Computational Simulations of Electric Propulsion Aircraft: the X-57 Maxwell

Jared C. Duensing, Jeffrey A. Housman, Daniel Maldonado, James C. Jensen, Cetin C. Kiris
NASA Ames Research Center

Seung Y. Yoo
NASA Armstrong Flight Research Center

AMS Seminar Series, June 13th, 2019
NASA Ames Research Center, Moffett Field, California
Outline

- Introduction
  - X-57 CFD task overview
  - Motivation

- Part I: Computational simulations without propulsion
  - Establishing CFD Best Practices
    - Grid generation
    - Mesh refinement study
    - Numerical methods
    - Wind tunnel validation study
  - Power-Off Aerodynamic Database Results

- Part II: Computational simulations with propulsion
  - Cruise Power-On Database
  - High-Lift Power-On Database

- Summary

Image source: https://www.nasa.gov/press-release/nasa-electric-research-plane-gets-x-number-new-name
Evolution of the X-57

NASA aims to achieve a 5X reduction in energy consumption for a private plane cruising at 150 knots through the latest 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

- Gearpods
- High-lift takeoff and landing propellers
- Stabilator
- Vertical Tail and Rudder
- Cruise propellers

Image source: https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20170001218.pdf
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.
Steps to Establish Best-Practices Towards X-57 Database

1. Initial grid generation (structured curvilinear and unstructured polyhedral)
2. Free-air mesh refinement study
3. Study effects of:
   a. Low Mach Number Preconditioning
   b. Turbulence Model
   c. Numerical dissipation has been studied
4. Wind tunnel validation studies

Sample X-57 structured overset grid

Sample X-57 structured overset grid in wind tunnel
Introduction

- X-57 CFD task overview
- Motivation

Part I: Computational simulations without propulsion

- Establishing CFD Best Practices
  - Grid generation
  - Mesh refinement study
  - Numerical methods
  - Wind tunnel validation study
- Power-Off Aerodynamic Database Results

Part II: Computational simulations with propulsion

- Cruise Power-On Database
- High-Lift Power-On Database

Summary
Mesh generation for this geometry occurs in three steps

- CAD preparation and clean-up using ANSA (http://www.beta-cae.com/ansa.htm)
- Hyperbolic marching for more complex grids in Chimera Grid Tools (CGT)
- Hyperbolic volume growth in CGT
- Domain connectivity using combination of DCF module in OVERFLOW and LAVA modified implicit hole offset procedure
Grid Generation Guidelines

- Guidelines established by the High-Lift Prediction Workshop 3 (HLPW-3) were used to develop the initial grid system for the pre-database studies
  - Stretching ratio < 1.25
  - Leading edge spacing of 0.1% of the local chord
  - 5 points on all finite-thickness trailing edges
  - Double fringe minimum overlap
- "Coarse" resolution targeted for initial grid for ease in consistent grid refinements
- Special procedure developed to generate grids over moving components

Wing leading edge (pink)  Wing trailing edge (pink)
Mesh Procedure for Moving Geometry

- Future database runs require the articulation of control surfaces to a specified angle
- A surface grid generation procedure was developed in which quadratic Bezier curves join open surfaces following a deflection

\[ \vec{B}(t) = (1 - t)^2 \vec{P}_0 + 2(1 - t)t \vec{P}_1 + t^2 \vec{P}_2 \]

\[ \delta_a = +10^\circ \]

\[ \delta_a = -10^\circ \]
Transfinite interpolation was then used to construct the control surface patch.

The resulting surface was then grown hyperbolically onto adjacent grids to create double fringe overlap.
Spacing and stretching guidelines for surface mesh generation similar to structured curvilinear grids

Volume mesh grown using prismatic cells in the boundary layer and arbitrary polyhedral cells in the far-field
Outline

- Introduction
  - X-57 CFD task overview
  - Motivation

- Part I: Computational simulations without propulsion
  - Establishing CFD Best Practices
    - Grid generation
    - Mesh refinement study
    - Numerical methods
    - Wind tunnel validation study
  - Power-Off Aerodynamic Database Results

- Part II: Computational simulations with propulsion
  - Cruise Power-On Database
  - High-Lift Power-On Database

- Summary
Solver Details

- LAVA (Launch Ascent and Vehicle Aerodynamics) Curvilinear would be the primary flow solver used for this study with the commercial solver Star-CCM+ also used for comparison.

