CFD ANALYSIS FOR REAL 2D FLOWS

Paolo Caccavale*, Carlo de Nicola, Benedetto Mele*, Antonio Piccolo*****
PhD Engineer, University of Naples Federico II,
****Associate Professor, University of Naples Federico II,***
*****Senior Engineer, ALENIA Technical and Design Dept. Loads Unit, Pomigliano d'Arco - Naples**

Keywords: *Airfoil flow, Bavette, High lift, Turbulence modeling, Unsteady flows, Buffet, Laminar bubble*

ABSTRACT

Validation and applications of the (U)RANS code developed at the Aeronautical Design and Project Department of University of Naples Federico II (DPA) are presented. While validation results for flows with concentrated and massive separation regions are quite satisfactory, the application to the very complex flow related to real high lift configuration may lead to some local descrepancies with acceptable global results.

1 INTRODUCTION

At present, CFD tools based on the full Navier-Stokes equations solution have reached a high level of sophistication and are widely and world-wide used to deal with a large number of fluidynamic problems. The capability and diffusion of such methods has been greatly increased by the availability on the market of more and more powerful computers at lower and lower costs. Till a few years ago the main CFD limits have been the capability of an acceptable accuracy in the prediction of attached flow cases, and historically, the first CFD applications (potential panel methods) have been concerned the airfoils analysis at low angles of attack. The main difficulties in the development of methods based on the Navier-Stokes equations solution have been concerned three crucial points: the "operative" optimisation in the grid-generation (i.e the improvement in the user-friendly modality of employ and the capability in the treatment of any complex geometry), the formulation of more and more suitable turbulence models and their " tuning" for any specific application, and the upgrade of the numerical techniques to accelerate the convergence in the solution of the equation system. These are still the crucial points for the future CFD development, as unchanged is the goal of all CFD developers: to improve the numerical methods capability to the description of any "real" flow case, where flow may be separated in any point on the body surfaces.

In the aeronautical industrial design study the availability of such sophisticated tools may represent a meaningful save of times and costs both in the development of a new product and in the routine upgrade of an existing product or, simply, to get fast any data one needs. In general in the aeronautical research and design activity the CFD has been always seen as an alternative to the more expensive wind-tunnel investigations.

For multi-element airfoils the flow description in the flap-well has always represented a hard obstacle, usually bypassed through the definition of a proper fairing (often coincident with a so called "dividing