Using CFD to Develop NASA's X-57 Maxwell Flight Simulator

Jared Duensing, Jeffrey Housman, James Jensen, Daniel Maldonado, Cetin Kiris
Computational Aerosciences Branch, NASA Ames

in collaboration with

Michael Frederick, Seung Yoo, Trong Bui
NASA Armstrong

Evolution of the X-57

NASA aims to achieve a 5X reduction in energy consumption compared to conventional propulsion through the X-57 design.

LeapTech Experiment

Demonstrated that distributed propulsion could provide nearly a 2X increase in lift relative to a traditional wing and propulsion system.

Mod-II

Proved the feasibility of two electrically driven propellers in place of traditional combustion engines.

Mod-III/Mod-IV

Combines distributed propulsion technology with electrically powered propellers.
Mod-III studies the cruise propellers only, Mod-IV studies the high-lift propellers only.

X-57 Design Overview

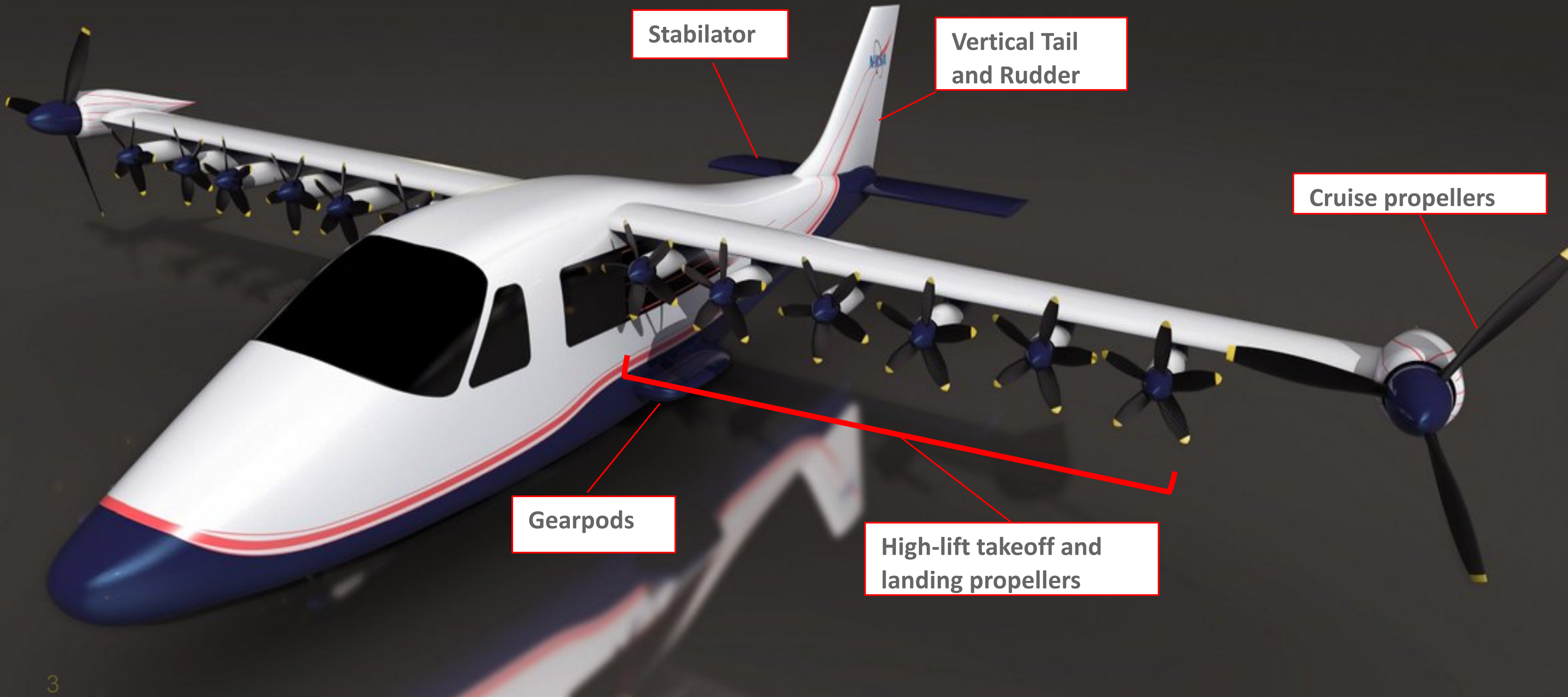
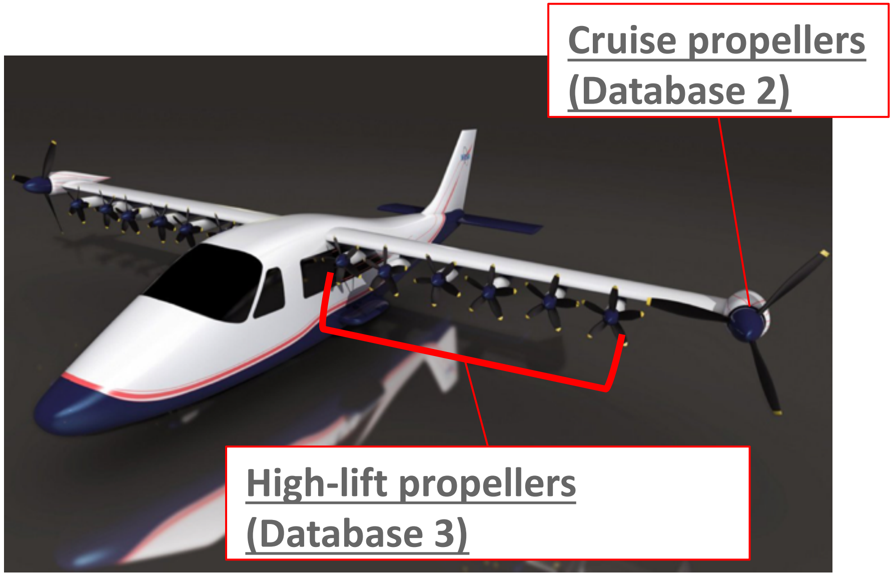


Image source:
https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20170001218.pdf

