CALCULATION OF THE A-AIRFOIL USING THE $k-\tau$ MODEL

P.-Å. Lindberg, C. Weber & A. Rizzi

Dept of Aeronautical Engineering
Royal Institute of Technology
S - 100 44 Stockholm, Sweden

Abstract

A two-equation turbulence model, $k-\tau$, has been used for calculation of an 2D airfoil close to stall. The $k-\tau$ model can be derived from the standard $k-\epsilon$ model if the turbulent time scale $\tau = k/\epsilon$ is introduced. In the present calculations damping functions based on results from direct numerical simulations are used. The requirements for the accuracy of the discretization of the $k-\tau$ model has been investigated. With a proper discretization the $k-\tau$ model has shown to be more robust than an equivalent $k-\epsilon$ model implemented in the same code. The results from the $k-\tau$ model agrees well with experimental results and are as good or better than other calculations using the $k-\epsilon$ model on the same test case by other groups. Some results using the Baldwin-Lomax model are also presented.

Introduction

The $k-\epsilon$ model has become a standard turbulence model in a great variety of computational fluid mechanics (CFD). It was originally developed for calculation of high Reynolds number flows far from influence of solid walls. The region close to walls was treated by wall functions were the law-of-the-wall was assumed to be valid. This approach is well suited for calculations were the velocity profiles are close to the fully developed logarithmic profile found in boundary layers etc. However, for more complicated flows, or when an accurate solution in the vicinity of the wall is needed, e.g. for calculation of skin friction or heat transfer, it is necessary to compute also the region close to the wall. In order to be able to this with the $k-\epsilon$ model a number of low-Reynolds number extensions has been suggested, see e.g. Patel, Rodi & Scheuerer [8]. Different modifications has been tested in various extensions to the standard $k-\epsilon$ model where the most important are the definition of a boundary condition for ϵ and introduction of damping functions in the ϵ equation. Also various extra terms which give a contribution close to the wall only has been tested.

A way of coming around the lack of a physically correct boundary condition for ϵ was suggested by Speziale, Abid & Anderson [11] by introducing the turbulent time scale τ , defined as $\tau = k/\epsilon$. If ϵ is substituted in the standard $k-\epsilon$ model one gets a new two-equation model for k and τ which has the advantage of having a physically correct boundary condition at solid walls.

Although it was some years ago the $k-\tau$ model was suggested it has not been used extensively. An explanation for this is probably that the discretization of the terms in the τ equation is critical and with some standard techniques it may give severe numerical problems. However, with a careful discretization the $k-\tau$ model may have some advantage over the $k-\epsilon$ model.

The $k-\tau$ model has been used here for calculation of the 2D airfoil A at low Mach numbers. For this test case accurate measurements and a number of calculated results are present as it was one of the test cases in the BRITE/EURAM project EUROVAL, Haase et al. [3]. In this test case the airfoil A was used for prediction of stall. Experimental data are available for several Reynolds numbers and angle of attacks. From the calculated contributions it was shown that algebraic and two-equations models in general have difficulties to calculate the flow field correctly close to stall. Only with an algebraic Reynolds stress model (ASM) the separation close to the trailing edge may be predicted, Davidson & Rizzi [2]. A probable explanation for this may be that the effects of stream line curvature and the normal Reynolds stresses not can be predicted correctly by the eddy viscosity based models. However, the ASM models or other Reynolds stress algebraic models, e.g Shih, Zhu & Lumley [10] requires a two-equation model as a base. It may anyhow be fruitful to use the $k-\tau$ model for this test case, although a correct prediction of the stall conditions may not be anticipated. As this may serve as a first step in the development of a such a model based on the $k-\tau$ model.

Equations

The compressible Reynolds averaged Navier-Stokes equations, which not presented here, see e.g. Hirsch [4] for a reference, are solved together with the $k-\tau$ model. Some results will also be presented using the Baldwin-Lomax model, Baldwin & Lomax [1] of which the presentation also is left out.

