文章编号:1000-8055(2014)02-0475-06 doi:10.13224/j.cnki.jasp.2014.02.029

Numerical simulation of airfoil flows with a turbulence model

GAO Lin, JIANG Li-jun, CUI Shu-xin, GAO Ge

(National Key Laboratory of Science and Technology on Aero-Engine Aero-thermodynamics,
School of Energy and Power Engineering,
Beijing University of Aeronautics and Astronautics, Beijing 100191, China)

Abstract: Turbulent flows over AS240 and NACA4412 airfoil were simulated numerically using a two-equation turbulence model named k-ξ model. The predictions of velocity profiles and the pressure coefficient of airfoil AS240 at 8°/19° attack angle and NACA4412 at 13.87° attack angle were calculated. The results were compared with those using k-ε and k-ω models, as well as experimental data. It indicates that the new k-ξ model offers more realistic prediction than the other two models. The main finding shows that the new k-ξ model is good at predicting separated flows around airfoils, and it captures the flow feature of pressure-induced separation adequately. All calculations are implemented as per openFOAM 1.7.1 (open source field operation and manipulation).

Key words: turbulence; airfoil; incompressible; separation; openFOAM

CLC number: V231 **Document code**: A

From the 1960s onwards the aerospace industry has integrated CFD techniques into the design, research and development (R&D) and manufacture of aircraft and jet engines[1]. The turbulent flow field over an airfoil at high incidence contains complex flow phenomenon, such as compressible effect, pressure gradient, unsteady effect, separation, etc, the prediction of which presents great challenges to Reynolds-averaged Navier-Stokes (RANS)[2]. As turbulent separated effects play a significant role in these flows, an adequate representation is crucial for successful aerodynamic performance prediction[3]. Nowadays eddy viscosity turbulence models are widely used due to their practicality. Simple flows can be analyzed using data correlations or algebraic eddy viscosities; but in more complicated flows, such as a massively separated boundary layer, a more elaborate level of modeling is require. It is widely believed that at least a two-equation transport model is required in such cases[4]. Dozens of two-equation turbulence models, such as k-ε, k-τ model, has been established and widely used since the first two-equation k-ω model proposed by Kolmogorov in 1942[5]. Over the last decades, scholars had made a great effort on turbulence modeling with both simplicity and accuracy.

The present work is aimed to test and verify a new two-equation turbulent model k-ξ and discuss the complex turbulent flow past an airfoil AS240 and NACA4412 airfoil at fixed incidence. The turbulent flow over AS240 at 8° attack angle and NACA4412 airfoil at 13.87° have gone with separation and other complex phenomenon, which is prefer for testing and verifying turbulent models. AS240 airfoil had been designed and manufactured by AIRBUS, which provide a detailed and well-established experimental database of AS240 for CFD validation purpose. And the low-speed flow around NACA4412 airfoil at maximum lift attack angle is set as standard

Received: 2013-03-21

E-mail: glinbuaa@sjp.buaa.edu.cn

benchmark by Stanford International Turbulence Conference[6].

1 Governing equations

There are two transport equations solved for two turbulent quantities in two-equation models, which are the turbulent kinetic energy k and the other variable to derive a turbulent length scale. Thus the eddy viscosity can be expressed as

$$\nu_t = C_\mu \cdot k^m \cdot \Psi^n \tag{1}$$

Where C_μ, m, n are constants, Ψ is considered as a generic length-scale variable. Based on numerical analysis, the second length-scale variable in turbulence modeling is very important, especially in prediction of separated flows. The larger the value of the sum of m and n, the larger the eddy viscosity since most turbulence variables are proportional to rate of strain in strong shear boundary layer regions. This will affect the ability of models to predict separated flows[7].

Table 1 presents the comparison of the sum of m and n of commonly used two-equation models.

Table 1 m and n value of different models

Turbulence model	Ψ	m	n	$\Sigma = m+n$
k-ε (Launder and Sharma)	ε	2	−1	1
k-ω (Wilcox)	ω	1	−1	0
k-ζ (Jiang)	ξ	1	−2	−1

In this paper, a eddy viscosity turbulence model, k-ξ model, has been adopted. It is directly derived from standard k-ε and k-ω models based on a new standpoint.