**Structured Curvilinear** (LAVA)
- Node-based steady-state RANS
- Second-order Roe convective flux discretization
- Van Albada flux limiter
- Spalart-Allmaras (SA) turbulence model with RC and QCR-2000

**Unstructured Arbitrary Polyhedral** (Star-CCM+)
- Cell-centered steady-state RANS
- Second-order Roe convective flux discretization
- Venkatakrishnan flux limiter
- Spalart-Allmaras (SA) turbulence model with RC
### Mesh Refinement Study Grid Statistics

**Structured Curvilinear**

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Grid Points</th>
<th># of Zones</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>80.0 M</td>
<td>315</td>
</tr>
<tr>
<td>Medium</td>
<td>159 M</td>
<td>315</td>
</tr>
<tr>
<td>Fine</td>
<td>322 M</td>
<td>315</td>
</tr>
</tbody>
</table>

**Unstructured Arbitrary Polyhedral**

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Grid Points</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>17.2 M</td>
</tr>
<tr>
<td>Medium</td>
<td>49.7 M</td>
</tr>
<tr>
<td>Fine</td>
<td>120 M</td>
</tr>
</tbody>
</table>

**Structured Coarse**

**Structured Medium**

**Structured Fine**
Mesh Refinement Reference Conditions

- Case selected based on experimental data collected in the 12-foot low-speed wind tunnel at NASA Langley Research Center

- **Objective:** Use this condition and experimental data to determine appropriate solver settings and grid refinement level for future simulations

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach Number</td>
<td>0.052</td>
</tr>
<tr>
<td>Reynolds Number (based on MAC)</td>
<td>121,600</td>
</tr>
<tr>
<td>Reference Static Temperature</td>
<td>288.1 K</td>
</tr>
<tr>
<td>Angle of Attack</td>
<td>6.0 °</td>
</tr>
<tr>
<td>Sideslip Angle</td>
<td>5.0 °</td>
</tr>
<tr>
<td>Aileron Deflection*</td>
<td>-10.0 °</td>
</tr>
<tr>
<td>Rudder Deflection*</td>
<td>-15.0 °</td>
</tr>
<tr>
<td>Stabilator Deflection*</td>
<td>-7.5 °</td>
</tr>
</tbody>
</table>

*Negative value corresponds to a trailing edge up (right for rudder) deflection*
Selecting Mesh Refinement Levels

- Second-order asymptotic convergence observed for drag, with the extrapolated drag for an “infinitely fine” grid within 1.1% error relative to other solver

<table>
<thead>
<tr>
<th>Grid</th>
<th>LAVA Curvilinear</th>
<th>Star-CCM+</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>CL</td>
<td>CD</td>
</tr>
<tr>
<td>Coarse</td>
<td>1.0072</td>
<td>0.1138</td>
</tr>
<tr>
<td>Medium</td>
<td>1.0070</td>
<td>0.1121</td>
</tr>
<tr>
<td>Fine</td>
<td>1.0070</td>
<td>0.1115</td>
</tr>
<tr>
<td>Experiment</td>
<td>1.068</td>
<td>0.099</td>
</tr>
</tbody>
</table>

![Graph showing convergence of drag coefficients](image)
Closer analysis of the pressure distribution at selected wing locations show strong agreement across all three refinement levels.

Virtually no change in pressure distribution between medium and fine levels.
Outline

- Introduction
  - X-57 CFD task overview
  - Motivation

- Part I: Computational simulations without propulsion
  - Establishing CFD Best Practices
    - Grid generation
    - Mesh refinement study
    - Numerical methods
    - Wind tunnel validation study
  - Power-Off Aerodynamic Database Results

- Part II: Computational simulations with propulsion
  - Cruise Power-On Database
  - High-Lift Power-On Database

- Summary
Numerical discretization is a second-order convective flux scheme with van Albada slope limiter.

Coarse level grid was used to determine whether flow is best modeled fully turbulent or fully laminar (unless determined necessary, transitional flow would not be modeled).

Effects of low-Mach preconditioning would be tested due to low speed flow.

Additional LAVA Unstructured simulations were also run to ensure consistency of solver settings within the LAVA framework.