$k - \tau$ model

The $k-\tau$ model, suggested by Speziale, Abid & Anderson [11] can be obtained from the standard $k-\epsilon$ model, Wilcox [12], by a variable transformation. If the turbulent time scale τ $(= k/\epsilon)$, is introduced in a low Reynolds number $k - \epsilon$ model the following equations are obtained after some manipulations.

$$\frac{\partial \rho k}{\partial t} + U_i \frac{\partial \rho k}{\partial x_i} = \tau_{ij}\frac{\partial U_i}{\partial x_j} - \frac{\rho k}{\tau} + \frac{\partial}{\partial x_i}\left[(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_i}\right] \tag{1}$$

$$\begin{aligned} \frac{\partial \rho \tau}{\partial t} + U_i \frac{\partial \rho \tau}{\partial x_i} = {} & (1 - C_{\epsilon 1} f_1)\frac{\tau}{k}\tau_{ij}\frac{\partial U_i}{\partial x_j} + \\ & \rho(C_{\epsilon 2} f_2 - 1) + \\ & \frac{2}{k}(\mu + \frac{\mu_t}{\sigma_\epsilon})\frac{\partial k}{\partial x_i}\frac{\partial \tau}{\partial x_i} - \\ & \frac{2}{\tau}(\mu + \frac{\mu_t}{\sigma_\epsilon})\frac{\partial \tau}{\partial x_i}\frac{\partial \tau}{\partial x_i} + \\ & \frac{\partial}{\partial x_i}\left[(\mu + \frac{\mu_t}{\sigma_\epsilon})\frac{\partial \tau}{\partial x_i}\right] + \\ & \frac{\tau}{k}\frac{\partial}{\partial x_i}\left[\mu_t\left(\frac{1}{\sigma_k} - \frac{1}{\sigma_\epsilon}\right)\frac{\partial k}{\partial x_i}\right] \end{aligned} \tag{2}$$

$$\tau_{ij} = \mu_t\left[\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right]$$

where f_1 and f_2 are damping functions and $C_{\epsilon 1}$, $C_{\epsilon 2}$, σ_k and σ_ϵ are model constants, which all are equivalent to the functions and model constants in the $k-\epsilon$ model. The definition of the eddy viscosity also follows from the variable transformation, i.e.

$$\mu_t = \rho C_\mu f_\mu k \tau$$

where C_μ and f_μ are equivalent to the constant and function from the $k - \epsilon$ model. In these equations Speziale et al. [11] suggested that the Prandtl numbers σ_k and σ_ϵ should be equal in which case the last term vanishes.

Close to the wall the leading order terms of the τ equation are the constant part of the destruction term and the part of the diffusion term representing molecular diffusion.

$$\frac{2\mu}{k}\frac{\partial k}{\partial y}\frac{\partial \tau}{\partial y} - \frac{2\mu}{\tau}\frac{\partial \tau}{\partial y}\frac{\partial \tau}{\partial y} + \mu\frac{\partial^2 \tau}{\partial y^2} - 1 = 0$$

These terms and the leading order terms of the k equation are exact. Which is a clear advantage over the $k - \epsilon$ model. In the ϵ equation the turbulent diffusion, viscous diffusion and the destruction terms have leading order terms. Of these terms only the viscous diffusion is exact, and the other two terms are modeled. Hence, one can expect that the $k - \tau$ model will be able to reproduce the asymptotic behavior for k and τ better than the $k - \epsilon$ model in the close vicinity of solid walls.

The term $C_{\epsilon 2} f_2 - 1$ in the τ equation corresponds to the dissipation and destruction terms in the k and ϵ equations. Apart from the variation of f_2 close to solid walls this term is constant. For homogeneous and isotropic turbulence without any mean shear τ will increase with a constant rate regardless of the initial state, i.e. regardless of turbulence level, anisotropic in the Reynolds stress tensor etc. Further, if the value of the constant $C_{\epsilon 2}$ is taken less than unity τ will instead decrease with a constant rate. This decoupled behavior from the physical processes is due to the approximation of the destruction term in the dissipation equation where

$$2\mu\overline{u_{i,k}u_{i,l}u_{k,l}} + 2\mu^2\overline{u_{i,kl}u_{i,kl}} \approx \rho C_{\epsilon 2} f_2 \frac{\epsilon^2}{k}$$

This behavior of the turbulent time scale is thus not a specific feature of the $k-\tau$ model. It is certainly present also in the $k - \epsilon$ model, although it is not obvious from the equations in their original form.