Video source: *https://www.youtube.com/watch?v=4X1FxZgfFbc*

Objectives

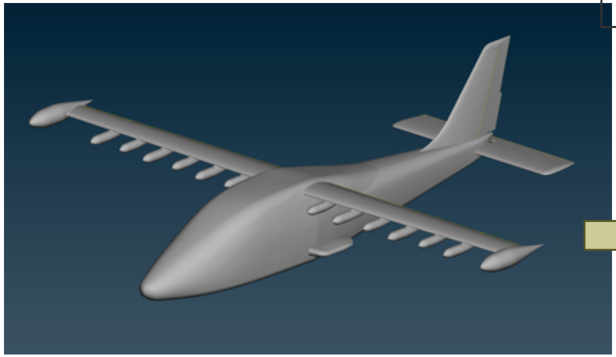
- **Establish best-practices to generate an aerodynamic database using the LAVA (Launch Ascent and Vehicle Aerodynamics) and Star-CCM+ flow solvers**
- **These best-practices are being applied to CFD databases which cover a variety of flight conditions**
    - Database 1 (188 simulations): Power-off
    - Database 2 (233 simulations): Cruise power-on
    - Database 3 (1000+ simulations): high-lift power-on
- **The database results will be used to design the flight simulator and control systems for the aircraft**



CFD Simulation Process

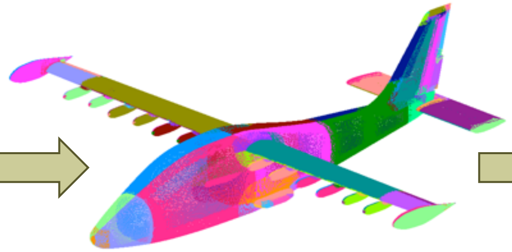
- **Computational fluid dynamics (CFD) is an engineering tool that applies physics, mathematics and computer science to predict how the X-57 will perform aerodynamically in a wide variety of flight scenarios.**
- **All results for the X-57 presented here were generated using the LAVA CFD code on the Mod. III geometry.**

Workflow for Simulating an Aircraft



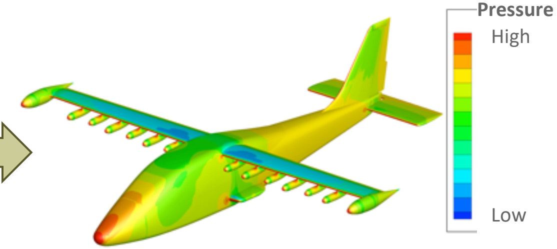
Geometry Repairs

Before any CFD application can begin, the aircraft geometry must be cleaned using Computer-Aided Design (CAD) software. The key is to remove complex details that will not greatly impact the solution since this will greatly simplify the grid generation process.



Grid Generation

Constructing a computational grid that will capture as much geometric detail as possible while also remaining coarse enough to be solved with available resources is crucial. Occasionally, this process can be iterative if initial simulations capture flow phenomena that the user wishes to resolve in greater detail.

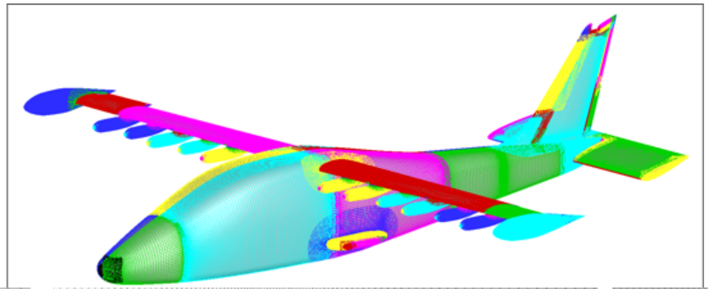


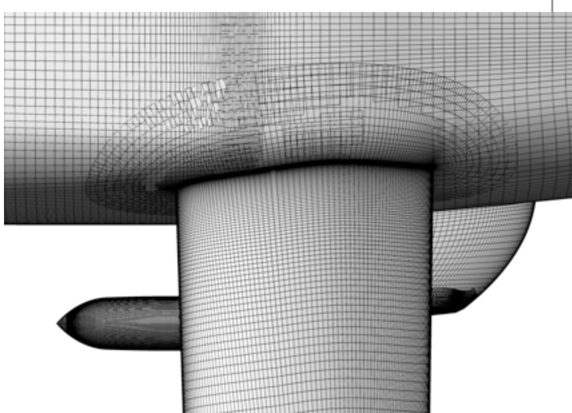
Flow Simulation

A well-written CFD code should be robust to handle fluid problems in all flow regimes. For example, the user must be aware that a low-speed subsonic application such as the X-57 might require solver inputs and settings for stability and convergence that supersonic applications do not require. Above is a sample solution showing the surface pressure distribution.

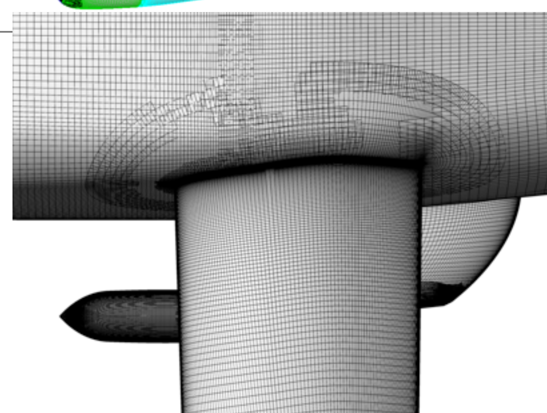
Mesh Generation

- **Mesh generation is a crucial step in obtaining an accurate solution when simulating aircraft performance**
- **A mesh refinement study, which analyzes the solution error caused by using a discrete computational domain, is often a vital step before extensive database calculations are begun**