<table>
<thead>
<tr>
<th>Modeling Approach</th>
<th>LAVA Curvilinear</th>
<th>LAVA Unstructured</th>
</tr>
</thead>
<tbody>
<tr>
<td>Laminar without Preconditioning</td>
<td>0.579</td>
<td>0.151</td>
</tr>
<tr>
<td>Laminar with Preconditioning</td>
<td>0.585</td>
<td>0.146</td>
</tr>
<tr>
<td>SA Turbulence without Preconditioning</td>
<td>1.015</td>
<td>0.119</td>
</tr>
<tr>
<td>SA Turbulence with Preconditioning</td>
<td>1.007</td>
<td>0.114</td>
</tr>
<tr>
<td>Experiment</td>
<td>1.068</td>
<td>0.099</td>
</tr>
</tbody>
</table>
Outline

- Introduction
  - X-57 CFD task overview
  - Motivation

- Part I: Computational simulations without propulsion
  - Establishing CFD Best Practices
    - Grid generation
    - Mesh refinement study
    - Numerical methods
    - Wind tunnel validation study
  - Power-Off Aerodynamic Database Results

- Part II: Computational simulations with propulsion
  - Cruise Power-On Database
  - High-Lift Power-On Database

- Summary
Medium grid for both mesh paradigms was determined to be sufficiently fine based on lift and drag values relative to converged values.

Error relative to experiment appears to be reduced as much as possible due to modeling and mesh variations.

Other possible sources of error relative to experiment due to uncorrected experimental data provided:
- Wind tunnel wall interference
- Buoyancy effects
- Mounting fixture interaction

A component build-up is desired to verify this theory and improve validation results.
Wind Tunnel Validation Reference Conditions

- Case selected based on experimental data collected in the 12-foot low-speed wind tunnel at NASA Langley Research Center

- **Objective:** Perform CFD simulations of increasing fidelity to reduce error relative to experiment for validation cases

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach Number</td>
<td>0.052</td>
</tr>
<tr>
<td>Reynolds Number (based on MAC)</td>
<td>121,600</td>
</tr>
<tr>
<td>Reference Static Temperature</td>
<td>288.1 K</td>
</tr>
<tr>
<td>Angle of Attack</td>
<td>2.0 °</td>
</tr>
<tr>
<td>Sideslip Angle</td>
<td>0.0 °</td>
</tr>
<tr>
<td>Aileron Deflection</td>
<td>0.0 °</td>
</tr>
<tr>
<td>Rudder Deflection</td>
<td>0.0 °</td>
</tr>
<tr>
<td>Stabilator Deflection</td>
<td>-15.0 °</td>
</tr>
</tbody>
</table>
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.
Tunnel simulation designed to emulate blockage effects of wind tunnel hardware

Test-section geometry extended 50 body lengths upstream and downstream (excludes inlet, diffuser, and surrounding recirculation chamber)

Image source: https://researchdirectorate.larc.nasa.gov/12-foot-low-speed-tunnel-12-ft-lst/
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

- Increased flow acceleration with wall blockage included
- Sharp flow deceleration near vehicle aft end due to sting interference
Validation Simulation Results ($C_p$ Contours on Aircraft Surface)

- 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

Increased flow acceleration over upper surface creates suction-induced lift
Pressure distribution at a selected spanwise wing location is compared for each build-up level.

Progressively increased pressure differential is seen as components are added.

Slight impact with the addition of sting and most substantial impact occurs with the wind tunnel walls.

### Build-Up Simulation Results

<table>
<thead>
<tr>
<th>Case</th>
<th>$C_L$</th>
<th>% err. $C_L$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Free Air</td>
<td>0.4575</td>
<td>20.6</td>
</tr>
<tr>
<td>Free Air + Sting</td>
<td>0.4782</td>
<td>17.8</td>
</tr>
<tr>
<td>Wind Tunnel + Sting + C-strut</td>
<td>0.5394</td>
<td>6.4</td>
</tr>
<tr>
<td>Experiment</td>
<td>0.5762</td>
<td>-</td>
</tr>
</tbody>
</table>
Most substantial reduction of error is seen through lift coefficient, where free-air modeling error of 20.6% was reduced to 6.4% for LAVA Curvilinear when simulated appropriately

Negligible change in drag error relative to experiment

<table>
<thead>
<tr>
<th>Case</th>
<th>LAVA Curvilinear</th>
<th>Star-CCM+ Unstructured</th>
</tr>
</thead>
<tbody>
<tr>
<td>Free Air</td>
<td>0.4575</td>
<td>20.6</td>
</tr>
<tr>
<td>Free Air + Sting</td>
<td>0.4782</td>
<td>17.8</td>
</tr>
<tr>
<td>Wind Tunnel + Sting + C-strut</td>
<td>0.5394</td>
<td>6.4</td>
</tr>
<tr>
<td>Experiment</td>
<td>0.5762</td>
<td>-</td>
</tr>
</tbody>
</table>
Comparison of multiple angles of attack in free air and with wind tunnel hardware further demonstrate modeling impacts