Near wall treatment

In the present calculations the original damping functions suggested by Speziale et al. [11] are replaced by new functions. These new functions are deduced from the asymptotic behavior of the exact terms close to walls and from comparison with DNS data, Lindberg [6]. The DNS data used for the derivation of the damping functions are from Mansour, Kim & Moin [7]. These DNS are performed at low Reynolds number not high enough to give Reynolds number independent values for the model constants, Rodi & Mansour [9]. This leaves some potential of further develop the damping functions using more recent DNS. The damping functions are expressed as functions of y^+. This is a physically well motivated choice for flows without recirculation etc. However, for separating flows, where the friction velocity goes to zero, this may not be the best choice.

The damping functions used in the present calculations are

$$f_1 = 1 + 2.05e^{-0.016(y^+-3)^2} - 0.6e^{-0.1y^{+2}}$$

$$f_2 = 1 - e^{-0.02y^{+2}} + 0.2ye^{-0.1(y^+-7)^2} + 0.04ye^{-0.11(y^+-12)^2}$$

$$f_\mu = \frac{\tanh 0.00039y^{+2}}{\tanh 0.0025y^{+3}}$$

and the model constants have the following values:

$$C_{\epsilon 1} = 1.44 \qquad C_{\epsilon 2} = 1.92 \qquad \sigma_k = 1.36$$

$$\sigma_\epsilon = 1.36 \qquad C_\mu = 0.09$$

Numerical method

The equations has been solved with a cell centered finite volume code, NSMB (Navier-Stokes Multi Block), using explicit Runge-Kutta time stepping. The equations for the turbulent quantities has been solved simultaneously with the other variables also using the explicit time stepping.

The transport equations has been discretized with standard second order methods. Which requires that node values are obtained from linear interpolation of cell center values. In most cases this is adequate if a fine enough mesh is used. However, k and τ are both $\sim y^2$ close to solid walls and the discretization of the k and τ equations must be able to resolve this quadratic behavior. The extra diffusion terms in the τ equation (third and fourth row in equation (2)) needs this special attention. If the terms are treated as source terms in a cell centered finite volume approach the gradient of k and τ in the cell center has to be calculated. The usual way to calculate these gradients is to take the surface integral over the cell volume which requires that the value at the cell surfaces are known. If these values are calculated as the mean of the cell center values large discretization errors will occur close to solid boundaries. If $y = 2h$ at the first node (surface) at a wall k and τ are $\sim 4h^2$ there. However the mean of the values in the first and the second cell is $\sim 5h^2$. This results in an error of 25%, as the gradients are multiplied with each other the total discretization error is over 50%.

If third order interpolation, using the four points around the surface, is used the error can be reduced significantly. However, the usual boundary condition for walls where the value in the ghost cell is set equal to minus the value in the first interior cell can not be used, as this assumes a linear profile. The values in the first and second ghost cells have to be set so that the third order polynomial is equal to zero at the wall.

The discretization of the viscous terms in the fourth row of equation 2 is also important. In finite volume approach the second order derivative is calculated as the surface integral over the gradients of τ. At solid walls the gradient is needed at the wall which requires integration over the ghost cell. Also here the usual boundary condition to put the ghost cell value to minus the value in the first interior cell gives large errors. As τ is quadratic close to the wall the gradient should be zero at the wall. In order to come around this problem a second order polynomial through the two first cells and the zero value at the wall was used to calculate the value in the ghost cell.