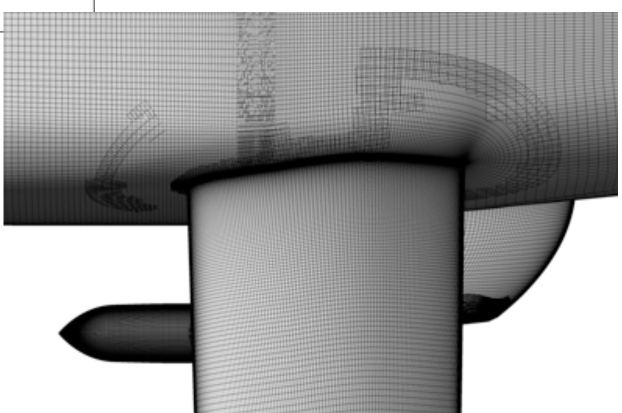




Structured Coarse



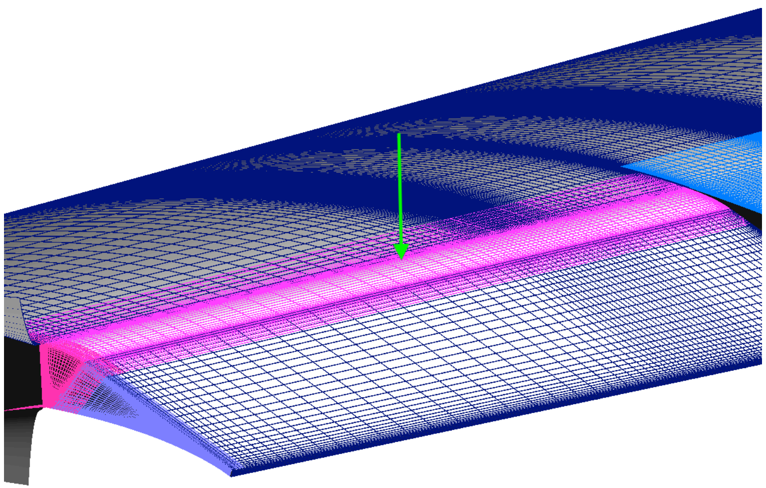
Structured Medium



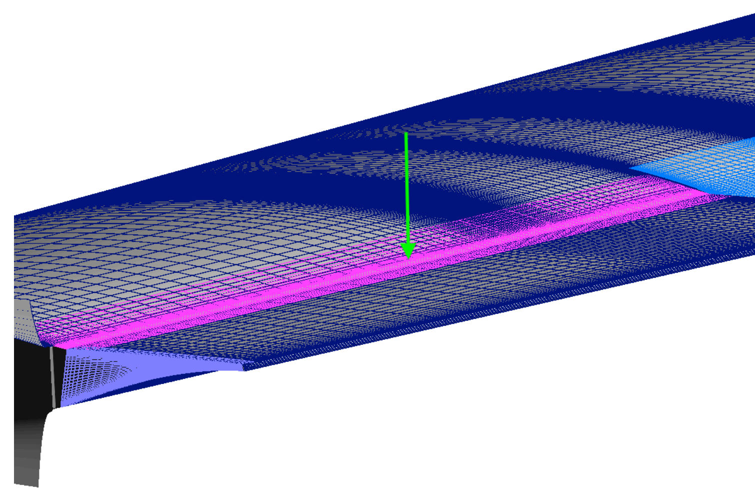
Structured Fine

Mesh Procedure for Moving Geometry

- **Database runs require the articulation of control surfaces to a specified angle**
- **An automation procedure was developed for this project which allows the user to freely deflect all controls to a desired angle and regenerate the mesh in less than 10 minutes.**



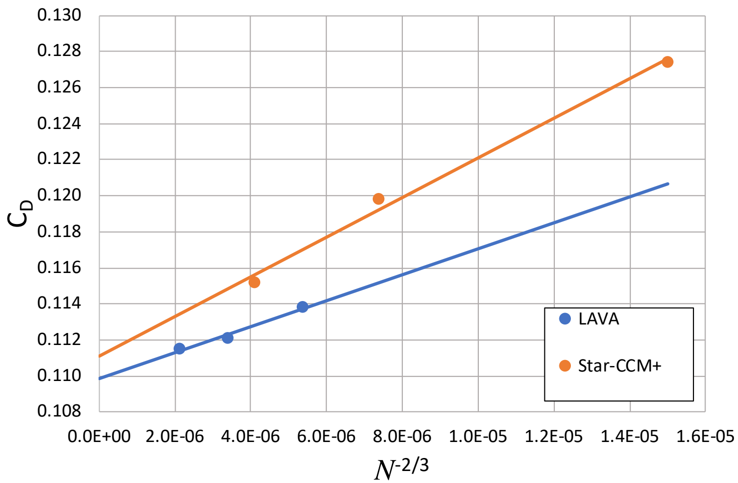
δ_a= +10°

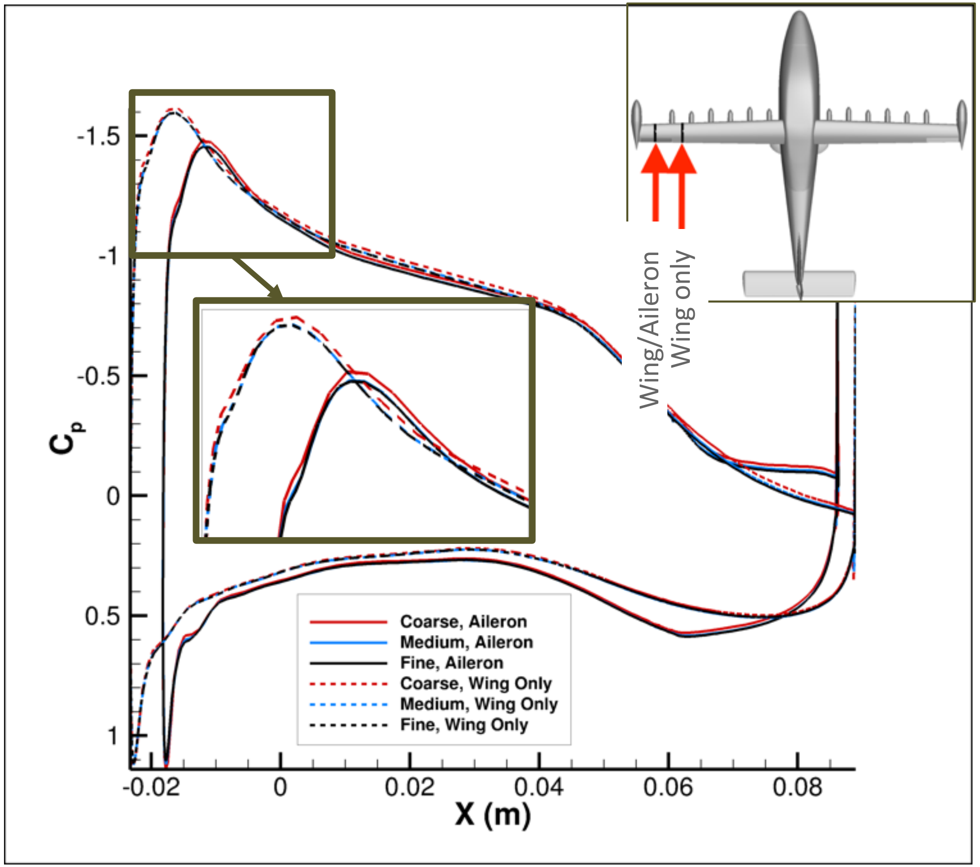


δ_a= -10°

Mesh Refinement Study Results

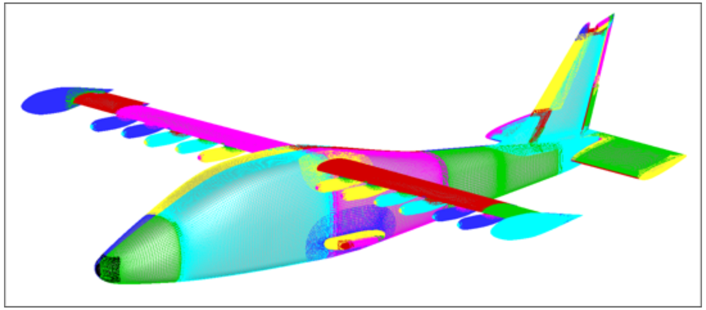
- **Analysis of the pressure distribution at selected wing locations show strong agreement across all three refinement levels**
- **Very small change in aerodynamic loading between medium and fine mesh levels**