For both codes, incorporating wind tunnel effects to the CFD simulation improve lift predictions considerably across the linear regime of the $C_L$ vs. $\alpha$ curve

Changes observed for drag and pitching moment, however no considerable change in accuracy

$FS = \text{Free Stream}$

$Wall = \text{Wind Tunnel} + \text{Sting} + \text{C-strut}$
Minor changes noted in drag and pitching moment in the pre-stall regime, but no significant change in error relative to experiment

Larger scatter in results observed post-stall independent of modeling approach

FS = Free Stream
Wall = Wind Tunnel + Sting + C-strut
Outline

- Introduction
  - X-57 CFD task overview
  - Motivation

- Part I: Computational simulations without propulsion
  - Establishing CFD Best Practices
    - Grid generation
    - Mesh refinement study
    - Numerical methods
    - Wind tunnel validation study
  - Power-Off Aerodynamic Database Results

- Part II: Computational simulations with propulsion
  - Cruise Power-On Database
  - High-Lift Power-On Database

- Summary
Findings from mesh refinement and modeling approach studies are then applied to the power-off aerodynamic database (188 RANS runs)

- Models flight performance for three flap settings
  - Take-off (10º deflection)
  - Cruise (0º deflection)
  - Landing (30º deflection)

Preliminary results for nominal cruise setting are presented in the following slides, more detailed database results will be presented in future publications.
Updates to existing wind tunnel grid were made to reflect new high-lift pylon design and added vortex generator.

Angles of attack were selected to determine aerodynamic performance with all control surfaces at their nominal position.

### AoA Sweep at Cruise Condition

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach Number</td>
<td>0.233</td>
</tr>
<tr>
<td>Reynolds Number based on MAC</td>
<td>2,790,000</td>
</tr>
<tr>
<td>Reference Static Temperature</td>
<td>288.1 K</td>
</tr>
<tr>
<td>Sideslip Angle</td>
<td>0.0 °</td>
</tr>
<tr>
<td>Aileron Deflection</td>
<td>0.0 °</td>
</tr>
<tr>
<td>Rudder Deflection</td>
<td>0.0 °</td>
</tr>
<tr>
<td>Stabilator Deflection</td>
<td>0.0 °</td>
</tr>
<tr>
<td>Flap Deflection</td>
<td>0.0 °</td>
</tr>
</tbody>
</table>

**Grid Points**: 139 M  
**Zones**: 333 grids
LAVA predicts lift within 0.5% of Star-CCM+ for pre-stall AoA, and within 8% post-stall.

This level of agreement is a result of the work completed prior to the database.

RANS simulations after the stall angle of attack require higher fidelity models (e.g., LES, hybrid RANS/LES, etc.). Current simulation methods are not valid in this regime.

*Cruise Condition AoA Sweep Results

*LAVA* predicted lift is within 0.5% of Star-CCM+ for pre-stall AoA, and within 8% post-stall.

This level of agreement is a result of the work completed prior to the database.

RANS simulations after the stall angle of attack require higher fidelity models (e.g., LES, hybrid RANS/LES, etc.). Current simulation methods are not valid in this regime.

Cruise Condition AoA Sweep Results

- LAVA Curvilinear and Star-CCM+ predict drag and pitching moment within 5% of each other pre- and post-stall.
Closer examination of flow structures at selected angles of attack confirm the similar predictive capabilities of LAVA Curvilinear and Star-CCM+

- **Note:** Particle seeding for streamlines randomly distributed for each figure

Pre-stall angles of attack predict attached flow with minor separation developing at high-lift pylon locations

16° angle of attack, LAVA Curvilinear

16° angle of attack, Star-CCM+
Establishing best practices prior to database generation is an important step to ensure success in future simulations

- Develop efficient mesh generation techniques
- Determine refinement level
- Determine numerical schemes
- Validate against experimental data

Properly establishing these best-practices minimizes the CFD error and can be applied to similar geometries

- Proper simulation techniques can potentially reduce CFD error relative to experiment from above 20% to about 6% as shown with lift coefficient
- Code-to-code error in aerodynamic loads as low as 0.5% was observed, as well as a prediction of $C_{L_{\text{max}}}$ within 0.6%

These concepts will be applied to X-57 simulations that include propulsion
Introduction
- X-57 CFD task overview
- Motivation

Part I: Computational simulations without propulsion
- Establishing CFD Best Practices
  - Grid generation
  - Mesh refinement study
  - Numerical methods
  - Wind tunnel validation study
- Power-Off Aerodynamic Database Results

Part II: Computational simulations with propulsion
- Cruise Power-On Database
- High-Lift Power-On Database

Summary
Best practices from the power-off CFD database preparation were also applied to power-on database simulations:

- Mesh refinement studies
- Numerical scheme/turbulence model determination
- Code-to-code comparison

Refinement level and numerical schemes/models identical to power-off simulations deemed adequate.