A standard Jameson 2nd and 4th order artificial dissipation has been used, Hirsch [4]. For the turbulent equations approximately a tenth of the artificial dissipation for the velocity was used, close to walls the artificial dissipation was damped by multiplying with an exponential function

$$1.0 - e^{-0.01y^{+2}}$$

If the artificial dissipation not was damped it was of the same order as the viscous terms and the shape of the mean velocity profile was distorted.

Transition from laminar to turbulent flow was forced at $x/c = 0.12$ on the suction side and at $x/c = 0.3$ on the pressure side of the airfoil. The source terms and the eddy viscosity was multiplied by a factor which was zero in the laminar part and varied linearly to unity in the transition region. As free stream conditions $k = 0.001 * U_0^2$ and τ was given a value so that the eddy viscosity was 0.1% of the molecular viscosity.

The calculations were performed using a C-mesh with 352x64 elements, it was the same mesh as the partners in EUROVAL.

Results

The results presented here are produced using the present $k-\tau$ model and to some extent the Baldwin-Lomax model. In the same code used for these calculations an equivalent $k-\epsilon$ model, with the same damping functions etc., is implemented. As the $k-\tau$ and $k-\epsilon$ models are mathematical identical they should give the same result except close to solid surfaces where the boundary conditions differ. However, they are not identical from a numerical point of view and the experience with the two models shows that the $k-\tau$ mode is more robust. It was no problem with starting the $k-\tau$ model from free stream conditions using different free stream values for k and μ_t. Hence it was not necessary to start from a Baldwin-Lomax calculation or to freeze the mean field for the first iterations as reported by some of the partners in EUROVAL, Haase et al [3]. The $k-\epsilon$ model in the present code required a free stream value of μ_t around ten times the molecular viscosity and also a smaller CFL number. A large eddy viscosity in the free stream does probably not influence the results to a large extent though the velocity gradients are small there, hence also the Reynolds stresses.

The experimental data used for comparison are the results from ONERA, France, used in one of the tasks in the BRITE/EURAM programme EUROVAL, Kourta [5]. In these experiments lift, drag, pressure distribution etc were measured for a cord Reynolds number $Re = 2 \times 10^6 - 5 \times 10^6$, $M = 0.15$ and for angles of attack between $\alpha = 7°$ and $\alpha = 15°$. In the experiments the transition point was fixed on the pressure side of the airfoil to $x/c = 0.3$ but free on the suction side.

In the EUROVAL project several partners participated and calculated the test case using algebraic, two-equation and ASM turbulence models. The integral results from the present calculations are presented in table 1 together with the EUROVAL results.

	$\alpha = 7.2°$		$\alpha = 12.3°$		$\alpha = 13.3°$	
	C_l	C_d	C_l	C_d	C_l	C_d
B-L	1.03	0.016	1.52	0.025	1.58	0.027
$k - \tau$	0.99	0.018	1.50	0.026	1.59	0.028
Exp	1.03	0.016	1.46	0.024	1.52	0.031
min	1.02	0.012	1.60	0.016	1.64	0.017
max	1.07	0.013	1.60	0.016	1.69	0.021

Table 1: Lift and drag coefficients, B-L: Baldwin-Lomax model, $k - \tau$: present $k - \tau$ model, Exp: Experiments EUROVAL, min/max: lowest/highest value from partners in EUROVAL using the $k - \epsilon$ model

The integral values are better predicted with the present $k - \tau$ model than by the partners in EUROVAL except for the lift coefficient for the lowest angel of attack. In this case the forced transition in the calculations may be too early compared to the free transition in the experiments which may influence the result.

