NUMERICAL SIMULATION AND DRAG PREDICTION OF NASA COMMON RESEARCH MODELS

RAMESH K. AGARWAL AND PAN DU

Department of Mechanical Engineering and Materials Science
Washington University in St. Louis, St. Louis, MO 63131, USA
E-mail: rka@wustl.edu, web page: http://www.mems.wustl.edu

Key words: Aerodynamic Drag, Wing-Body, Wing-Body-Tail, Wing-Body-Nacelle-Pylon

Summary. This paper describes the results of 3D turbulent flow simulations to predict the drag of Wing-Body-Tail (WBT) and Wing-Body-Nacelle-Pylon (WBNP) aircraft configurations from NASA Common Research Models. These configurations were part of the 4th and 6th AIAA Drag Prediction Workshops in which CFD modelers participated worldwide. The computations are performed using ANSYS FLUENT. The compressible Reynolds-Averaged Navier-Stokes (RANS) equations are solved using two turbulence models – the Spalart-Allmaras (SA) and SST k-\omega. Lift and drag coefficient curves are obtained at different angles of attack at a constant Mach number. Pressure distributions and flow separation analysis are presented at different angles of attack. Comparisons of computational results for WBT and WBNP models with the experimental data show good agreement. The WBNP results are compared with the WB results for the drag increment study.

1 INTRODUCTION

A great deal of effort has been devoted over past several decades to obtain accurate numerical solutions for flow past a transonic commercial aircrafts and other aerospace industry relevant configurations using the tools of Computational Fluid Dynamics (CFD). The accuracy of drag prediction has been the most challenging among all the aerodynamic coefficients. Since 2001, a series of drag prediction workshops (DPWs) have been organized by the AIAA Applied Aerodynamics Technical Committee whose main purpose has been to assess the state-of-the art computational technology as a tool for drag, lift and moment predictions of aircrafts. NASA common research model is used by participants worldwide to compare results for aerodynamic coefficients from various codes using different algorithms and turbulence model on typical meshes, and the experimental data to assess and determine the best practices for CFD simulations. In this paper, the results for WBT and WBNP configurations are presented using the SA and SST k-\omega models and are compared with the results reported in the literature [1- 4] and the experimental data. Present computations are in good agreement with the experiment and computations of other participants in the workshops.

2 RESULTS and DISCUSSION
Figure 1(a) shows the pressure distribution on WBT at M = 0.85, Re = 5x10^6 and \alpha = 2.38° using the SST k-\omega model. Figure 1(b) and 1(c) show the variation in computed Cl and Cd with \alpha and its comparison with experiment and computations of other investigators using SA and SST k-\omega model. Figure 2(a) shows the pressure distribution on WBNP at M = 0.85, Re =
5x10^6 and \( \alpha = 2.75^\circ \) using the SST k-\( \omega \) model. Figure 2(b) and 2(c) show the variation in computed Cl and Cd with \( \alpha \) and its comparison with experiment and computations using FUN3D [1] using SST k-\( \omega \) model.

![Figure 1](image1.png)

Figure 1: (a) Pressure distribution on WBT at \( M = 0.85, \text{Re} = 5x10^6 \) and \( \alpha = 2.38^\circ \), (b) Cl vs. \( \alpha \) and (c) Cd vs. \( \alpha \).

![Figure 2](image2.png)

Figure 2: (a) Pressure distribution on WBNP at \( M = 0.85, \text{Re} = 5x10^6 \) and \( \alpha = 2.75^\circ \), (b) Cl vs. \( \alpha \) and (c) Cd vs. \( \alpha \).

3 CONCLUSIONS

Computations of 3D turbulent flow past two NASA Common Research Models: Wing-Body-Tail (WBT) and Wing-Body-Nacelle-Pylon (WBNP) aircraft configurations show good agreement with experimental data and computations from participants from AIAA 4th and 6th drag prediction workshops DPW4 and DPW6.

REFERENCES