Additional studies need to be performed to determine best propulsion modeling approach.
Developing the X-57 flight simulator requires quantifying the aerodynamic loads for a variety of power-on flight scenarios
- Cruise propellers only
- High lift propellers only
- Failure scenarios

Mod-III and Mod-IV power-on aero databases study the effects of propulsion and quantify the “aero deltas” relative to power-off simulations.

Cruise propellers (Mod-III)
High lift propellers (Mod-IV)
Bus failure scenario (Mod-IV)
Propulsion Modeling Approach

- **Actuator zones** model the momentum imparted from the propeller to the surrounding flow field without the computational cost of simulating the moving blade.
- Axial forces (thrust) and tangential forces (torque) as a function of propeller radius are needed to define the actuator zone model.
- Radial thrust and torque distribution options studied with LAVA:
  - Constant
  - Goldstein Optimum
  - Arbitrary

\[ F_x \text{ (per blade)} \ (N) \]

\[ r/R_{prop} \]

Comparison of Thrust Distribution Profiles (High Lift)

Outline

➢ Introduction
  ➢ X-57 CFD task overview
  ➢ Motivation

➢ Part I: Computational simulations without propulsion
  ➢ Establishing CFD Best Practices
    ➢ Grid generation
    ➢ Mesh refinement study
    ➢ Numerical methods
    ➢ Wind tunnel validation study
  ➢ Power-Off Aerodynamic Database Results

➢ Part II: Computational simulations with propulsion
  ➢ Cruise Power-On Database
  ➢ High-Lift Power-On Database

➢ Summary
Cruise Power-On Simulation Method

- Similar preparations were performed for power-on simulations as power-off
  - Mesh refinement study
  - Numerical dissipation
  - Turbulence model corrections

- Selected solver settings
  - Steady-state RANS
  - Second-order convective flux with Koren limiter
  - SA turbulence with RC/QCR2000 correction enabled

- Actuator zone modeling
  - Cruise propellers are modeled with actuator zone source terms using the Goldstein radial thrust and torque distributions
The Mod-III power-on database simulates 10 flight conditions, for each of which the control surfaces and power settings would be varied.

Power-off mesh modified to include actuator zone regions where the propulsion source term would be applied.

Sample results shown will be for the following condition:
- Altitude: 2,500 ft, $V_\infty = 150.0$ ft/s, $Mach = 0.136$, $Re_{MAC} = 1,921,000$

**Grid points:**
- 120.9 M
- 128.7 M
Cruise Power-On Simulation Sample Results

- Altitude: 2,500 ft, $V_\infty = 150.0$ ft/s, Mach = 0.136, $Re_{MAC} = 1,921,000$
- Lift increases linearly with angle of attack until near stall, and shifts upward with power setting
- Drag increases parabolically with angle of attack, and shifts downward with increased thrust and slightly upward near 16 degrees AoA around stall
Cruise Power-On Simulation Sample Results

- Altitude: 2,500 ft, $V_\infty = 150.0$ ft/s, Mach = 0.136, $\text{Re}_{\text{MAC}} = 1,921,000$
- Nose-down pitching moment increases with applied thrust, and with angle of attack

![Pitching Moment](image_url)
Condition: $\alpha = 12.0^\circ$, $\beta = 0^\circ$, 
Altitude = 2,500 ft, $V_\infty = 150.0$ ft/s, Mach = 0.136, $\text{Re}_{\text{MAC}} = 1,921,000$

Dimensionless streamwise velocity ($U/U_{\text{ref}}$) is shown on slice plane, pressure coefficient on aircraft surface
Condition: $\alpha = 12.0^\circ$, $\beta = 0^\circ$, Altitude = 2,500 ft, $V_\infty = 150.0$ ft/s, Mach = 0.136, $Re_{MAC} = 1,921,000$

