SIMULATION OF UNSTEADY TURBULENT FLOW OVER A STALLED AIRFOIL

B. STOCKDILL† G. PEDRO† A. SULEMAN† F. MAGAGNATO‡ N. DJILALI†

Abstract
Unsteady, turbulent flow simulations are carried out for a NACA 0012 airfoil. The objective is to assess the performance of various turbulence models for application to fluid-structure interaction and airfoil optimization problems. Unsteady simulations are carried out at an angle of attack of 20 degrees and a chord Reynolds number of $10^5$. The Spalart Allmaras and Speziale $k-\tau$ models are used in unsteady Reynolds-Averaged Navier-Stokes (URANS) simulations. Filtered Navier Stokes simulations in conjunction with an adaptive $k-\tau$ model proposed by Magagnato are also presented. This adaptive model can be used at all grid or temporal resolutions and tends asymptotically to DNS or RANS in the limit of fine grid resolution or steady state simulations. This model produces physically realistic flow patterns, rich vortex dynamics and non-periodic lift and drag signals. The $k-\tau$ model also yields unsteady flow patterns, but these remain purely periodic, whereas only steady state solutions are obtained with the Spalart Allmaras model.

Key Words: Unsteady RANS; Turbulence model; Flow separation; Stall

1 INTRODUCTION
Simulation of massively separated flows over an airfoil beyond stall is, along with prediction of stall itself, a very challenging problem. Panel methods [1] and inviscid Euler computations, which up until recently have been the work horses of aerodynamic design in Industry, are unable to predict flow separation and stall, and extensive reliance on wind tunnel measurements is still required to characterize these flow regimes [2].

These regimes are characterized by unsteadiness and complex turbulence dynamics. Though in principle represented in the Navier-Stokes (NS) equations, the computational resolution of these processes is prohibitive for most aerodynamic flows of practical importance. The practical limit of a Direct Numerical Simulation (DNS) is $Re_c = 10^4$, two orders of magnitude less than an airfoil in the landing configuration [3].

Since designers are primarily interested in the time averaged values of shear stress, pressure and velocity rather than the time dependent details, practical information and design assessment can often be obtained using the Reynolds Averaged Navier Stokes (RANS) simulations in conjunction with appropriate closure models to represent the effect of turbulent stresses. Engineering turbulence models, such as the classical $k-\epsilon$ model, often achieve this via a turbulent eddy viscosity. A weakness common to both eddy-viscosity and Reynolds stress RANS models is the assumption that a single time (or length) scale characterizes both turbulence transport and dissipation of turbulent kinetic energy, and the energy cascade from large to small eddies is not accounted for. Some of these shortcomings can be addressed using multiple scale models (e.g. [4]), but it is clear that Large Eddy Simulations (LES) offer a more satisfactory representation of turbulence transport by resolving most of the dynamics of the flow and confining turbulence modelling to unresolved small scale fluctuations only [5]. Very encouraging progress has been made in LES computations coupled with dynamic subgrid models, but this approach is computationally expensive and is not viable for routine engineering applications, particularly at higher Reynolds numbers. Consequently, intermediate approaches based on Unsteady Reynolds Averaged Navier-Stokes methods (URANS) as well as hybrid URANS/LES methods [5] are seen as the most
viable short and mid-term alternative.

Major efforts have been devoted to simulation and turbulence model validation for airfoils near maximum lift, and an overview of these is provided in the next Section. In this paper we investigate the ability of three turbulence models to predict massively separated flows and vortex shedding in the post stall regime. Unsteady simulations are presented using two classical eddy viscosity models (Spalart Allmaras one-equation, Speziale k − τ two-equation), and a new adaptive turbulence model [7] which has the property of adapting to all cell Reynolds numbers and which reduces the simulations to a DNS in the limit of fully resolved Kolmogorov scales, and to RANS in the limit of steady state simulations. To keep the computational requirements reasonable, and, in the interest of predicting NACA 0012 characteristics at a Reynolds number for which there is little experimental data, the set of simulations is carried out at $Re_c = 10^5$. The flow over the NACA 0012 airfoil flow remains laminar at low angles of attack ($\alpha < 10^5$) for $Re_c = 10^5$, whereas separation, Kelvin Helmholtz instabilities, turbulence and vortex shedding are induced at post stall angles of attack [8].