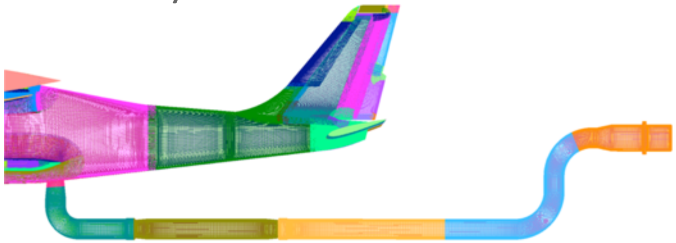


Wind Tunnel Validation

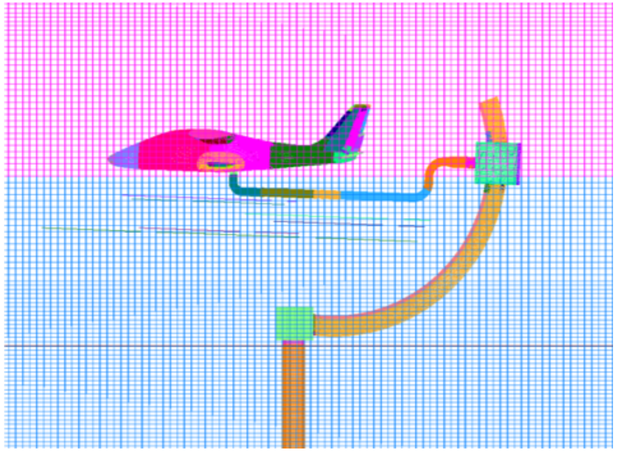
- **Component build-up incorporates wind tunnel hardware into the CFD simulation that could potentially influence aircraft loading**



Free air: Baseline simulation approach used in refinement study.



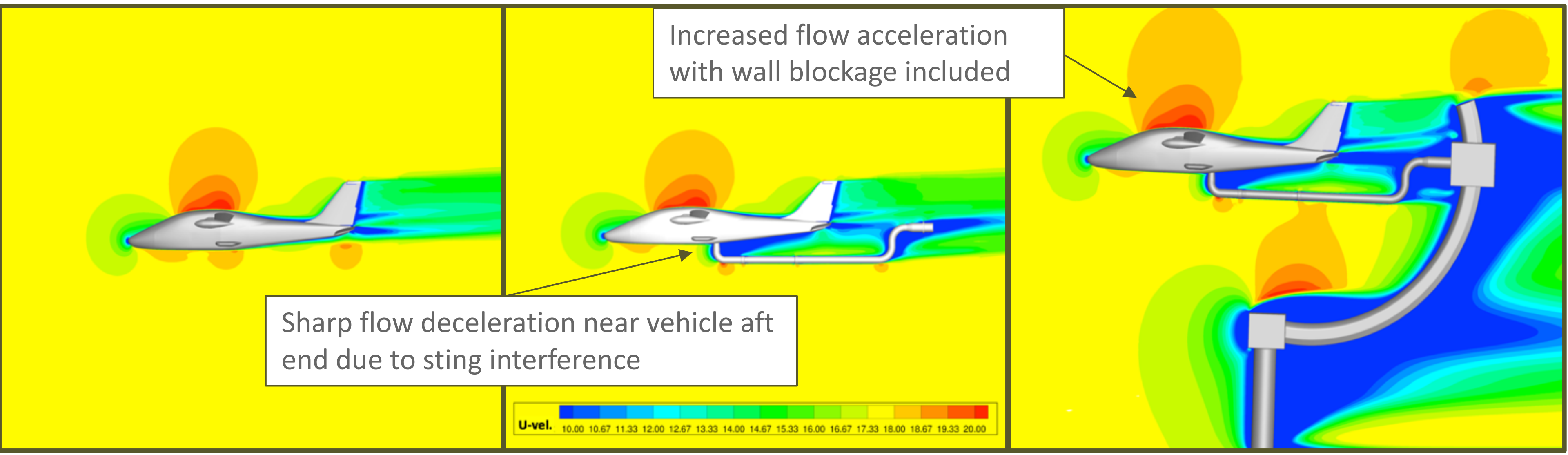
Free air + sting: Adds the sting mounting fixture to the free air simulation.



Free air + sting + wind tunnel: Adds the C-strut mount and encloses the aircraft in a 12 ft. x 12 ft. octagonal channel similar to the low-speed test section.

Validation Simulation Results (U-Velocity (m/s) on Symmetry Plane)

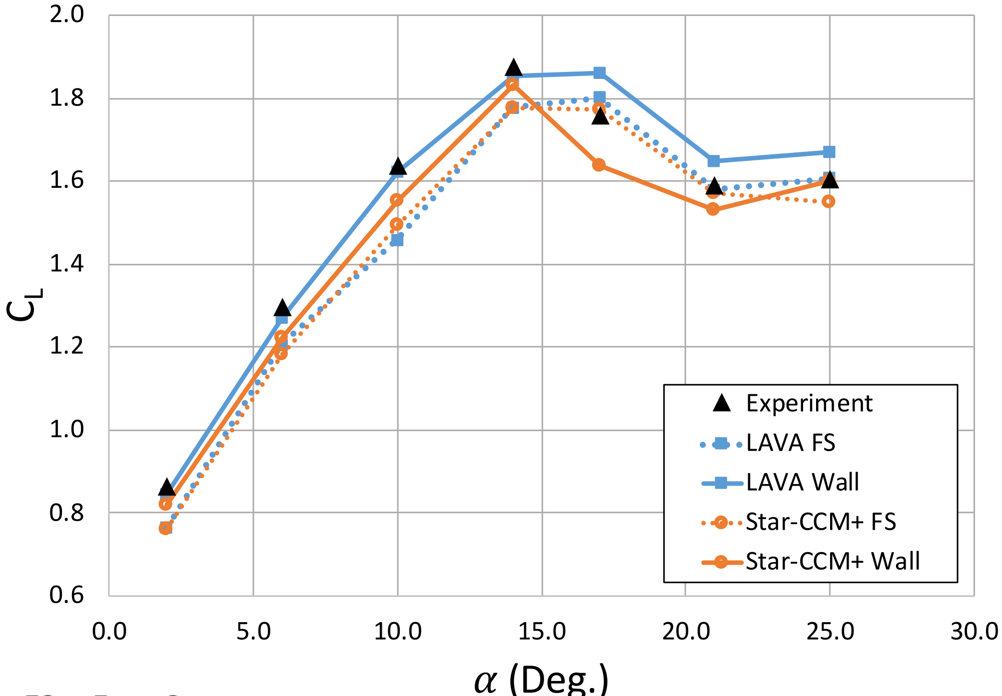
- **Substantial qualitative differences in fluid dynamics resulting from sting, C-strut and wind tunnel walls**
- **Hardware locally impacts flow field while effects also propagate upstream to test article location**



