Application of CFD to High Speed External Flow: Part I Benchmark CFD simulation

Weerawut Charubhun*, Navapan Nutkumhang, Weerachat Kulsirikasem

Defence Technology Institute
47/433, 4th Floor, Office of the Permanent Secretary of Defence Building
Changwattana Road, Pakkred, Nonthaburi 11120, Thailand
Tel: (66)-2-980-6612 to 15, Fax: (66)-2-980-6688 ext 300

* Contact person: weerawut.c@dti.or.th, (66)-89-815-5540

Abstract

Computational Fluid Dynamics (CFD) is being used as a design tool in military engineering systems such as missile. In present work, the commercial codes such as Fluent and ICEM CFD were applied to facilitate the evaluation of aerodynamic performance for a projectile body. Time-invariant numerical simulations of flow over a projectile body were performed. In Part I, CFD prediction of projectile body using Steady Reynolds-Averaged Navier-Stokes (SRANS) equations with commonly-used turbulence modes, standard k-epsilon and k-omega, was investigated at Mach numbers ranging from 0.6 to 4. An aerodynamic drag force at zero angle of attack was selected for computational and experimental comparisons. The SRANS simulations showed that the predictive abilities of both turbulence models to predict the drag forces are excellent (< 20% and < 5% different from experimental ones in all compressible flow regimes with and without mesh improvement, respectively). The numerical visualization revealed the shock developments and their configurations depended upon the compressible flow regimes.