The k-ξ model base on the standard model and a new variable ξ is introduced, which is the square root of specific dissipation $\xi=\sqrt{\omega}=\sqrt{\varepsilon}/\sqrt{k}$. Space lacks for a detailed description of the detailed deducing process.

The transport equations of k and ξ are as followed:

$$\frac{\partial \xi}{\partial t} + U_j \frac{\partial \xi}{\partial x_j} = \alpha \frac{\xi}{k} P_k - \beta \xi^3 + \frac{\partial}{\partial x_j}\left[(\nu + \sigma_\xi \nu_t)\frac{\partial \xi}{\partial x_j}\right] + \sigma_c \frac{1}{\xi^2}\frac{\partial k}{\partial x_j}\frac{\partial \xi}{\partial x_j} \tag{2}$$

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - k\xi^2 + \frac{\partial}{\partial x_j}\left[(\nu + \sigma_k \nu_t)\frac{\partial k}{\partial x_j}\right] \tag{3}$$

Where the production rate P_k is $P_k = \nu_t S_{ij}^2$, with the strain rate S_{ij} is $S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial u_i}\right)$.

The eddy-viscosity is defined as

$$\nu_t = C_\mu \frac{k}{\xi^2} \tag{4}$$

The constants α, β, σ_k, σ_ξ for turbulence model used in this paper are given in table 2.

Table 2 Constants for k-ξ turbulence model

α	β	σ_k	σ_ξ	C_μ	σ_c (to ensure to be non-negative)
0.26	0.4	1.0	1.0	0.09	$0, \text{if } \frac{\partial k}{\partial x_j}\frac{\partial \xi}{\partial x_j} \leqslant 0$ $1.5, \text{if } \frac{\partial k}{\partial x_j}\frac{\partial \xi}{\partial x_j} > 0$

2 Computational methods

In this paper numerical results obtained with k-ξ model had compared with experimental data to test and verify the new eddy viscosity turbulent model.

For both cases, the inlet condition of k and ξ can be set as followed:

$$k = \frac{3}{2}(U \cdot i)^2 \tag{5}$$

And

$$\xi = \sqrt{C_\mu k / \nu_t} \tag{6}$$

The eddy viscosity is nearly ten times of fluid viscosity $\nu_t = 10\nu$, and outlet variables can be set as zero gradients. No-slip wall conditions are used on the airfoils surface. If the mesh is coarse ($y^+ \geqslant 30$), standard wall-functions for the turbulent variables are adopted. When the mesh is fine enough ($y^+ \approx 1$), turbulent kinetic energy is set to zero and $\xi = \sqrt{6\nu/0.075 y^2}$ are directly derived from the wall boundary condition for ω with a fine mesh. An under-relaxation method is used for stability and faster convergence. The pressure under-relaxation factor is 0.3 and a value of 0.7 is used for the other equations.

In this paper, all calculation and simulation are implemented on openFOAM 1.7.1. open-

FOAM is an open source library written in C++. It is a well-structured code, mostly used to implement CFD solvers, although it is also used in other applications. OpenFOAM is based on the finite volume method, but there are also implementations of the finite area and finite element methods. With regards to basic features, such as turbulence models and discretization schemes, OpenFOAM is a serious and high quality CFD tool that is constantly evolving[8-11]. The solution is affected by an iterative pressure-correction semi-implicit method for pressure-linked equations (SIMPLE) algorithm for incompressible flow. The advective volume-face fluxes are approximated using second-order total variation deminishing (TVD)-limited linear differencing. Preconditioned conjugate/bi-conjugate gradient matrix solver methods are used to solve the discretised matrix equations.

3　Numerical results and discussions

Based on OpenFOAM, The turbulent flow over AS240 airfoil at 8°/19°attack angle and NACA4421 airfoil at 13.87°attack angle have been simulated numerically, which are standard benchmarks with series of reliable experimental data.