Free air | Free air + sting | Free air + sting + wind tunnel

Angle of Attack Sweep Results

- **Comparison of multiple angles of attack in free air and with wind tunnel hardware further demonstrate modeling impacts**
- **For all codes, incorporating wind tunnel effects to the CFD simulation improve lift predictions considerably across the linear regime of the C_L vs. α curve**



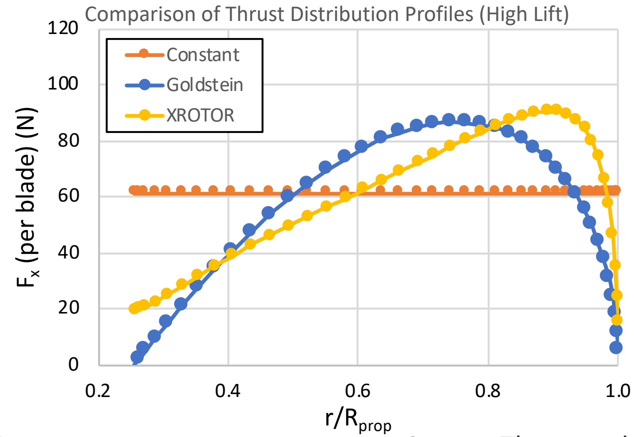
FS = Free Stream

Wall = Wind Tunnel + Sting + C-strut

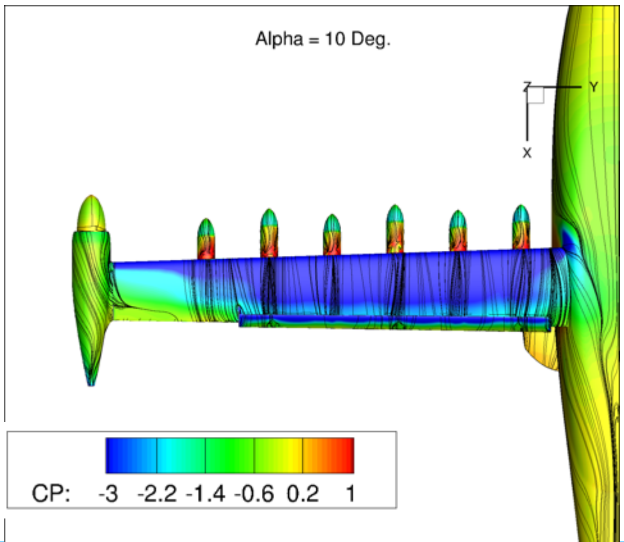
Power-On Database Results

Selecting Actuator Zone Distributions

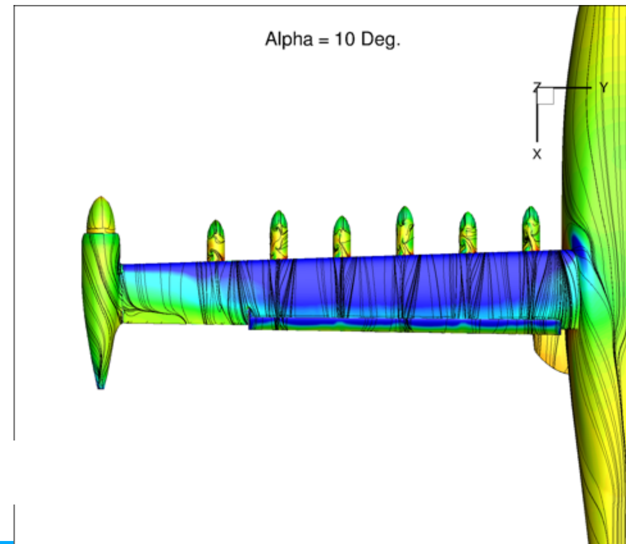
- **For high-lift and cruise propulsion cases, actuator zones are used to model the effects of spinning propellers on the surrounding flow field**
- **Initial CFD simulations were performed using LAVA to understand impact of thrust and torque distributions on the solution (Altitude: 6,000 ft., Re_{MAC} = 1,235,000, Mach = 0.098, α = 10°)**



