Computers & Fluids 109 (2015) 67–71

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

Technical note

On the implementation of a turbulence model for low Reynolds number flows

Pietro Catalano [a,1], Benedetto Mele [b,2], Renato Tognaccini [b,*,2]

[a] Centro Italiano Ricerche Aerospaziali (CIRA), Capua (CE) 81043, Italy
[b] Università di Napoli Federico II, Piazzale V. Tecchio 80, 80125 Napoli, Italy

ARTICLE INFO

Article history:
Received 24 March 2014
Received in revised form 10 November 2014
Accepted 5 December 2014
Available online 24 December 2014

Keywords:
Turbulence model
Low Reynolds number flow
Laminar separation
Transition

ABSTRACT

Some aspects in the implementation of a turbulence model revealed critical for the correct prediction of the flow characteristics in the case of a low Reynolds number flow (10^4–10^5) around an airfoil. In particular the treatment of the transition point setting strongly influenced the computed solution. This effect was not found in case of high Reynolds number.

The aim of the present short note is to report an experience describing how changes in an apparent detail of the implementation of a turbulence model provided unexpected large differences on the results. A discussion on this aspect has not been found in the literature, thus it has been considered worth to present it here.

Recently, a modification of the k–ω SST turbulence model (identified here as k–ω SST-LR) has been proposed to improve, by standard RANS CFD analyses, the simulation of the flow around airfoils in the low Reynolds number regime (10^4–10^5) [1]. A detailed comparison with LES results showed very good agreement up to and beyond stall [2], provided a correct setting of the transition point inside the laminar separation bubble (LSB). In the same paper an engineering criteria for identifying the transition location was also suggested.

The results proposed in [1,2] were obtained by CIRA[3] RANS solver ZEN described in [3]. The solver is block structured with standard central space discretization and explicit adaptive 2nd and 4th order artificial dissipation. Pseudo time-marching is performed by a Runge–Kutta multistage scheme.

In Fig. 1 the skin friction and pressure coefficient distributions obtained for the SD 7003 airfoil at freestream Mach number $M_\infty = 0.1$, Reynolds number $Re_\infty = 6 \cdot 10^4$ and angle of attack $\alpha = 4^\circ$ are proposed. The adopted input parameters are summarized in Table 1.

The transition position on the upper surface along the chord of length c was specified at $x_{tr}/c = 0.53$ as indicated by the experiments [4]. In the same figure a grid convergence study is also proposed (in the remaining of the paper all results are proposed for the 768×176 grid). The plots also show a very satisfactory agreement with a reference LES calculation, although in the turbulent reattached part of the flow the skin friction seems still under predicted.

The modified turbulence model was also introduced in the RANS solver FLOWer developed by DLR[4] [5] and widely adopted in the applied research and industrial community. The technical characteristics of this solver are very similar to CIRA ZEN code: block structured with standard central space discretization, explicit adaptive 2nd and 4th order artificial dissipation and pseudo time-marching by Runge–Kutta multistage scheme.

Quite surprisingly and despite of the same adopted test cases and inputs, FLOWer code provided a completely different description of the LSB. Why two flow solver codes, adopting the same numerical technique and turbulence models and using the same grid and input are providing different results?

Previous comparisons in high Reynolds number conditions adopting the standard k–ω SST model showed satisfactory agreement between the two solvers; therefore the numerical

* Corresponding author.
E-mail addresses: p.catalano@cira.it (P. Catalano), benmele@unina.it (B. Mele), rtogna@unina.it (R. Tognaccini).
[1] Fluid Dynamics Department, Italy.
[2] Dipartimento di Ingegneria Industriale, Italy.
[3] Italian Aerospace Research Center.
[4] German Aerospace Center.

http://dx.doi.org/10.1016/j.compfluid.2014.12.009
0045-7930/