Available online at www.sciencedirect.com
ScienceDirect
Procedia Engineering 99 (2015) 368 – 376
Procedia Engineering
www.elsevier.com/locate/procedia

“APISAT2014”, 2014 Asia-Pacific International Symposium on Aerospace Technology, APISAT2014

The Numerical Simulation of Civil Transportation High-lift Configuration

Hao Xuan*, Su Cheng, Hu Ning, Zhang weimin

China Academy of Aerospace Aerodynamics, 17 Yungang West Road, Beijing, 100074, P. R. China

Abstract

The complex multiform flow phenomenon including boundary layer transition, lamination separation, turbulence attach line and re-lamination, boundary shock interaction and so on will appear when the air flow over the high lift configuration, which makes it very difficult to predict by numerical simulation. In order to validate the CFD code prediction capability of high lift configuration of swept wing with high aspect ratio, AIAA held two workshops of CFD high lift prediction in 2010 and 2012. The NASA Trap wing model and DLR-F11 model were chosen to research the mesh resolution, turbulence model, transition prediction and Reynolds effect by numerical simulation. Based on the published work of the1st and 2nd high lift prediction workshop, some numerical simulation research to validate the rationality of mesh generation are done in this paper. The two configurations of NASA Trap wing and DLR F-11 are simulated by solving the Reynolds-averaged Navior-Stokes equations with the S-A turbulence model. The numerical data in linear region agree very well with the experiments but the discrepancy between numerical data and experiment data near the stall region is perceptible. It can be conclude generally from the numerical results of two configurations that in the engineer design phase the Reynolds-averaged Navior-Stokes equations can give correct results of the aerodynamic characteristic with carefully generated mesh and appropriate turbulence model.

Peer-review under responsibility of Chinese Society of Aeronautics and Astronautics (CSAA)

Keywords: High lift Device; Numerical Simulation; Trapwing; DLR F-11

* Corresponding author. Tel.: +86-13810025289
E-mail address: haoxuan0722@sohu.com

Peer-review under responsibility of Chinese Society of Aeronautics and Astronautics (CSAA)
doi:10.1016/j.proeng.2014.12.549

1. Introduction

The high lift device is one of the most important parts during the civil transportation taking off and landing and has a direct relationship to the reliability, economy and environment protecting of the aircraft. The configuration and flow field of the high lift device is very complicated with the flow phenomenon including boundary layer transition, lamination separation, turbulence attach line and re-lamination, boundary shock interaction and so on when the air flow over the high lift configuration, which makes it very difficult to predict by numerical simulation. The maximum lift coefficient and stall angle of the transportation aircraft during the civil transportation taking off and landing are the two key parameters of high lift device design. Limited by the turbulence model, simulation method and so on, it is difficult to get these parameters accurately by CFD (Computational Fluid Dynamics) simulation. Although as the rapid development of computer, the application of CFD during the aircraft aerodynamic design are more and more popular, the computation of high lift flow is still a big challenge.

In order to develop the proper numerical method, turbulence model and mesh generation standard of high lift configuration of swept wing with high aspect ratio and comprehend the physics of high lift flow expressly, AIAA held two workshops of CFD high lift prediction in 2010 and 2012. The NASA Trap wing model and DLR-F11 model were chosen to research the mesh resolution, turbulence model, Reynolds effect, transition prediction and geometry details by numerical simulation. Based on the published work of the1st and 2nd high lift prediction workshop[1]-[5], some numerical simulation research of Trapwing and DLR-F11 model are done in this paper, with the emphases to validate the rationality of mesh generation.

2. Numerical Methods

The steady Reynolds averaged Navior-Stokes equation is chosen in this paper to solve the high lift flow field full turbulently with SA turbulence model. In inertial Cartesian coordinates, the integration form of the steady N-S equation can be written as:

$$\frac{\partial}{\partial t}\iiint_{\Omega}\overline{Q}d\Omega+\iint_{S}\left(\overline{G}-\overline{Q}\vec{q}_b\right)\cdot d\vec{S}=\frac{1}{\mathrm{Re}}\iint_{S}\overline{F}^V\cdot d\vec{S} \tag{1}$$

$$\overline{Q}=\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ \rho e_t\end{bmatrix},\quad \overline{G}=\begin{bmatrix}\rho u & \rho v & \rho w\\ \rho u^2+p & \rho vu & \rho wu\\ \rho uv & \rho v^2+p & \rho wv\\ \rho uw & \rho vw & \rho w^2+p\\ \rho uh_t & \rho vh_t & \rho wh_t\end{bmatrix},\quad \overline{F}^V=\begin{bmatrix}0 & 0 & 0\\ \tau_{xx} & \tau_{yx} & \tau_{zx}\\ \tau_{xy} & \tau_{yy} & \tau_{zy}\\ \tau_{xz} & \tau_{yz} & \tau_{zz}\\ \varphi_x & \varphi_y & \varphi_z\end{bmatrix}, \tag{2}$$

The $\overline{G}$, $\overline{\mathrm{F}}^{\mathrm{V}}$ is convention term and dispersion term separately, where Ω is the control volume. S is the control surface, and $d\vec{S}$ is the out normal surface vector of the S.

3. Numerical Results and Analysis

3.1. NASA Trapwing model

3.1.1Model

The Trapwing model in the wind tunnel is showed in Fig. 1. The config1 in test cases with full spanwise flap deployed at 25 degree and slat deployed at 30 degree is simulated. The main geometry parameters are listed in Tab. 1.