2 Overview of CFD Methods for Airfoil Aerodynamics

Recent CFD airfoil validation projects include the European Initiative on Validation of CFD Codes (EUROVAL), see [9,10] and the European Computational Aerodynamics Research Project (ECARP). The goal of these projects was to determine the ability of various turbulence models to predict separation and maximum lift just before stall. Aerospatiale’s A-Airfoil, for which there is extensive experimental data, was used for validation in both projects. A standard grid with 96,512 grid points was used for all simulations. High Reynolds number flow in this conditions is one of the most challenging cases for CFD. ECARP concluded that algebraic and standard eddy viscosity models fail to correctly predict separation and therefore maximum lift at high angles of attack.

Weber and Ducros [10] conducted RANS and LES simulations over the A-Airfoil near stall at $Re_c = 2.1 \times 10^6$. They used the ECARP C type grid with 512 points along the airfoil surface, 64 points in the cross stream direction and 498 points in the downstream direction. They allowed 10 chord lengths to far field boundaries in the upstream and cross stream directions. Wall functions were not used; the first grid point from the wall was located at approximately $y' = 2$. While the classic turbulence models were able to predict many features such as a laminar boundary layer close to the leading edge, separation at the trailing edge and the interaction of two shear layers in the wake, they failed to correctly predict the separation zone at high angles of attack. Spalart Allmaras model results were found to be quantitatively close to LES, which required 140 times more computer resources than the Spalart Allmaras model on the same grid. Convergence with RANS was however not obtained at high angles of attack, and neither RANS nor LES were able to predict boundary layer separation with the correct back flow characteristics.

Chaput [9] carried out validations of turbulence models under ECARP using the EUROVAL Aerospatiale A-Airfoil grid. Sixteen partners are involved in the project, including BAe, DLR, Dornier/Dasa-LM and SAAB. The focus is a comparison between the performance of various turbulence models at maximum lift. The extensive database of numerical and experimental data from the EUROVAL project is used as a starting point. The project does not include the algebraic Baldwin Lomax and the standard $k-c$ models. While these models are widely used in industry, their widespread failure to predict separation is well documented. Examples include Refs.[11,12].

EUROVAL and ECARP found that Navier Stokes methods are capable of accurate prediction of boundary layer separation in pre-stall conditions but that accurate prediction of maximum lift is not achieved. Good results can be obtained by using turbulence models which take into account the strong non-linear equilibrium processes at high angles of attack. Transition to turbulence near the leading edge is still problematic; turbulence models do not reliably predict transition on their own.

3 Definitions

3.1 Airfoil Dimensionless Parameters

The forces which act on an airfoil are functions of density $\rho$, molecular viscosity $\mu$, free stream velocity $U_{\infty}$ and chord length $c$. Through dimensional analysis, these variables can be combined into a single non-dimensional parameter, the chord Reynolds number $Re_c$ which is a measure of the ratio of inertial to vis-

![Fig.1: Layout of NACA 0012 Airfoil showing Chord, Leading Edge (LE) and Trailing Edge (TE)](attachment:image)
dynamic pressure

1

cohesive forces:

\[ \text{Fig. 2: Nomenclature of Flow Separation and Associated Velocity Profiles} \]

\[ \text{Re}_c = \frac{\rho U_{\infty} c}{\mu} = \frac{U_{\infty} c}{\nu} \]

where \( \nu = \frac{\mu}{\rho} \) is the kinematic viscosity. In analogy to the Reynolds number, the lift, drag and friction forces can also be written as non-dimensional coefficients. These are the coefficient of lift \( C_l \), the coefficient of drag \( C_d \), the coefficient of pressure drag \( C_{dp} \), the coefficient of friction drag \( C_{df} \) and the skin friction coefficient \( C_f \). Forces are non-dimensionalized by the dynamic pressure \( \frac{1}{2}(\rho U_{\infty}^2) \) and the wing area \( S \). In this work \( S \) is set to 1 since the simulations are two dimensional.

