

Aerospace Engineering Master's Defense

Parametric Analysis of Hypersonic Inlet
using Computational Fluid Dynamics

School for Engineering of Matter, Transport and Energy

Daniel Oliden

Advisor: Dr. Tae-Woo Lee

abstract

For CFD validation, hypersonic flow fields are simulated and compared with experimental data specifically designed to recreate conditions found by hypersonic vehicles. Simulated flow fields on a cone-ogive with flare at Mach 7.2 are compared with experimental data from NASA Ames Research Center 3.5" hypersonic wind tunnel to study viscous interaction effects. Simulated flow fields on a blunt cone with flare at Mach 10 are compared with experimental data from CUBRC LENS hypervelocity shock tunnel to study vibrational and chemical non-equilibrium effects. Following the validation studies is a parametric analysis of a hypersonic inlet from Mach 6 to 20. Compressor performance is investigated for various cowl leading edge locations up to Mach 10. Investigation of the hypersonic inlet at different altitudes is performed to study the effects of Reynolds number, and consequently, turbulent viscous effects on compressor performance. Mach numbers up to 20 are investigated to study the effects of vibrational and chemical non-equilibrium on compressor performance



November 20, 2013; 1:00 PM; ISTB4 596