3.1　AS240 airfoil at 8° attack angle

The AS-240 airfoil has a 16% thickness involving a very progressive stall and a velocity overshoot close to the leading edge low enough to prevent immediate transition. The chord of AS240 airfoil is 0.6 m and the magnitude inlet velocity is 50 m/s, therefore the mean Reynolds number based on the chord length and inlet velocity is 2.1×10^6. The flow can be treated as incompressible flow approximately as the Mach number $Ma_\infty\approx0.15$ in the Gleyzes' experiment. The turbulent intensity is about 0.1%. Computation domain is decuple of the chord length and a C-type grid of 210×88 size is adopted.

Fig. 1 presents the pressure coefficient C_p of AS240 at 8° attack angle, of which numerical result using k-ξ model compared with experimental data and results of k-ε model and k-ω model. As seen there is a severe decline of pressure coefficient line at around 40% chord. That is because the laminar-turbulent transition occurred in a mid-chord laminar separation bubble. There is no obvious difference between the three models in pressure coefficient.

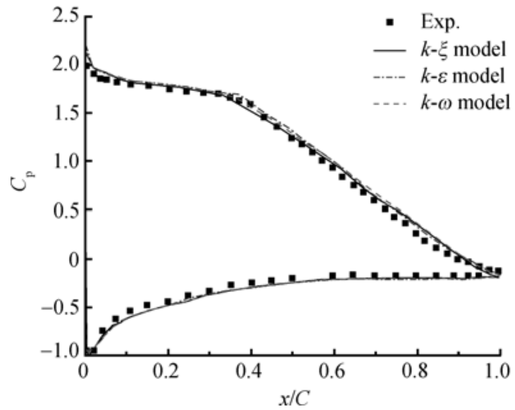

Fig. 1　Pressure coefficient of AS240 at 8° attack angle

The transitional separation bubble occurs in a weaker adverse pressure gradient, compared to a leading edge one and its location is consequently more difficult to compute accurately[12]. The streamwise velocity profiles of AS240 airfoil at 8° attack angle are presented in Fig. 2. At the location of $x/C=$ 0.825, 0.865, 0.910, 0.950, the streamwise velocity profiles (U/U_{ref}) were obtained by the k-ξ model, compare with results of experiment, k-ε model and k-ω model. As seen, the numerical results agree better with experimental data than the other two. According to Fig. 2, computing results of the boundary layer thickness with the k-ξ model are very close to experimental data, and the variability of the velocity profile from being full to being deficit under the adverse pressure gradient has been revealed basically.

3.2　AS240 airfoil at 19° attack angle

The flow over AS240 airfoil at 19°attack angle is fully turbulent with obvious separation. As a result of the transition occurred at the leading edge of the airfoil, a huge separation bubble formed at the trailing edge under the adverse pressure gradient[13]. Wall-pressure coefficient obtained with the k-ξ model is compared to