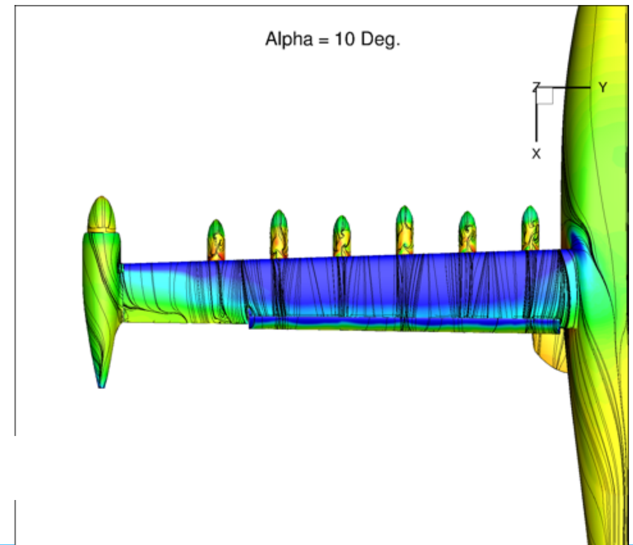
Constant Thrust and Torque



Goldstein Thrust and Torque



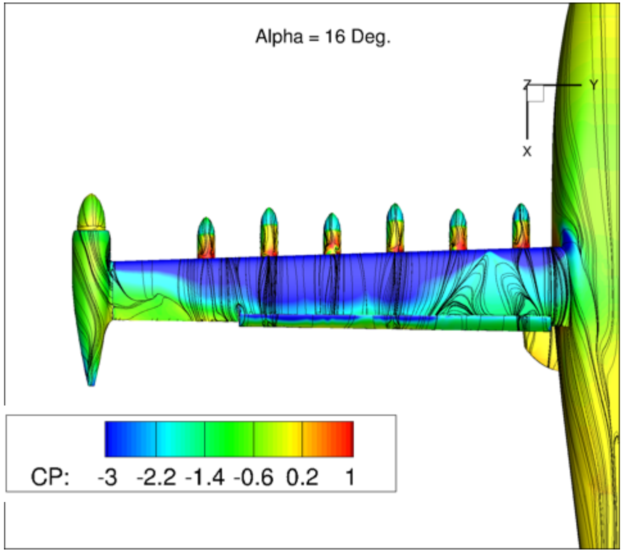
Custom Thrust and Torque



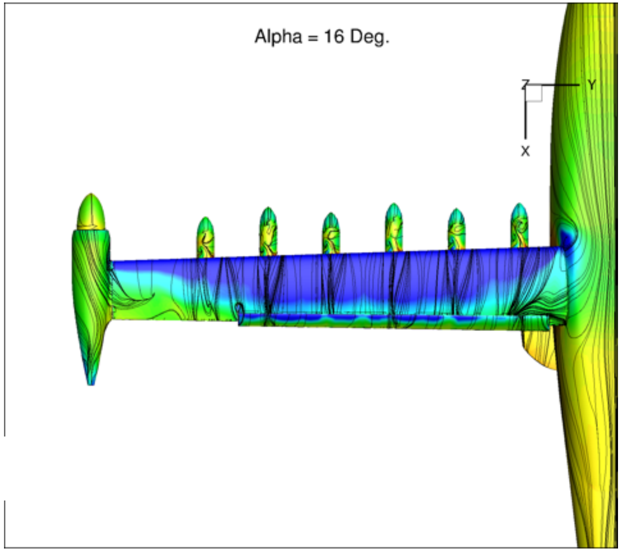
Selecting Actuator Zone Distributions

- **Initial CFD simulations were performed using LAVA to understand impact of thrust and torque distributions on the solution**
- **Altitude: 6000 ft., ReMAC = 1,235,000, Mach = 0.098, α = 16º shown below**
- **Separation behavior at high angle of attack highly dependent on thrust and torque distribution**

