

Validation of the new modeling capabilities of the Ansys Fluent CFD high-speed solver for the simulation of supersonic combustion in SCRAMjets and Rotating Detonation Engines

Bruce Crawford, Ishan Verma, Stefano Orsino, Jean-Sebastien Cagnone, and Shaoping Li

Ansys, Inc, Canonsburg, PA, 15317, USA

With the rise in interest in new hypersonic propulsion systems in the past few years, there has been a parallel push to improve the predictive capabilities of numerical tools such as CFD; these tools provide unmatched capabilities for the design and analysis of such propulsion systems. In the present work we focus on the recently-expanded capabilities of the Ansys Fluent CFD high-speed Density Based Navier-Stokes (DBNS) solver to improve i) the solver speed and ii) the fidelity for the modeling and design of supersonic combustion ramjet engines (SCRAMjets) and rotating detonation engines (RDE).

The summary of the work being presented here will show the validation of the new capabilities of Ansys Fluent CFD. The DLR, Burrows-Kurkov Scramjet, and AFRL/NETL RDE experiment tests and model geometries are used for the CFD modeling and results validation. DLR scramjet and AFRL RDE geometries are meshed using Ansys Fluent Poly-Hexcore Mosaic meshing, in which these meshes are used to assess the global combustion model with the direct source method solution speed capabilities and using detail chemistry kinetic mechanism. The validation cases presented use the spectrum of turbulence models for the validation, with the Realizable KE, K-Omega SST, and LES.

submissions@iccf.org

Call for Abstracts for ICCFD11