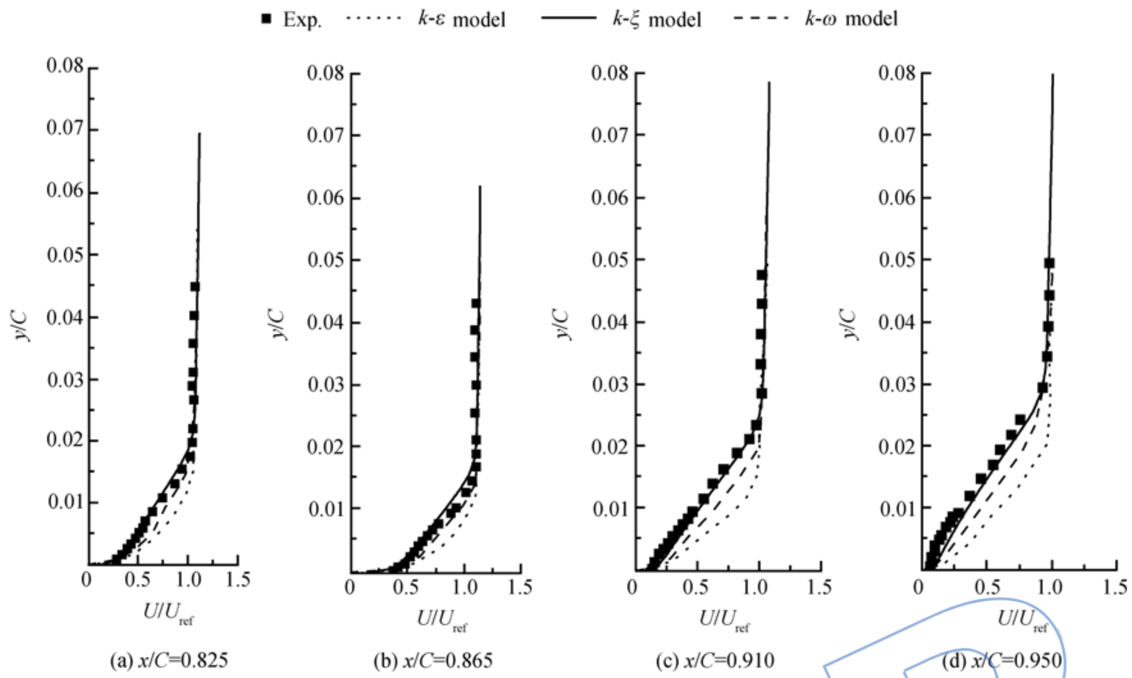

Fig. 2 Streamwise velocity profile at the location of AS240 at 8° attack angle

experimental data in Fig. 3. The k-ξ model captures the flow feature of that pressure-induced separation adequately and the numerical result agrees well with the experimental data.

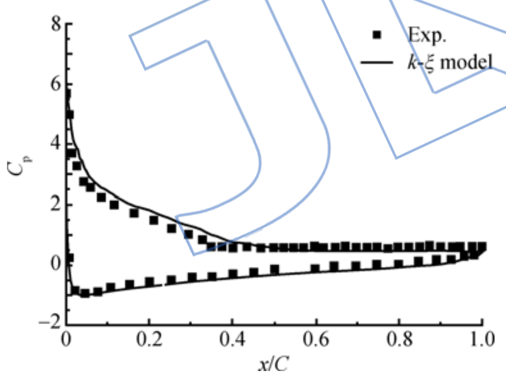

Fig. 3 Pressure coefficient of AS240 at 19° attack angle

3.3 NACA4412 airfoil at 13.87° attack angle

The experimentally obtained stable characteristic of mean flow over NACA4412 at 13.87° attack angle may provide excellent data for turbulent research[14]. The Reynolds number with respect to chord length (C=90.12 cm) and free stream velocity (U_∞=29.73 m/s) is Re=1.52×10^6[15]. The geometry and associated flow are resolved by a 465×89 C-type mesh, with first points away from the wall at the location corresponding to y^+ around 30. The far field boundary is located at 10 chords. The turbulent intensity in the free stream is less than 0.1%. The inlet and outlet conditions can be set as the part 3 statement.

Comparisons of computed wall pressure coefficients C_p with experimental data are presented in Fig. 4. The result of calculation with the k-ξ model is in accordance with the experimental data.

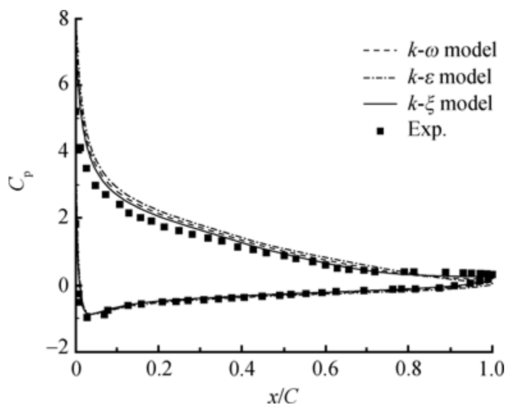

Fig. 4 Pressure coefficient of NACA4412 airfoil at 13.87° attack angle

The streamwise velocity profiles at different location of the upper surface of NACA4412 airfoil at 13.87° attack angle are presented in Fig. 5. The numerical results agree well with the experimental data. The k-ξ model achieves a successful prediction of tail separation. Fig. 6