- Dimensionless streamwise velocity ($U/U_{ref}$) is shown on slice plane

Cruise Power-On Flow Visualizations

- 0% Power
- 6% Power
- 80% Power
- 125% Power
Introduction

- X-57 CFD task overview
- Motivation

Part I: Computational simulations without propulsion

- Establishing CFD Best Practices
  - Grid generation
  - Mesh refinement study
  - Numerical methods
  - Wind tunnel validation study
- Power-Off Aerodynamic Database Results

Part II: Computational simulations with propulsion

- Cruise Power-On Database
- High-Lift Power-On Database

Summary
Selecting Actuator Zone Distributions

- For high-lift propulsion cases, XROTOR\textsuperscript{2} data was available to define an arbitrary radial thrust and torque distribution.
- Initial CFD simulations were performed using LAVA to understand impact of thrust and torque distributions on the solution (Altitude: 6000 ft., Re\textsubscript{MAC} = 1,235,000, Mach = 0.098, $\alpha = 10^\circ$).

Initial CFD simulations were performed using LAVA to understand impact of thrust and torque distributions on the solution.

Altitude: 6000 ft., $\text{Re}_{\text{MAC}} = 1,235,000$, Mach = 0.098, $\alpha = 16^\circ$ shown below.

Separation behavior at high angle of attack highly dependent on thrust and torque distribution.

**Selecting Actuator Zone Distributions**

- **Constant Thrust and Torque**
- **Goldstein Thrust and Torque**
- **XROTOR Thrust and Torque**
High-Lift Power-On Simulation Method

- **Solver settings**
  - Steady-state RANS
  - Second-order convective flux with Koren limiter
  - SA turbulence with RC/QCR2000 correction enabled

- **Actuator zone modeling**
  - All high-lift propellers are modeled with actuator zone source terms that utilize custom XROTOR radial thrust and torque distributions for a given flight condition
  - Sample distributions shown for 3,962 RPM, Mach 0.119, 2500 ft. altitude condition
High-Lift Power-On Power-On Sample Simulations

- The Mod-IV power-on database simulates 10 flight conditions, at which the control surfaces, flap deflections, and high lift power settings would be varied.
- Power-off mesh modified to include actuator zone regions where the propulsion source term would be applied.
- Sample results shown will be for the following condition:
  - Altitude: 2,500 ft, Mach = 0.119, $Re_{MAC} = 1,682,000$, Flaps = 30°

Grid points: 154.4 M  
Grid points: 171.1 M
High-Lift Power-On Sample AoA Sweep Results

- Altitude: 2,500 ft, Mach = 0.119, $\text{Re}_{\text{MAC}} = 1,682,000$, Flaps = 30°
- Solid curves show high-lift power-on results, dashed curves show power-off for comparison
- LAVA predicts power-on lift within 1.4% of Star-CCM+ pre-stall, and within 8.5% post-stall
- LAVA predicts power-on drag within 3.1% of Star-CCM+ pre-stall, and within 5.7% post-stall
Part II Summary

- The best practices established during power-off simulations were applied to database simulations that include propulsion
  - Determined mesh refinement level
  - Determined numerical schemes

- Selected best propulsion modeling method
  - Thrust and torque distributions used for simulation have a large impact on flight performance, particularly near stall

- Strong code-to-code agreement pre-stall persists when propulsion is included in the simulation
  - Maximum difference in lift and drag between codes pre-stall for all high-lift motors powered on is observed to be 1.4% and 3.1%, respectively
  - $C_{L,\text{max}}$ value is predicted to within 0.4% and within $\sim1.0^\circ$ angle of attack
Summary of 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

- **Star-CCM+**
  - Run on a cluster located at NASA Armstrong Research Center
  - Calculations performed on various node types and core counts depending on availability
  - 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*
Acknowledgments

- This project is funded by the Scalable Convergent Electric Propulsion Technology and Operations Research (SCEPTOR) program under the NASA Aeronautics Research Mission Directorate (ARMD)
- Michael Frederick and Trong Bui of the NASA Armstrong Aerodynamics and Propulsion Research Branch
- Experimental data provided by David Cox at NASA Langley Research Center
- Computer resources provided by NASA Advanced Supercomputing (NAS) Pleiades facility
Questions?
Supplemental Slides
Cruise Condition AoA Sweep Results

- At stall, higher separation at 50% chord and aft predicted with both codes and exaggerated at high-lift pylon locations

4° angle of attack, LAVA Curvilinear

4° angle of attack, Star-CCM+
Cruise Condition AoA Sweep Results

- Post-stall, near total flow separation predicted for entirety of the wing with small pockets of attachment for both codes

22° angle of attack, LAVA Curvilinear

22° angle of attack, Star-CCM+