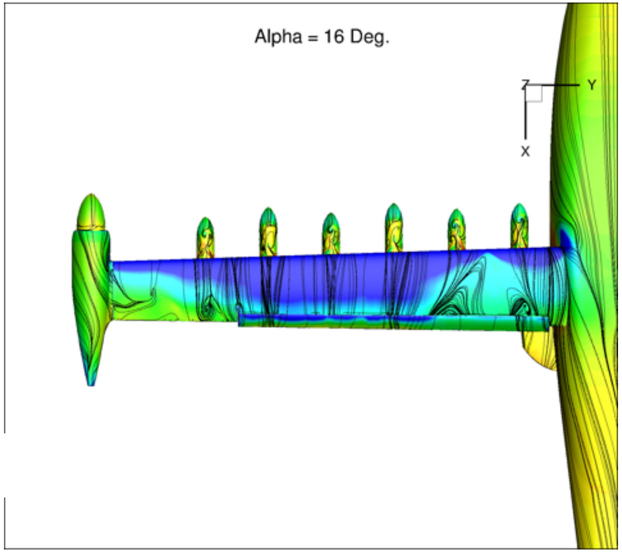
Constant Thrust and Torque



Goldstein Thrust and Torque

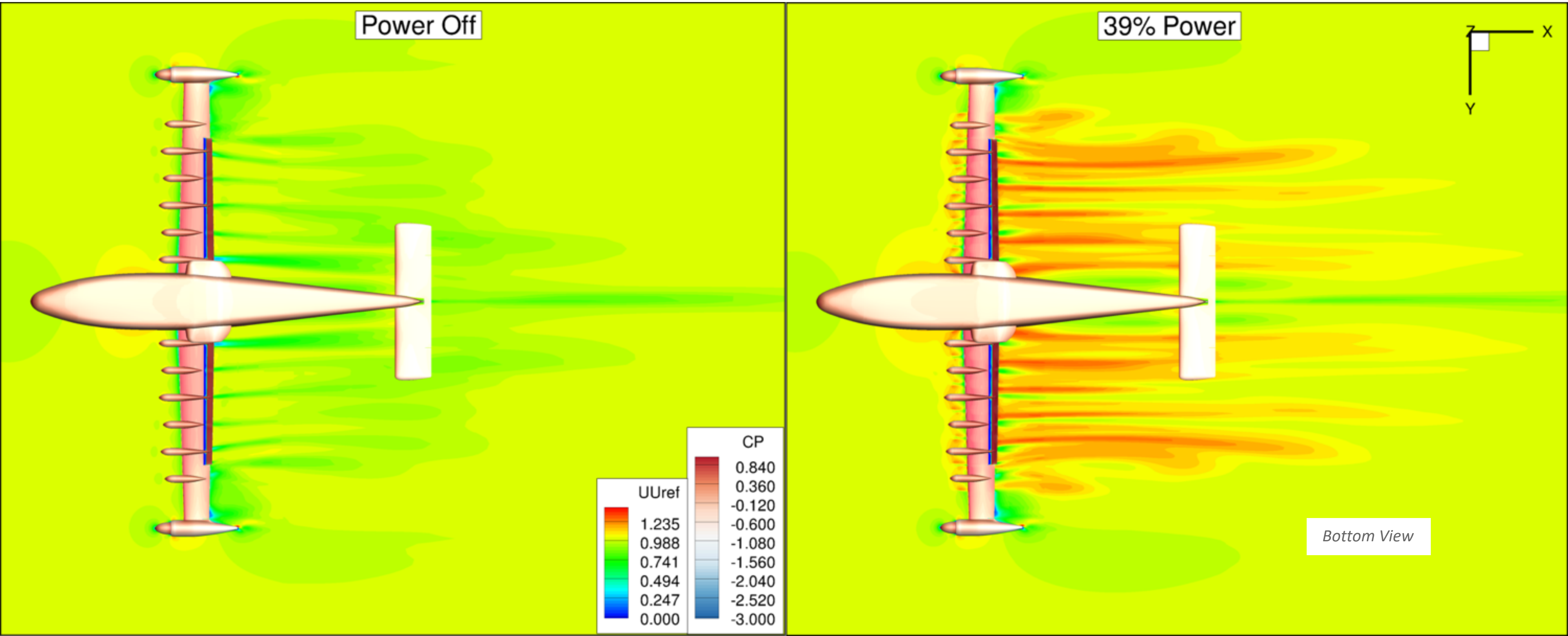


Custom Thrust and Torque



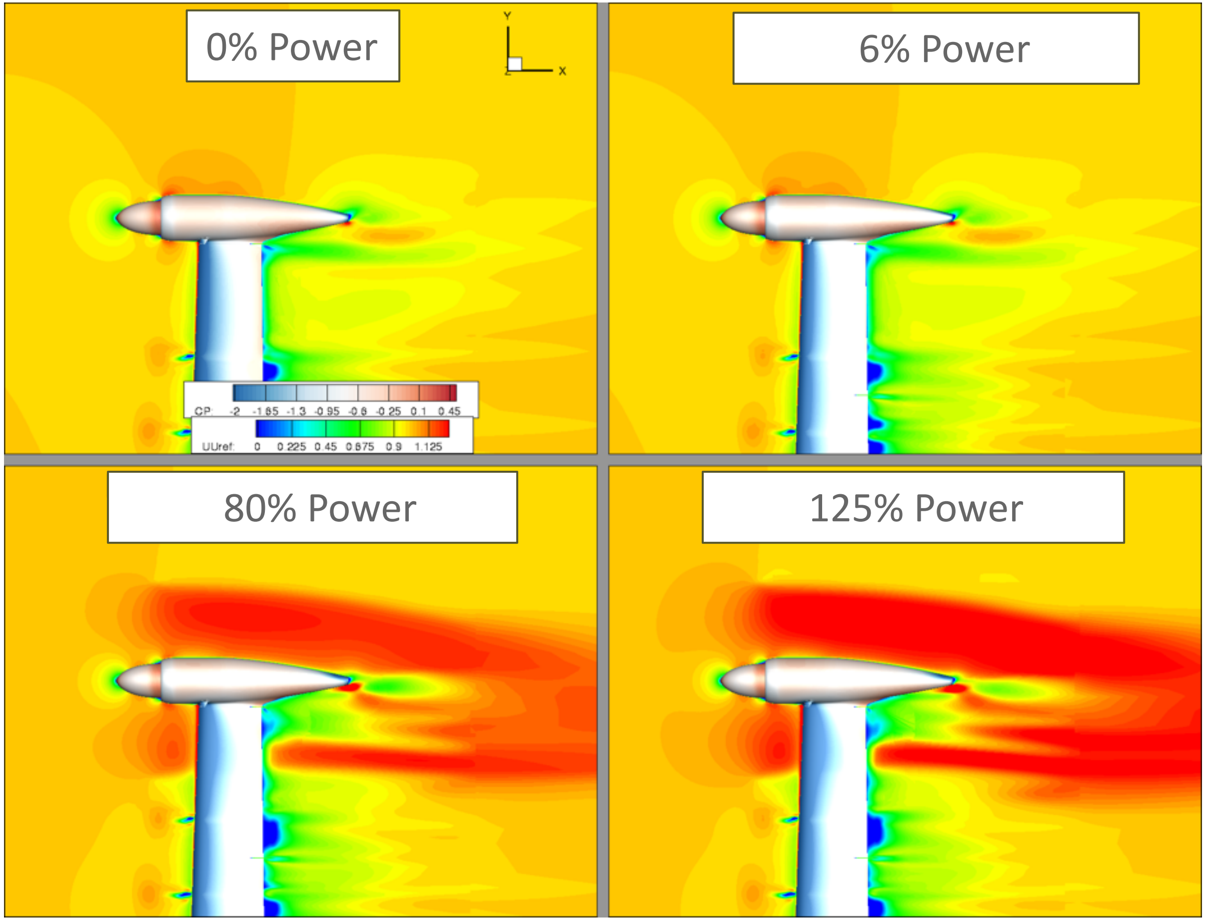
High-Lift Power-On Flow Visualizations

- **Accurate aerodynamic deltas can now be computed between power-off and power-on cases once the desired thrust and torque distributions are selected**
- **Dimensionless streamwise velocity (*U/U_{ref}*) is shown on slice plane, pressure coefficient on aircraft surface**



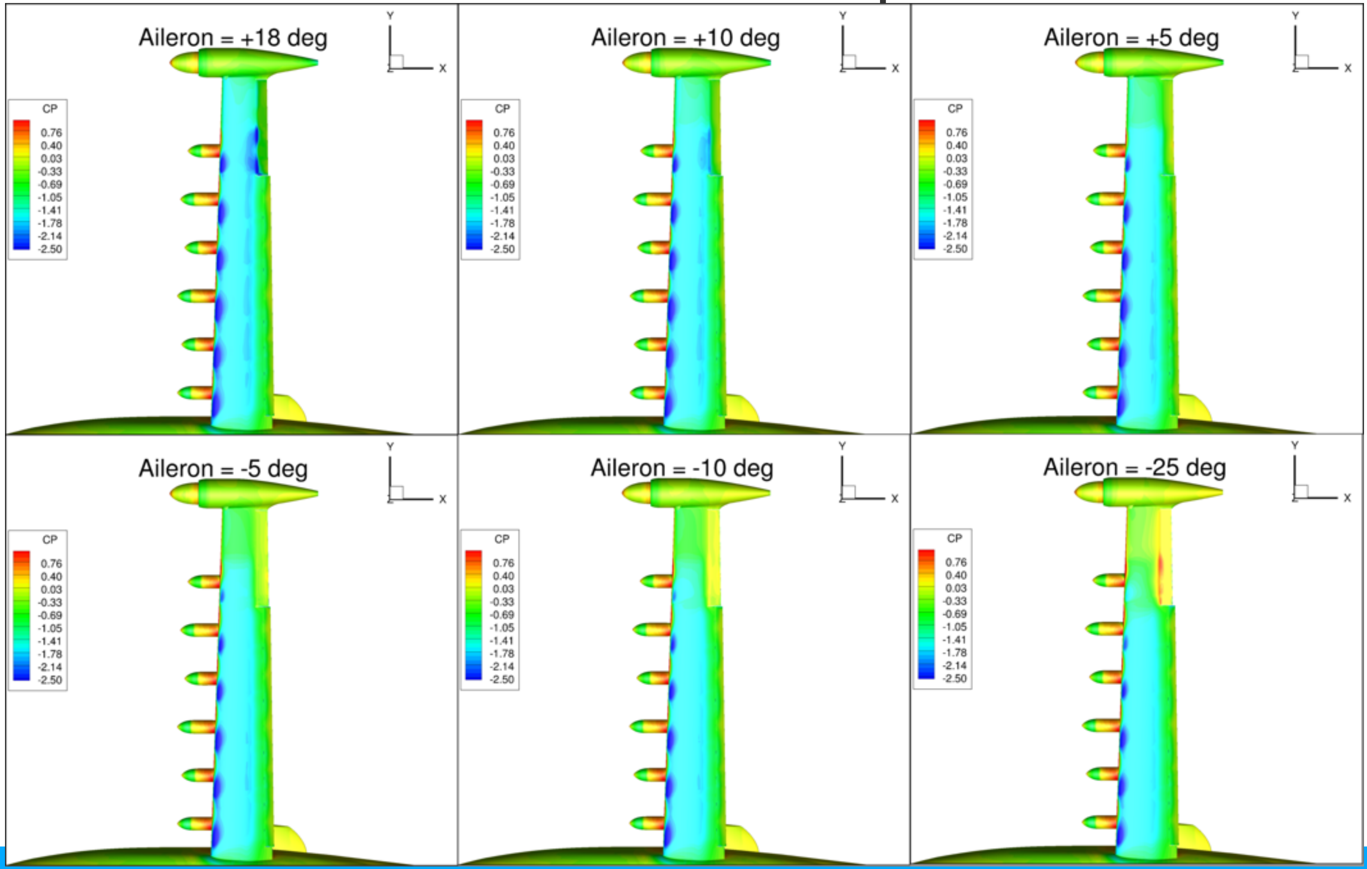
LAVA Cruise Power-On Flow Visualizations