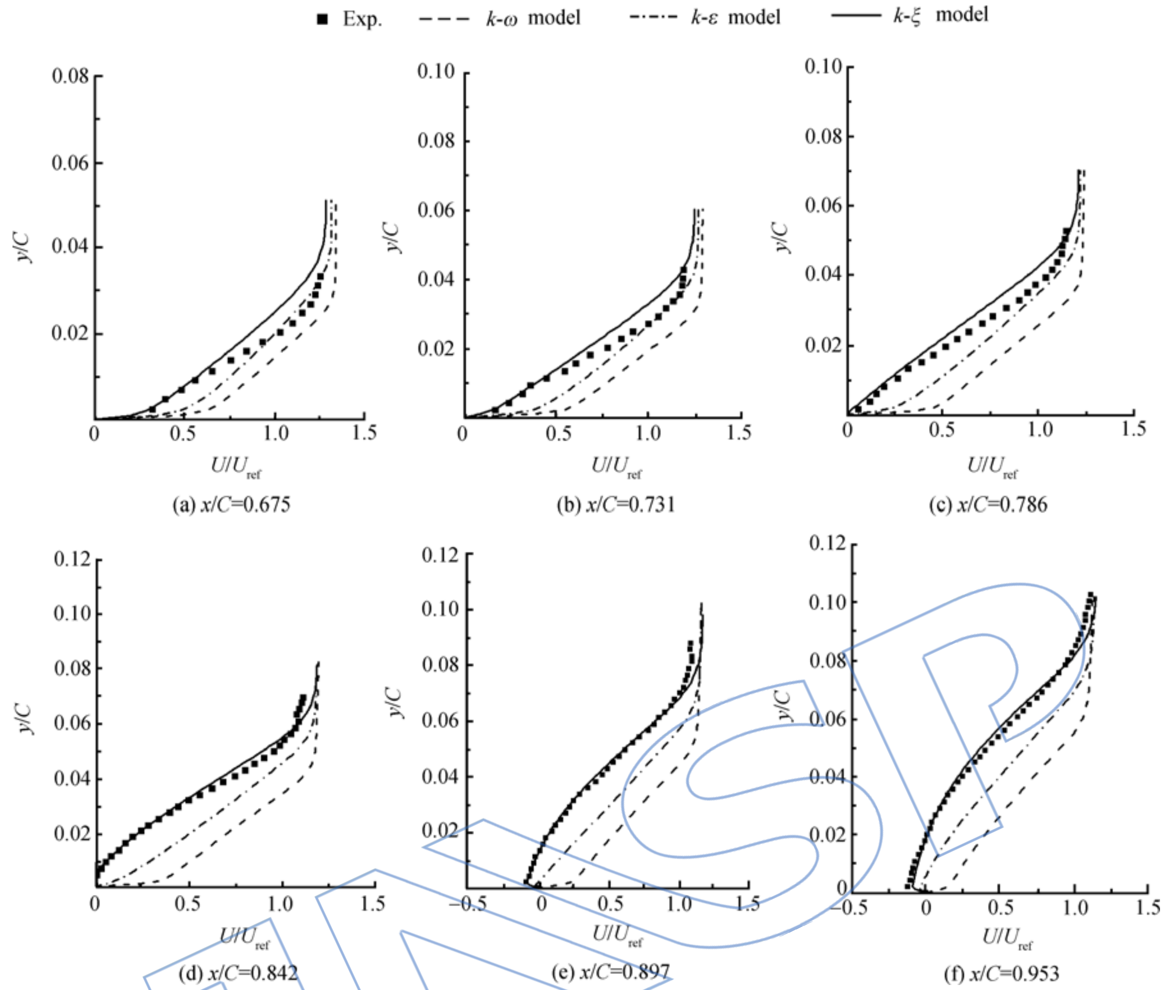

Fig. 5　Streamwise velocity profile at the location of NACA4412 airfoil at 13.87° attack angle

(a) Result of k-ε model

(b) Result of k-ω model

(c) Result of k-ξ model

Fig. 6　Mean flow streamlines of NACA4412 airfoil at 13.87° attack angle

presents numerical results of the mean flow streamlines of NACA4412 airfoil at 13.87° attack angle of k-ξ model against k-ε model and k-ω model. As seen, the k-ε model fails to predict complex flow with separation. The k-ξ model gives a more accurate result near the trailing edge.