\[ C_l = \frac{\text{lift}}{\frac{1}{2} \rho U_{\infty}^2 S} \]

\[ C_d = \frac{\text{drag}}{\frac{1}{2} \rho U_{\infty}^2 S} \]

\[ C_{df} = \frac{\text{friction drag}}{\frac{1}{2} \rho U_{\infty}^2 S} \]

\[ C_{dp} = \frac{\text{pressure drag}}{\frac{1}{2} \rho U_{\infty}^2 S} \]

\[ C_f = \frac{\text{shear stress}}{\frac{1}{2} \rho U_{\infty}^2} \]

Pressure forces can also be non-dimensionalized using the difference between the local pressure \( P \), the free stream pressure \( P_{\infty} \) and the dynamic pressure:

\[ C_p = \frac{P - P_{\infty}}{\frac{1}{2} \rho U_{\infty}^2} \]

Figure 1 shows the geometry of a symmetric NACA 0012 airfoil.

- Chord (c): straight line joining the leading and trailing edges.
- Leading Edge (LE): point at the front of the airfoil where the chord line intersects the airfoil surface.
- Trailing Edge (TE): point at the rear of the airfoil where the chord line intersects the airfoil surface
- Angle of Attack (\( \alpha \)): Angle between the chord line and the free stream velocity vector \( U_{\infty} \).

Only the symmetric (zero camber) NACA 0012 airfoil is used in this work.

4 Vortex Shedding and Stall

It is well established that many structures, particularly bluff bodies, shed vortices in subsonic flow [13]. Vortex street wakes tend to be very similar regardless of the body geometry. A well documented example of vortex shedding is the flow around circular cylinders (see, e.g. [14]). The characteristic non-dimensional frequency or Strouhal number \( St \) for such flows is defined as:

\[ f_s = \frac{StU_{\infty}}{L} \]

where \( f_s \) is the vortex shedding frequency, \( U_{\infty} \) the wind speed, and \( L \) the characteristic length scale:

Vortex shedding is caused by flow separation, a viscous flow phenomenon associated with either abrupt changes in geometry or adverse pressure gradients. There are two main separated flow regimes for the NACA 0012 airfoil. The first corresponds to a laminar separation bubble form the leading edge (at about \( C_l \approx 0.9 \) and \( \alpha \approx 9^\circ \) for \( Re_c \approx 3 \times 10^6 \) [15]). The separated shear layer undergoes transition prior to reattachment.

The second separated flow regime corresponds to massive stall at about \( \alpha \approx 16^\circ \); a large separation bubble begins to form at the trailing edge and gradually moves forward. This regime is associated with the shedding of large vortices.

Stall for the NACA 0012 section is difficult to quantify experimentally. Gregory and O'Reilly [15] found that for \( Re_c \approx 3 \times 10^6 \), the stall may originate from the collapse of the leading edge laminar separation bubble. However, for higher \( Re_c \), the stall appears to originate entirely from the forward propagation of the trailing edge separation bubble.

Whereas vortex shedding for cylinders at low Reynolds numbers in the laminar regime is a stationary, harmonic two-dimensional phenomenon, this is not the case for turbulent flow. Vortex shedding at
high Reynolds numbers occurs over a narrow band of shedding frequencies and is highly three dimensional. In analogy with circular cylinder flow, airfoils form a vortex street at a Strouhal number of $St \approx 0.2$ based on the width $d$ of the boundary layers at the trailing edge [16,17]. Further, if $d$ is defined as the width between separation points, $St \approx 0.2$ applies over broad ranges of Reynolds numbers regardless of section geometry [17]. The characteristic dimension used in calculating the Strouhal number in this work is the chord length $c$.

5 Turbulence Models

The simulations presented in this paper are obtained using three different turbulence models. The first two are classical eddy viscosity turbulence models solved in conjunction with the unsteady Reynolds averaged Navier-Stokes equations (URANS). The third model is an adaptive two-equation model solved in conjunction with the filtered Navier-Stokes equations, and which adapts to the unresolved turbulent fluctuations.

5.1 Spalart-Allmaras One Equation Model

Closure of the governing equations is obtained by expressing the turbulent stress tensor in terms of a turbulent kinetic energy.

$$\tau_{ij} = -\rho\nu_{t} u_{j} = \mu_{t} \left( \frac{\partial u_{i}}{\partial x_{j}} + \frac{\partial u_{j}}{\partial x_{i}} \right) - \frac{2}{3} \rho \kappa \delta_{ij} \quad (9)$$

Where $\nu_{t}$ is the eddy viscosity and $k$ the turbulent kinetic energy.

