8th. World Congress on Computational Mechanics (WCCM8)
5th. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 – July 5, 2008
Venice, Italy

ANALYSIS OF WING AND TAIL INTERFERENCE

Zhong Lei[1], Dong-Youn Kwak[2] and Mitsuhiro Murayama[3]

[1] Japan Aerospace Exploration Agency
Aviation Program Group
Osawa 6-13-1, Mitaka, Tokyo, Japan 181-0015
lei.zhong@chofu.jaxa.jp

[2] Japan Aerospace Exploration Agency
Aviation Program Group
Osawa 6-13-1, Mitaka, Tokyo, Japan 181-0015
kwak.dongyoun@chofu.jaxa.jp

[3] Japan Aerospace Exploration Agency
Aviation Program Group
Osawa 6-13-1, Mitaka, Tokyo, Japan 181-0015
murayama.mitsuhiro@chofu.jaxa.jp

Key Words: *CFD, Aerodynamics, Wing, Tail, Interference, Vortex Flow.*

ABSTRACT

Flow field of an experimental airplane [1] is simulated to investigate the wing and tail interference at low speed. For this configuration, the wing and the horizontal tail strongly interact with each other, and the effect of interference on the aerodynamics performance becomes very important, especially at take-off and landing conditions. In this study, influence of the horizontal tail on the wing and the wing on the horizontal tail is investigated by wind tunnel experiment and numerical simulation.

Computational fluid dynamics (CFD) plays an important role in the design process of an aircraft. CFD can provide detail analysis of the flow field and increase the knowledge to understand the flow physics. To simulate the flow field, TAS (Tohoku university Aerodynamic Simulation) code [2], which includes an unstructured mesh generator and a flow solver, is used in this study. Reynolds-averaged Navier-Stokes (RANS) equations are solved on the unstructured mesh by a cell-vertex finite volume method. HLLEW method is used for the numerical flux computations. Second-order spatial accuracy is realized by a linear reconstruction of the primitive variables. LU-SGS implicit method is used for time integration. Fully turbulent flow is assumed in the computation, and the Spalart-Allmaras model is applied to simulate the turbulence. A hybrid unstructured mesh is shown in Fig. 1. To resolve the boundary layer flow, prism mesh is applied near the surface. The spacing of the first grid to the surface is set $9\times10^{-6}m = 0.02\sqrt{\mathrm{Re}}$, and 30 points are distributed by a stretching factor 1.25. The tetra mesh is generated in the outer region. The Mach number is 0.2.

Here are some results of the influence of operated horizontal tail on the wing. Predicted lift coefficients are given in Fig. 2 for all components of the airplane. A nonlinear change due to the deflection of the horizontal tail is presented. The lift is decreased as the horizontal is deflected down. In Fig. 3, the total pressure distributions at the station cross the horizontal tail are compared. It shows that the vortex generated on the wing has an effect of upwash on the flow around the horizontal tail.

REFERENCES

[1] A. Murakami, “Silent Supersonic Technology Demonstration Program”, *the 25th Congress of the International Council of the Aeronautical Sciences*, ICAS 2006-1.4.2, 2006.

[2] K. Nakahashi, F. Togashi, T. Fujita, and Y. Ito, “Numerical Simulations on Separation of Scaled Supersonic Experimental Airplane from Rocket Booster at Supersonic Speed,” AIAA Paper 2002-2843, 2002.

