Aerospace Europe Conference 2023
Joint 10th EUCASS – 9th CEAS Conference

Abstract #XXX *(to be filled by the organizers)*
Preferred Topics: CFDMPS/PROPHY
Corresponding author: ÖZKARADUMAN Berker
e-mail of corresponding author: *berker.ozkaraduman@gmail.com*
Type: Oral
Status of corresponding author: Regular

Title

Computational Study for Axisymmetric Nozzle at Transonic and Supersonic Flow Regime

Authors

*Berker ÖZKARADUMAN [1]**

** Corresponding author*
[1] Flight Sciences Department, Turkish Aerospace, 06980 ANKARA, Turkey, berker.ozkaraduman@gmail.com

Abstract

Computational Fluid Dynamics (CFD) analyses of the axisymmetric circular-arc boattail nozzle model[1] from NASA are performed in this study to investigate both the external and internal flow field around the exhaust plume for both the transonic and supersonic regimes. The nozzle geometry corresponding to supersonic cruise is selected as the validation case[1]. The assumptions, inputs, and cases are based on previous studies [2],[3]that used experiment[1] as the validation case. External afterbody and internal nozzle pressure distributions are computed and compared with experimental data. Moreover,Mach number contours are generated to interpret the flow field. An unstructured grid is generated via ANSYS Workbench for the analyzes. The steady-state analyzes are performed at freestream Mach numbers of 0.9 and 1.2. At a freestream Mach number of 0.9, nozzle pressure ratios (NPR) of 4 and 6 are used. However, for a freestream Mach number of 1.2, an NPR of 4 is used, as in a previous study of Dalbello *et al.*[2]. Three different grids are generated for 2D, 3D quarter and 3D full geometries. NPR of 4 at freestream Mach number of 0.9 is run for the 3D full geometry grid merely. ANSYS Fluent and SU2 CFD solvers are utilized to perform analyses. For all 2D and 3D quarter grids, the Roe for flow convection scheme is used with two different turbulence models, namely Spalart Allmaras (SA) and Shear Stress Transport (SST). The results of ANSYS Fluent for a Mach number of 1.2 in the freestream show an oscillatory behavior in the pressure distribution of the external afterbody. Then, different grids are generated to overcome this oscillation and the results are examined. Next, a grid is generated for a 3D quarter geometry that fits the results of the experiment with minor differences. For the 3D full geometry, Roe and JST flow schemes in SU2 are used with the SA and SST turbulence models. In addition, the pressure-based coupled solver of ANSYS Fluent is used instead of the density-based solver of ANSYS Fluent. The Roe convective flow scheme with the SST turbulence model in SU2 yields the best overall agreement with the experimental data.

References

[1] G. T. Carson Jr. and E. E. Lee Jr., "Experimental and Analytical Investigation of Axisymmetric Supersonic Cruise Nozzle Geometry at Mach Numbers From 0.6 to 1.3," NASA, Hampton, Virginia, 1981.

[2] T. DalBello, N. J. Georgiadis, D. A. Yoder and T. G. Keith, "Computational Study of Axisymmetric Off-Design Nozzle Flows," in 42nd AIAA Aerospace Sciences Meeting and Exhibit, Reno, 2004.

[3] J. R. Carlson, "Computational Prediction of Isolated Performance of an Axisymmetric Nozzle at Mach Number 0.90," National Aeronautics and Space Administration, Hampton, Virginia, 1994.