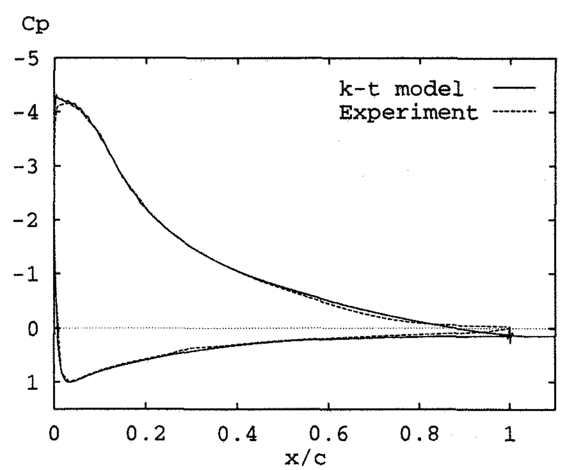

Figure 1: C_p calculated with $k - \tau$ model for $\alpha = 13.3°$ and $Re = 2 \times 10^6$ compared with the EUROVAL experiment

The pressure distribution calculated for $\alpha = 13°$, figure 1, closely follows the experimental data except at the trailing edge. In this region the flow has separated in the experiment but not in the calculations.

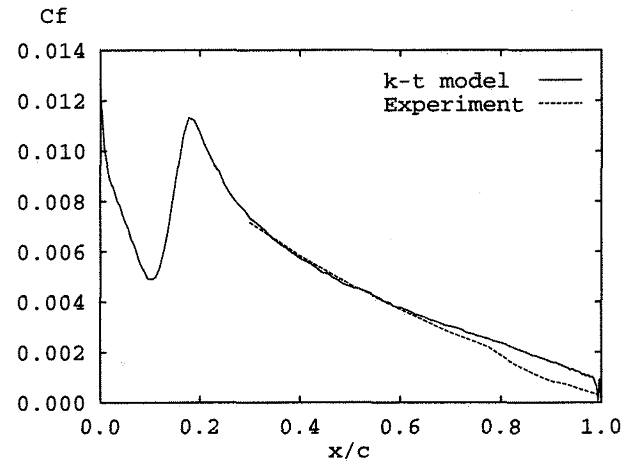

Figure 2: C_f calculated with $k - \tau$ model for $\alpha = 7.2°$ and $Re = 2 \times 10^6$ compared with the EUROVAL experiment

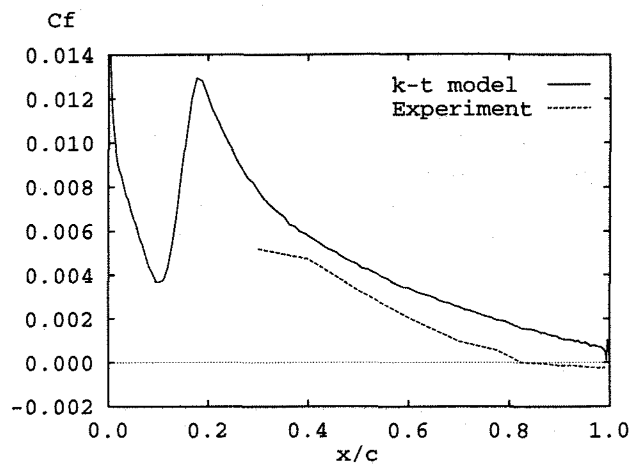

Figure 3: C_f calculated with $k - \tau$ model for $\alpha = 13.3°$ and $Re = 2 \times 10^6$ compared with the EUROVAL experiment

The friction coefficient on the suction side of the profile, figures 2 & 3, closely follows the experiments for the lower angel of attack, $\alpha = 7°$, where no separation occurs. For $\alpha = 13°$ the flow separates around $x/c \approx 0.8$ in the experiments but is attached in the calculations, which is seen more clearly in the velocity profiles below. The transition from laminar to turbulent flow is obtained by linearly increase the source terms and eddy viscosity between $x/c = 0.07$ and $x/c = 0.17$ which also were used by the EUROVAL contributions.

The choice of transition point does indeed influence the result which calls for reliable transitions models for use in practical calculations where the transition point is unknown.