- ***Condition:*** **α = 12.0°, β = 0°, Altitude = 2,500 ft, V_∞ = 150.0 ft/s, Mach = 0.136, Re_{MAC} = 1,921,000**
- **Dimensionless streamwise velocity (U/U_{ref}) is shown on slice plane, pressure coefficient on aircraft surface**



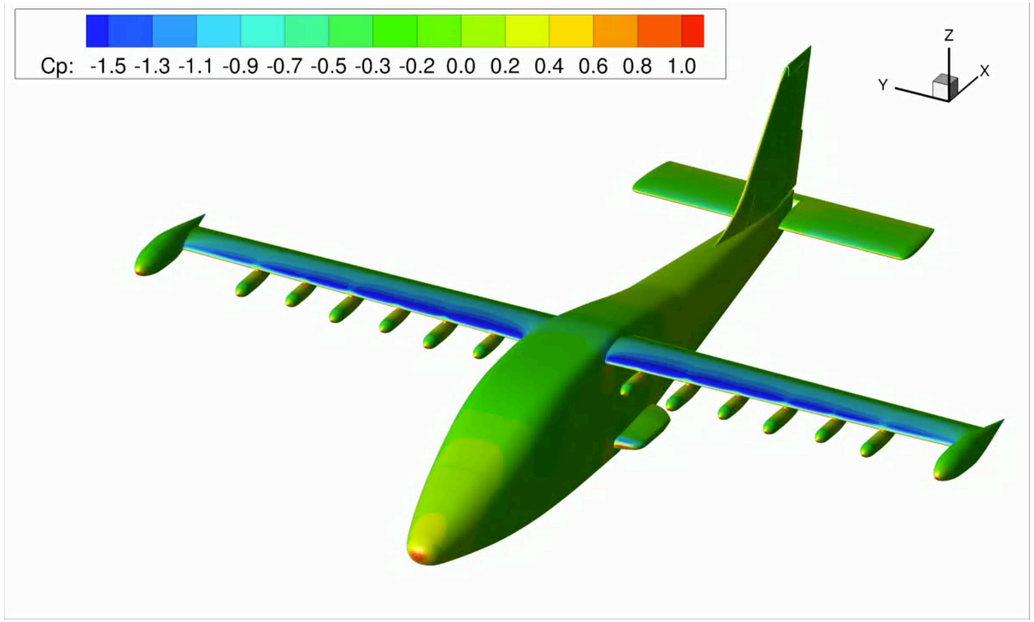
Additional High-Lift Power-On Flow Visualizations

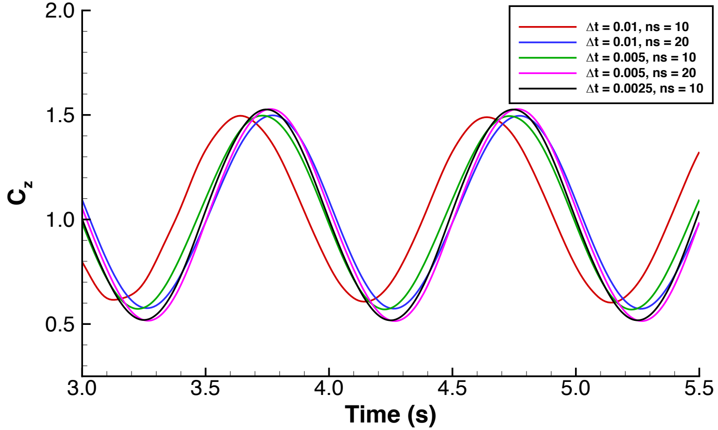
- *Condition:* 58 KCAS, Thrust = 49.3 lbf, Altitude = 2,500 ft, Re = 1.295×10^6, M_∞ = 0.092, α = 2.0°
- Pressure distribution shown on surface for multiple aileron deflection cases

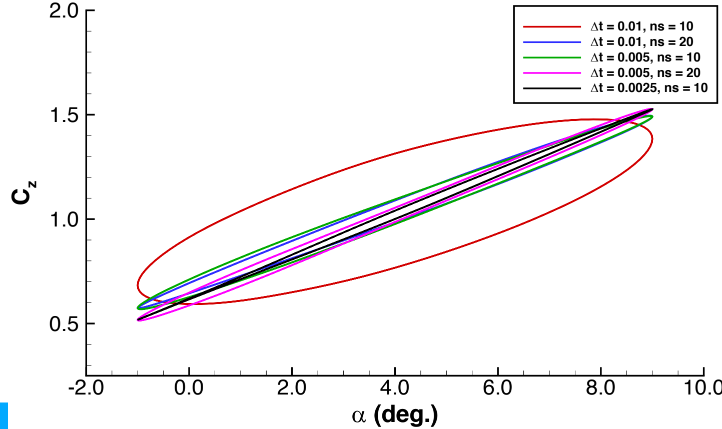


Additional CFD Analysis Tasks

- Additional analysis independent from database work was performed to assess the aircraft's dynamic stability in roll, pitch and yaw
- An additional 30 case database to be completed for analysis
- Unsteady RANS simulations such as these require five days to run on Intel Ivy Bridge nodes totaling 2,000 cores per simulation







Summary of Compute Resources Used

LAVA Curvilinear

- Intel Ivy Bridge E5-2680 Nodes on the Pleiades Supercomputer at NASA Ames Research Center
- 1100-1200 cores were used for all computations presented here, with 900 cores for the ”coarse grid” cases and 1520 cores for the “fine grid” cases
- Compute time: 12-16 hours/case
- Approximately 15 M core hours used to date

Star-CCM+

- Run on a cluster located at NASA Armstrong Research Center
- Calculations performed on various node types and core counts depending on availability, up to 1200 cores
- 100k-200k cells per core were utilized on average
- Compute time: 24-48 hours/case

Image source: *https://www.nasa.gov/centers/ames/news/releases/2010/10-45AR.html*

Slide Master