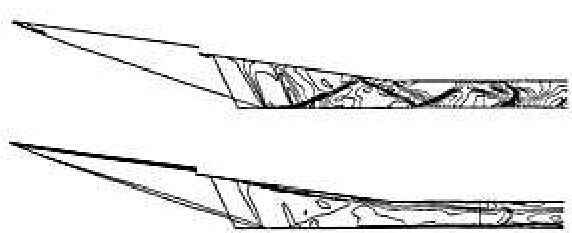
Computational Fluid Dynamics Analysis Method Developed for Rocket-Based Combined Cycle Engine Inlet

Renewed interest in hypersonic propulsion systems has led to research programs investigating combined cycle engines that are designed to operate efficiently across the flight regime. The Rocket-Based Combined Cycle Engine is a propulsion system under development at the NASA Lewis Research Center. This engine integrates a high specific impulse, low thrust-to-weight, airbreathing engine with a low-impulse, high thrust-to-weight rocket. From takeoff to Mach 2.5, the engine operates as an air-augmented rocket. At Mach 2.5, the engine becomes a dual-mode ramjet; and beyond Mach 8, the rocket is turned back on. One Rocket-Based Combined Cycle Engine variation known as the "Strut-Jet" concept is being investigated jointly by NASA Lewis, the U.S. Air Force, Gencorp Aerojet, General Applied Science Labs (GASL), and Lockheed Martin Corporation. Work thus far has included wind tunnel experiments and computational fluid dynamics (CFD) investigations with the NPARC code.

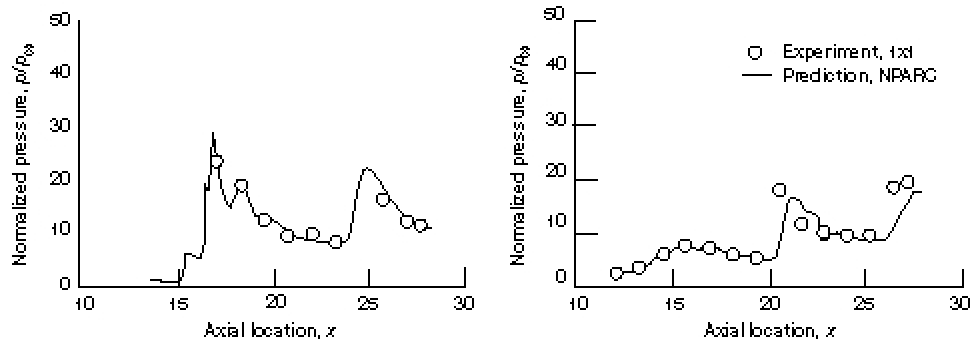
The CFD method was initiated by modeling the geometry of the Strut-Jet with the GRIDGEN structured grid generator. Grids representing a subscale inlet model and the full-scale demonstrator geometry were constructed. These grids modeled one-half of the symmetric inlet flow path, including the precompression plate, diverter, center duct, side duct, and combustor. After the grid generation, full Navier-Stokes flow simulations were conducted with the NPARC Navier-Stokes code. The Chien low-Reynolds-number k-ε turbulence model was employed to simulate the high-speed turbulent flow. Finally, the CFD solutions were postprocessed with a Fortran code. This code provided wall static pressure distributions, pitot pressure distributions, mass flow rates, and internal drag. These results were compared with experimental data from a subscale inlet test for code validation; then they were used to help evaluate the demonstrator engine net thrust.



Subscale inlet pressure ($p/p_{infinity}$) and Mach number contours for a supercritical flow of $M_{infinity}$ = 5. Top: pressure contour ranges of 1.4 to 32.2. Bottom: Mach number contour ranges of 0.2 to 5.0.

The top contour plot shows contours of static pressure on the inlet centerplane of the subscale inlet. These contours indicate a series of strong oblique shocks initiated by the precompression plate. Mach number contours in the bottom contour plot indicate a large region of low-speed flow along the body side of the inlet resulting from a shock-induced boundary layer separation.

The following graphs compare the static pressures obtained from the NPARC code to experimental measurements made in NASA Lewis' 1- by 1-Foot Supersonic Wind Tunnel. Very good agreement is observed for the pressures along the cowl and body centerline.



Subscale inlet pressure ($p/p_{infinity}$) distribution for a supercritical flow of $M_{infinity}$ = 5 in the NASA Lewis 1- by 1-Foot Supersonic Wind Tunnel (1x1). Left: Cowl centerline. Right: Body centerline.

After good agreement was observed between the CFD solutions and subscale wind tunnel experimental data, the NPARC code was applied to the demonstrator engine to provide pretest predictions. Pressure distributions and internal drag force calculations were obtained to guide the demonstrator engine tests. The CFD method developed here (including grid generation, flow computation, and postprocessing) will allow for analyses of future combined-cycle engine concepts.

For more information, visit the NPARC Alliance.

Bibliography

DeBonis, J.R.; and Yungster, S.: Rocket-Based Combined Cycle Engine Technology Development: Inlet CFD Validation and Application. AIAA Paper 96-3145 (NASA TM-107274), 1996.