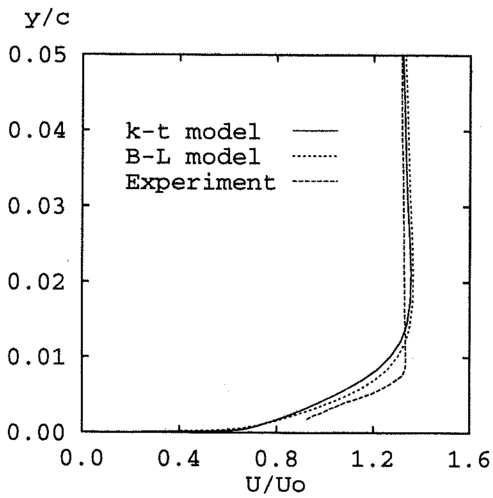

Figure 4: Mean velocity on the suction side of the profile scaled with free stream velocity U_o at x/c=0.4 and $\alpha = 7.2°$

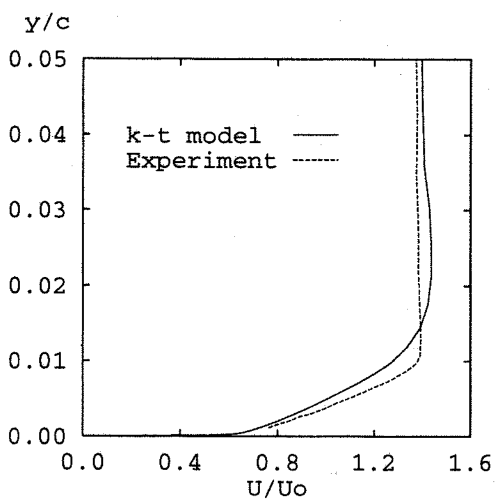

Figure 5: Mean velocity on the suction side of the profile scaled with free stream velocity U_o at $x/c = 0.4$ and $\alpha = 13.3°$

The mean velocity profiles at $x/c = 0.4$, figures 4 & 5, are predicted fairly good. The sharp outer edge of the boundary layer is not reproduced in the calculations. Mainly because of the calculation grid which is quite coarse in this region. In figure 4 there are about four grid points between $y/c = 0.005$ and $y/c = 0.01$. Hence, the rapid change in the velocity gradient is smeared out. However, at the airfoil surface the spacing is adequate with the first point located at $y^+ < 1$.

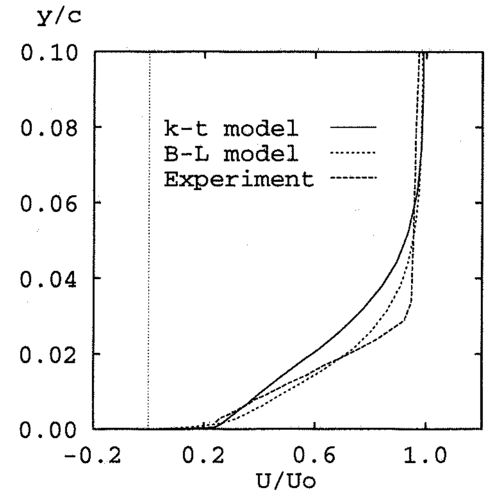

Figure 6: Mean velocity on the suction side of the profile scaled with free stream velocity at $x/c = 0.93$ and $\alpha = 7.2°$

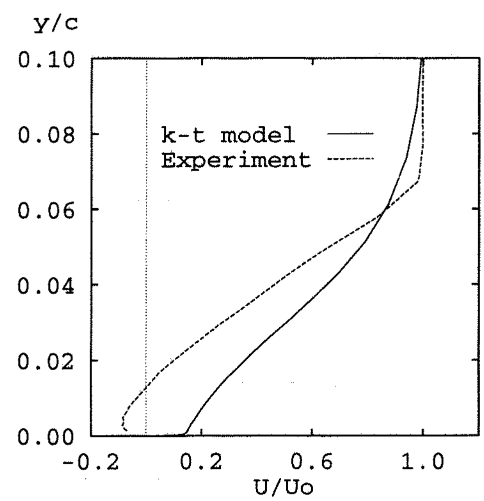

Figure 7: Mean velocity on the suction side of the profile scaled with free stream velocity at $x/c = 0.93$ and $\alpha = 13.3°$

Further down stream, figures 6 & 7, the mean velocity at $\alpha = 7°$ is still predicted reasonably, although the edge of the boundary layer is even more smeared out. For the higher angle of attack, $\alpha = 13°$, where the flow is separated in the experiments the calculated velocity is positive at the wall. In this case the curvature effects may be important and this can only be captured by an algebraic or full Reynolds stress model. This is also confirmed by the EUROVAL results where only the ASM contribution gave a correct velocity profile at this location.

