Italian Association of Aeronautics and Astronautics
XXII Conference
Napoli, 9-12 September 2013

NUMERICAL AERODYNAMIC ANALYSIS ON A TRAPEZOIDAL WING WITH HIGH LIFT DEVICES: A COMPARISON WITH EXPERIMENTAL DATA

P. Della Vecchia[1*], D. Ciliberti[1]

[1]Dept. of Industrial Engineering, University of Naples "Federico II", 21 Via Claudio 80125 Naples (Italy)

*pierluigi.dellavecchia@unina.it

ABSTRACT

The aerodynamic analysis on the DLR-F11 high lift configuration model has been performed on the supercomputing grid infrastructure SCoPE of the University of Naples "Federico II". The model geometry is representative of a wide-body commercial aircraft, which experimental investigations at high Reynolds number have been performed at the European Transonic Wind-tunnel (ETW) for the 2nd AIAA High Lift Prediction Workshop. The commercial CAE package Star-CCM+ has been used to solve the Reynolds-averaged Navier-Stokes equations. Inviscid, viscous incompressible, and compressible analyses have been performed with mesh refinement. The inviscid calculations have been used to assess how far is the eulerian prediction from experimental data. Viscous and compressible calculations have been realized using the Spalart-Allmaras turbulence model at 0.175 Mach number and 15.1 million Reynolds number. Results show that the simple Spalart-Allmaras turbulence model can predict quite accurately the stall and post-stall behaviour, getting the angle of stall and underestimating the maximum lift coefficient by less than 5%. Comparisons among numerical and experimental pressure coefficients at several sections are also shown. Finally, the stall path is described.

Keywords: aircraft, aerodynamics, CFD, computing grid

1 INTRODUCTION

Nowadays the numerical aerodynamic analysis of high lift configuration of commercial aircraft is a crucial item to reduce the number of wind tunnel tests and give a well-suited instrument for the industrial design of the high lift systems. The numerical simulation of these configurations is very complex, due to difficulties to simulate separations phenomena, unsteadiness, confluent boundary layers, transition, and so on.

The successful first AIAA High Lift Prediction workshop (HiLiftPW-1) [1] proved the request and demand for a sustained international CFD validation exercise for 3-dimensional high lift configurations. For HiLiftPW-1, the NASA trapezoidal wing was selected as a reference configuration, with a three-element high lift wing configuration with a body pod. A low aspect ratio wing ($AR = 4.56$), untwisted, and with no dihedral angle was used [1].

The considerations to identify a reference configuration for the 2nd AIAA High Lift Prediction Workshop were driven by the request to take a more realistic high lift