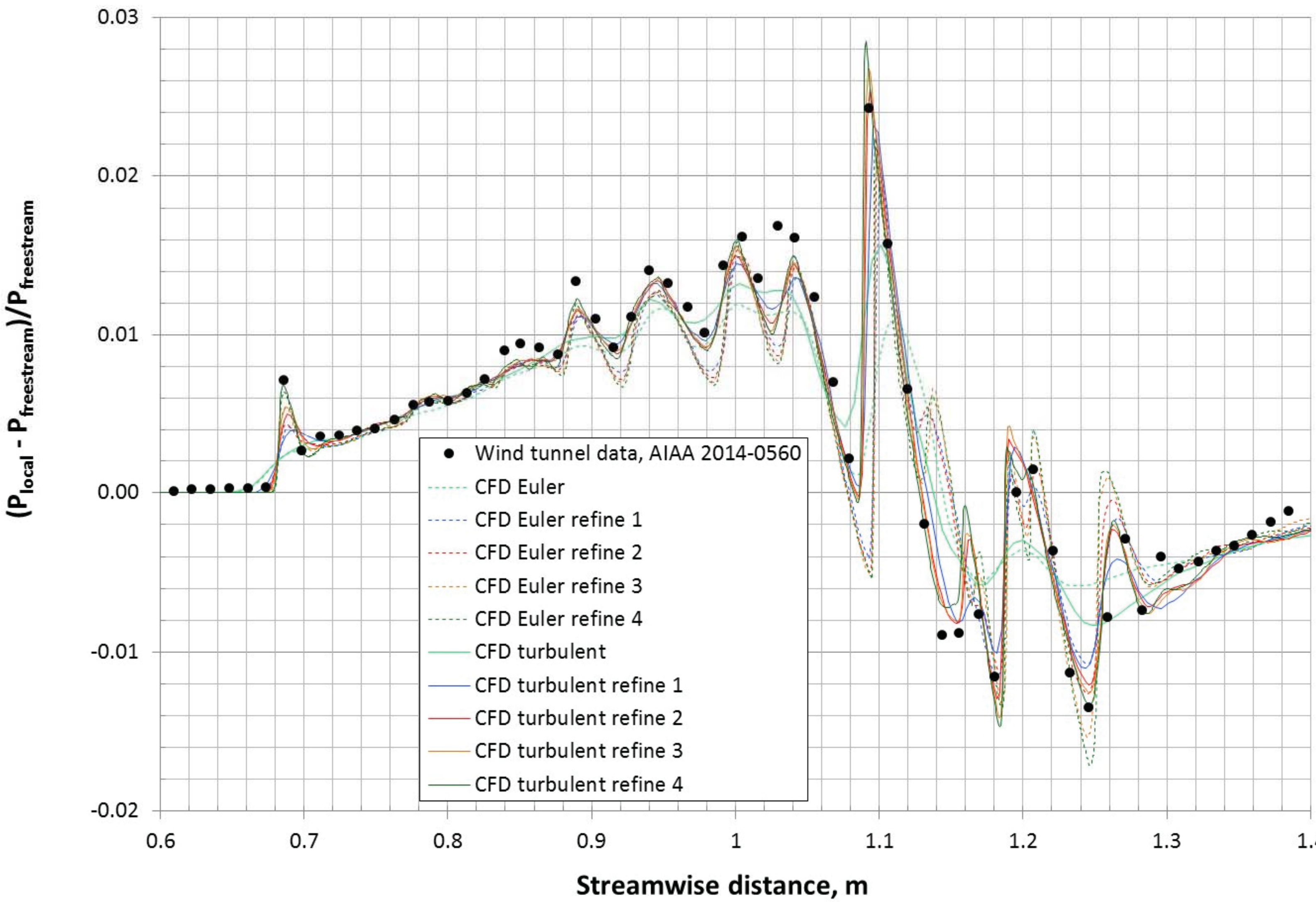
# AUSM+ Inviscid Flux Problem



# SEEB-ALR Results



# LM-1021Results



# Lessons Learned

- The Star-CCM+ CFD code provides good results for low-speed wing stall analysis
- Overset meshing approach is required for ground effects wing stall analysis
- The extrusion approach is required for accurate sonic boom prediction and propagation
- AUSM+ inviscid flux is “shock-happy” and can add extra nonphysical shocks in the CFD solution
- Viscous effects might be required for complex aircraft geometries for accurate sonic boom prediction – Different shock strengths, numbers, and positions than inviscid Euler solutions

# Benefits and Challenges in Using STAR-CCM+

- Benefits:
  - Fast, highly automated CAD healing and grid generation capabilities allow for faster CFD analysis turnaround times
  - Flow solver is highly accurate for low-speed as well as high-speed aerodynamic analyses
  - Overset meshing enables complex CFD analyses
- Challenges:
  - Better IGES and STEP import quality
  - Overset solver significantly increases the RAM and CPU requirements
  - The post-processing computation of forces and moments for arbitrary boundaries is very “clumsy” after a converged solution is obtained. We need to decouple post-processing from the solver’s Region->Boundaries topology
  - Streamlines and streaklines tool could use more precise and easier placement of seeds.
  - AUSM+ inviscid flux function is too “shock-happy” and is inconsistent
- Wish list:
  - Automatic, solution-adaptive 3D and 2D mesh refinement and de-refinement capability
  - Better streamlines and streaklines placement tool
  - Better ruler tool for measurements with more accuracy and precision
  - Less computer memory and file size requirements, especially for the overset flow solver