Moreover, the damping functions used in the present $k-\tau$ model are based on y^+. At the separation point the friction velocity is zero and hence y^+ is zero along a line normal to the wall. This may lead to unwanted effects. In this case a damping function based on e.g. the turbulent Reynolds number $Re_t = \rho k^2/\mu\epsilon$ is more adequate.

The Baldwin-Lomax model gives results in quite good agreement with the experiments. It is well suited for calculations where the flow is not far from equilibrium, e.g. for low angles of attack. As pointed out before higher angles of attack requires some sort of ASM models and therefore the presentation of the Baldwin-Lomax results are limited here. More results from calculations with the Baldwin-Lomax model can be found in Kourta [5].

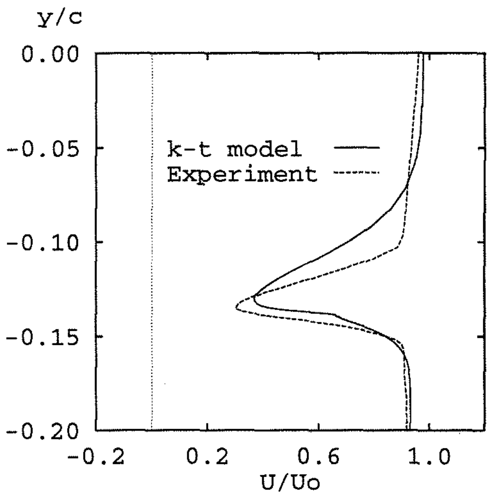

Figure 8: Mean velocity along a line perpendicular to the free stream velocity scaled with free stream velocity at $x/c = 1.05$ and $\alpha = 7.2°$, where $y = 0$ is located at the leading edge in lab. coordinates

The velocity field in the wake, figures 8 & 9, follows the same lines as close to the trailing edge. The wake is smeared out and the recirculation zone seen in the experiments is not reproduced by the calculations.

The turbulent kinetic energy compares fairly well with the experiments, figure 10 shows a typical example. The calculated k is somewhat higher than the experimental value but the location of the maximum is correct.

Conclusions

Calculations of the airfoil A has been performed for $Re = 2 \times 10^6$, $M = 0.15$ and $\alpha = 7° - 13°$ using the $k-\tau$ model. The experience with the model shows that if the discretization of the k and τ equations is accurate

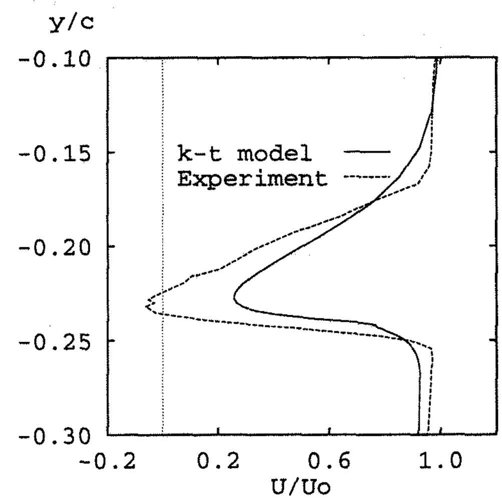

Figure 9: Mean velocity along a line perpendicular to the free stream velocity scaled with free stream velocity at $x/c = 1.05$ and $\alpha = 13.3°$, where $y = 0$ is located at the leading edge in lab. coordinates

and is able to correctly handle the quadratic behavior of k and τ close to solid walls the model is more robust than an equivalent $k-\epsilon$ model, using the same damping functions. The results indicates that the model predicts lift and drag coefficient in with comparable or better agreement with experimental data than the $k-\epsilon$ models used by the contributions to EUROVAL, Haase et al. [3]. An explanation for the better agreement may be the new damping functions. These damping functions are based on comparison with DNS data. They may though be further improved for calculation of separated flows. For a correct prediction of the airfoil stall a more advanced model has to be used, e.g. an algebraic Reynolds stress model. The present calculations shows that the $k-\tau$ model is a good candidate to serve as a base for such a model.

