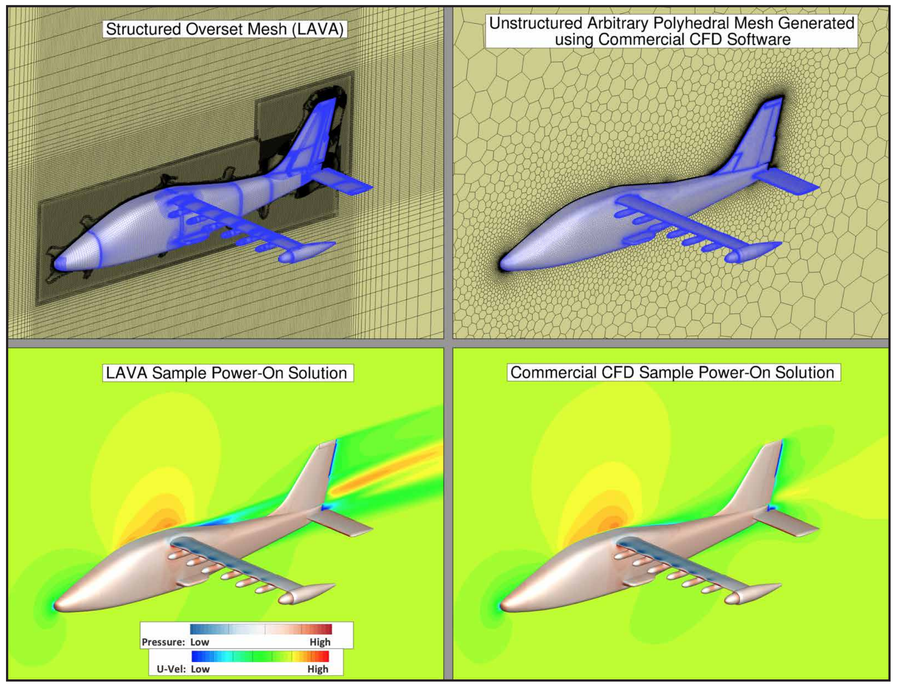
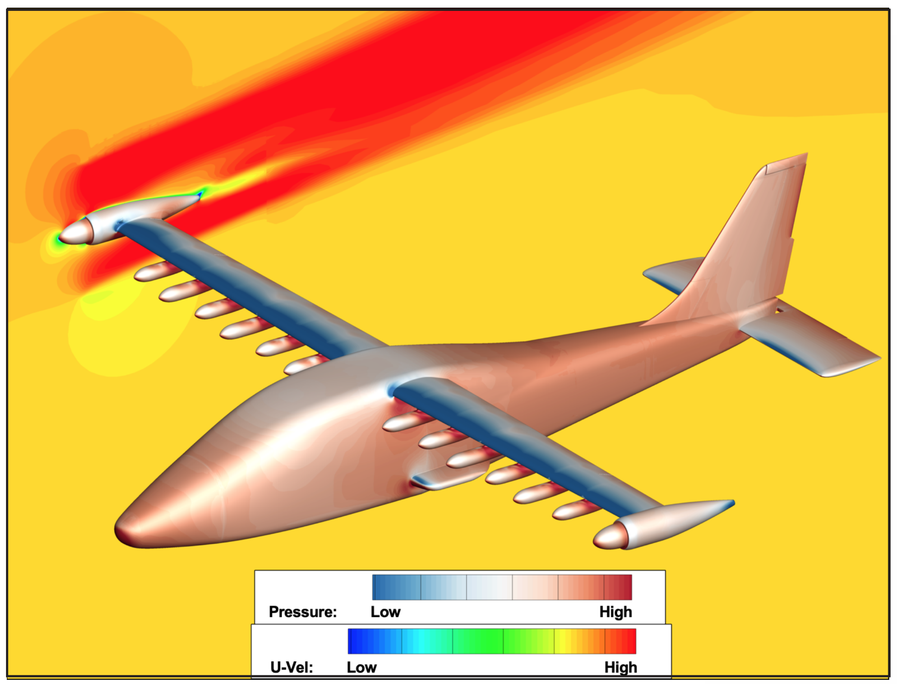


Computational meshes (top) are shown with corresponding CFD solutions (bottom), where pressure is shown on NASA's X-57 "Maxwell" aircraft surface and velocity is shown on the symmetry plane. Multiple grid paradigms and flow solvers have been used for CFD analysis, including the LAVA flow solver and the Star-CCM+ commercial flow solver. Despite discrepancies in the wake due to mesh resolution differences, pressure distributions between the solvers agree very well and yield consistent integrated quantities. *Jeff Housman, NASA/Ames, Seung Yoo, NASA/Armstrong*



Modeling aircraft propulsion efficiently and accurately is key to understanding the aerodynamic effects of the X-57's high-lift and cruise propellers. Using actuator zones—a method to replicate the propulsion effects of moving propellers—balances computational cost and accuracy in the power-on CFD simulation case. In this simulation image, pressure is shown on the aircraft surface and streamwise velocity is shown on the planar slice for selected cruise propeller settings. *Jared Duensing, NASA/Ames*

National Aeronautics and Space Administration

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

NASA's X-57 "Maxwell" electric aircraft aims to achieve a fivefold reduction in energy consumption compared to conventional aircraft propulsion. Aeronautics research teams from NASA's Ames Research Center, Armstrong Flight Research Center, and Langley Research Center are collaborating to construct multiple aerodynamic databases in order to develop a stability and control model for the X-57 flight simulator. Populating this database requires computational fluid dynamics (CFD) simulation of multiple flight conditions, aircraft configurations, and propulsion settings. To date, 800 cases have been completed and more than 2,000 additional simulations will be run using the agency's Launch Ascent and Vehicle Aerodynamics (LAVA) flow solver.

Jared Duensing, Cetin Kiris, NASA Ames Research Center