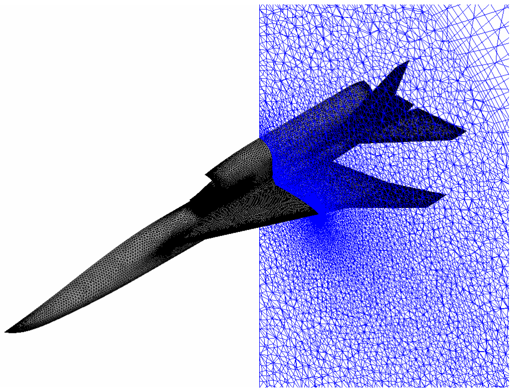
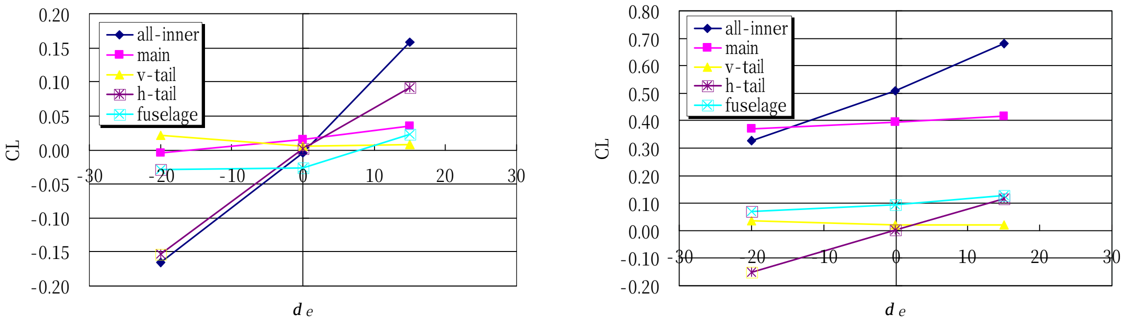
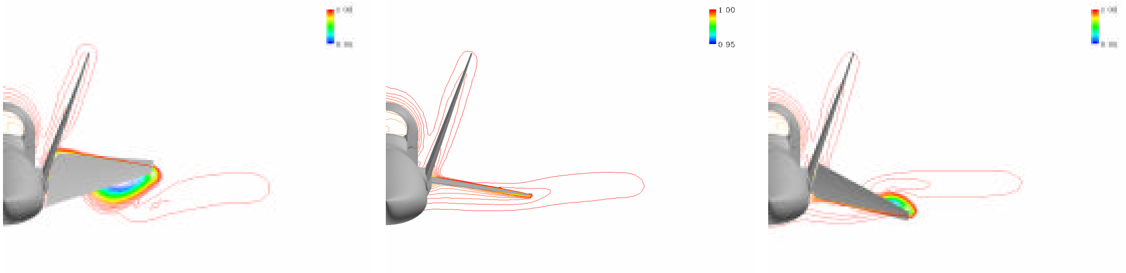


Fig. 1 Unstructured mesh used in simulation.

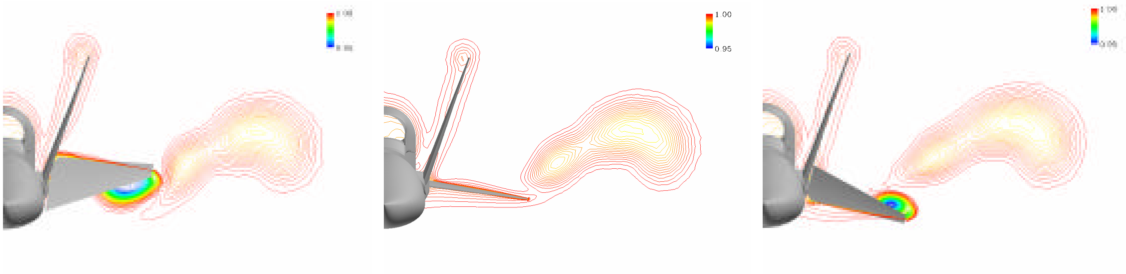


(a) a=0° (b) a=10°

Fig. 2 Components of aerodynamic lift coefficient



(a) a=0°, d_e=-20° (b) a=0°, d_e=0° (c) a=0°, d_e=15°



(d) a=10°, d_e=-20° (e) a=10°, d_e=0° (f) a=10°, d_e=15°

Fig. 3 Total pressure distribution at x=13.3m