References

[1] B.S. Baldwin and H. Lomax. Thin layer approximation and algebraic model for separated turbulent flows. *AIAA-paper 78-257*, 1978.

[2] L. Davidson and A. Rizzi. Navier-stokes computation of airfoil in stall using algebraic reynolds-stress model. *AIAA-paper 92-0195*, 1992.

[3] W. Haase, F. Bradsma, E Elsholz, M. Leschziner, and D. Schwamborn, editors. *EUROVAL - An European Initiative on Validation of CFD Codes*, volume 42 of *Notes on Numericall Fluid Mechanics*. Vieweg, 1994.

[4] C. Hirsch. *Numerical Computation of Internal and External Flows*. John Wiley & Sons, 1988.

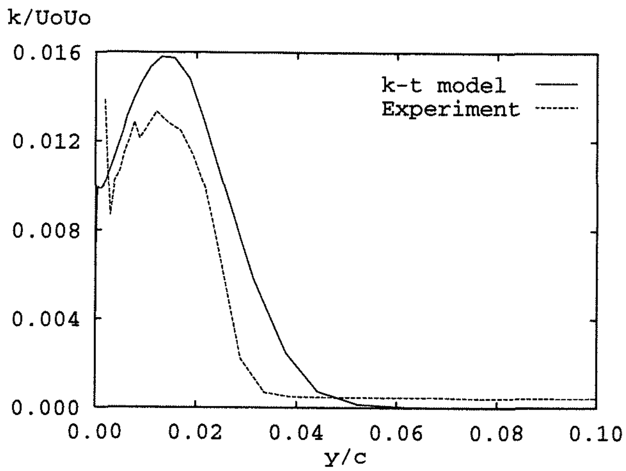

Figure 10: Turbulent kinetic energy on the suction side of the profile, scaled with free stream velocity, at $x/c = 0.93$ and $\alpha = 7.2°$

[5] A. Kourta. Maximum lift investigation for a-airfoil. In Haase W. et al., editors, *EUROVAL - An European Initiative on Validation of CFD Codes.* Vieweg, 1994.

[6] P.-Å. Lindberg. Near-wall turbulence models for 3d boundary layers. *Applied Scientific Research*, 1994.

[7] N.N. Mansour, J. Kim, and P. Moin. Reynolds-stress and dissipation-rate budgets in a turbulent channel flow. *Journal of fluid mechanics*, 194, 1988.

[8] V.C. Patel, W. Rodi, and G. Scheuerer. Turbulence models for near-wall and low reynolds number flows: A review. *AIAA Journal*, 23(9), 1985.

[9] W. Rodi and N.N. Mansour. Low reynolds number $k-\epsilon$ modeling with the aid of direct simulation data. *Journal of fluid mechanics*, 250, 1993.

[10] T.-H. Shih, J. Zhu, and J. Lumely. A realizable reynolds stress algebric equation model. Technical Report TM 105993, NASA, 1992.

[11] C. Speziale, R. Abid, and C. Anderson. A critical evaluation of two-equation models for near-wall turbulence. *AIAA paper 90-1481*, 1990.

[12] D. Wilcox. *Turbulence Modeling for CFD.* DCW Industries, Inc., La Cañada, California, 1993.

Acknowledgement

The calculations presented here have been performed at the computer center at Ecole Polytechnique Federale de Lausanne, Switzerland. Without this support this work had not been possible and it is gratefully acknowledged. Part of this work has been sponsored by the Swedish National Board for Industrial and Technical Development and by the EURASMUS exchange programme.