4 Conclusions

The turbulent flow over airfoils (AS240 airfoil at 8°/19° attack angle and NACA4412 airfoil at 13.87° attack angle) had been simulated numerically in the present work. A new two-equation turbulence model, named k-ξ model which is directly derived from standard k-ε and k-ω model based on a new standpoint, is applied in calculating the complex turbulent flow over airfoil with separation. This new turbulence model keeps the simplicity and elegance of two-equation turbulence models and can be directly integrated through the viscous sub-layer as k-ω model.

k-ξ model offers realistic prediction, and the numerical results agree well with the experimental data. The results show that the k-ξ model is good at predicting separated flows and could capture the flow feature of that pressure-gradient-induced separation adequately.

References:

[1] Fudhail A M, Zin M R M, Azwadi N C S, et al. Accurate numerical prediction of incompressible fluid flow in lid-driven cavities[J]. Applied Mechanics and Materials, 2012, 110/111/112/113/114/115/116: 4365-4372.

[2] GAO Lin, XU Jinglei, GAO Ge. Numerical simulation of turbulent flow past airfoils on openFOAM[J]. Procedia Engineering, 2012, 31: 756-761.

[3] Franke M, Rung T, Thiele F. Advanced turbulence modelling in aerodynamic flow solvers[M]. Berlin: Springer Berlin Heidelberg, 2005: 225-240.

[4] Durbin P A. Separated flow computations with the k-epsilon-v-squared model[J]. AIAA Journal, 1995, 33(4): 659-664.

[5] Kolmogorov A N. The equations of turbulent motion in an incompressible fluid[J]. Izvestiya Academy of Sciences, Physics, 1942, 6(1/2): 56-58.

[6] Coles D, Wadcock A J. Flying-hot-wire study of flow past an NACA 4412 airfoil at maximum lift[J]. AIAA Journal, 1979, 17(4): 321-329.

[7] Jiang L, Tabor G, Gao G. A new turbulence model for separated flows[J]. International Journal of Computational Fluid Dynamics, 2011, 25(8): 427-438.

[8] Jasak H, Jemcov A, Tukovic Z. OpenFOAM: A C++ library for complex physics simulations[R]. Dubrovnik, Croatia: International Workshop on Coupled Methods in Numerical Dynamics, 2007.

[9] Mangani L, Bianchini C, Andreini A, et al. Development and validation of a C++ object oriented CFD code for heat transfer analysis[R]. Vancourver, Canada: ASME-JSME, Thermal Engineering and Summer Heat Transfer Conference, 2007.

[10] Petit O, Page M, Beaudoin M, et al. The ERCOFTAC centrifugal pump openFOAM case-study[C]// 3rd IAHR International Meeting of the Workgroup of Cavitation and Dynamic Problems in Hydraulic Machinery and Systems. Brno, Czech: Czech Republic, 2009: 523-232.

[11] Weller H G, Tabor G, Jasak H, et al. A tensorial approach to computational continuum mechanics using object-oriented techniques[J]. Computers in Physics, 1998, 12(6): 620-631.

[12] Gleyzes C, Capbern P. Experimental study of two AIRBUS/ONERA airfoils in near stall conditions: Part Ⅰ boundary layers[J]. Aerospace Science and Technology, 2003, 7(6): 439-449.

[13] Guilmineau E, Piquet J, Queutey P. Two-dimensional turbulent viscous flow simulation past airfoils at fixed incidence[J]. Computers and Fluids, 1997, 26(2): 135-162.

[14] REN Xin, LI Baihe, YIN Xingyu, et al. Calculation of airfoil flows using GAO-YONG turbulence equations[J]. Journal of Aerospace Power, 2007, 22(1): 73-78. (in Chinese)

[15] Coles D, Wadcock A J. A Flying-hot-wire study of two-dimensional mean flow past an NACA 4412 airfoil at maximum lift[R]. Seattle, US: AIAA IITH Fluid and Plasma Dynamics Conference, 1978.