Coupled Aerothermal Analysis of a Rocket Geometry using Conjugate Heat Transfer in OpenFOAM

Abstract

This case study presents a computational aerothermal analysis of a simplified rocket geometry using the open-source CFD toolbox OpenFOAM. The focus of the simulation is on predicting the heat transfer from high-speed airflow around the rocket surface into its solid structure, representing skin and internal components. The study employs the chtMultiRegionFoam solver to model conjugate heat transfer between the fluid and solid domains. The rocket's external flow is simulated under steady-state conditions using turbulent airflow, while the internal conduction is resolved through the solid region. Boundary conditions are applied to replicate realistic operating conditions, including aerodynamic heating from atmospheric re-entry or high-speed ascent. The simulation workflow includes STL preparation, region decomposition, meshing with snappyHexMesh, solver configuration, and post-processing in ParaView. The results include surface temperature distribution, internal thermal gradients, and local heat flux, providing insights into the effectiveness of thermal shielding and highlighting critical hot spots. This study demonstrates the applicability of OpenFOAM in multiphysics simulations relevant to aerospace thermal management systems.