Workflow Development for CFD Analysis on An Aerospace S-Duct

Ishan Nande

Application Engineer, ishan.n@beta-cae.in/+91-9689144433,

BETA CAE Systems India Pvt. Ltd(Software Company),

Vascon Platinum Square, Unit 202, Nagar Rd, Sakore Nagar, Viman Nagar, Pune, Maharashtra 411014, India

Ducts are encountered widely in aerospace applications ranging from HVAC systems to jet engines. For jet engines specifically, S-shaped ducts act as airflow inlet paths. They ensure that the incoming free stream airflow is provided to the engine with minimal flow distortion and pressure losses. The need for such ducts is pronounced in upcoming aircraft designs, as the world actively explores the possibility of Blended Wing Body concepts for commercial needs.

This case study utilizes capabilities of the ANSA pre-processor, OpenFOAM, and the META post-processor to perform CFD analysis on an S-duct. The goal here is to demonstrate a workflow that makes use of three different software to simulate compressible subsonic internal flow. The setup of this study utilizes the geometry as well the flow conditions used in the AIAA's 3rd Propulsion and Aerodynamics Workshop. The numerical setup makes use of the *rhoSimpleFoam* solver, SST k-ω turbulence model for wall resolved flows, combined with a block-structured mesh generated using the HexaBlock tool in ANSA. Finally, the pressure loss and flow distortion parameters obtained from simulations are to be compared with the available experimental data. This would be done using META to validate the overall approach.