In the classical Spalart-Allmaras model, a single equation is used to represent eddy viscosity transport based on empiricism, dimensional analysis, Galilean invariance and molecular viscosity $\mu$ where appropriate [19]. This local model is designed to be compatible with grids of any structure and can be used for 2D and 3D problems. It is calibrated for aerodynamic flows such as flat plate boundary layers, boundary layers with pressure gradient, mixing layers and wakes. [19] notes that while two equation models are quite sensitive to upstream and free stream turbulent parameters of length scale and turbulent intensity, the SA model can deal with zero values making it easier to set boundary conditions. This model has been quite popular in computational aerodynamics due to its robustness and relatively low computational overhead.

The transport equation for the eddy viscosity reads:

$$\frac{D\tilde{\nu}}{Dt} = \frac{C_{b1} [1 - f_{t2} \tilde{S} \tilde{\nu}]}{\nu_{t}} + \frac{1}{\sigma} \left[ \nabla \cdot (\nu + \tilde{\nu}) \nabla \tilde{\nu} + C_{b2} (\nabla \tilde{\nu})^{2} \right]$$

$$- \left[ C_{w1} \frac{\nu}{\kappa^2} f_{t2} \right] \left( \frac{\tilde{\nu}}{d} \right)^{2}$$

$$+ f_{t1} \Delta U^{2} \quad \text{tripping source term}$$

The role of each term is highlighted in the equation. Here, $\tilde{\nu}$ is equivalent to the eddy viscosity $\nu_{t}$ except in the viscous region where a viscous damping function $f_{t1} \Delta U$ is invoked to account for low Reynolds number near-wall flows. $\tilde{S}$ is the magnitude of the vorticity and $d$ is the distance from the nearest wall.

The implementation of the SA model in SPARC uses equation (10) but neglects the tripping source term. This term is designed to artificially increase the eddy viscosity in a near wall region around a user-specified turbulence transition point. The tripping point is not specified in this work; its importance is minimized by running simulations at either high Reynolds numbers where the transition occurs essentially at the leading edge, as will be shown later in shear stress plots, or at high angles of attack where the separation point is very near the leading edge and the location of the transition point is not expected to have much influence on the lift and drag coefficients.

5.2 Speziale Two Equation $k-\tau$ Model

In two equation models, the eddy viscosity is expressed in terms of a turbulent velocity scale and a turbulent length scale, which are obtained from the solution of appropriate transport equations. The $k-\epsilon$ model is the most widely used two-equation model. This model is robust and provides reasonable predictions for boundary layer and simple free shear flows. Its performance is however unsatisfactory in many respects for complex and/or separated flows, see e.g. [20]. In addition the standard version of this model does not account for low Reynolds number near-wall transport and requires the use of wall functions, which, again are not entirely appropriate for separated flows. Finally, there is no rigorous way of specifying wall boundary condition for $\epsilon$ [21]. The two equation $k-\tau$ model of Speziale, Abid and Anderson [21] was developed to overcome some of the
shortcomings of the standard $k - \epsilon$ models. Both the $k - \epsilon$ and $k - \tau$ models use transport equations for turbulent kinetic energy, but instead of the dissipation, which is used in the $\epsilon$ as the second transport equation, the $k - \tau$ model uses the turbulent time scale $\tau \equiv k/\epsilon$. Thus,

$$\frac{Dk}{Dt} = \tau_{ij} \frac{\partial \tau_{ij}}{\partial x_j} - \epsilon - D + \nu \nabla^2 k$$  \hspace{1cm} (11)

$$\frac{D\tau}{Dt} = \frac{\tau}{k} \tau_{ij} \frac{\partial \tau_{ij}}{\partial x_j} - \frac{\tau}{k} \frac{D\tau^2}{k} P_\epsilon + \frac{\tau^2}{k} \Phi_\epsilon + \frac{\tau^2}{D} D + \frac{2\nu}{k} \frac{\partial k}{\partial x_i} \frac{\partial \tau}{\partial x_i} + \nu \nabla^2 \tau$$  \hspace{1cm} (12)

Here $D$ represents the transport of turbulent kinetic energy $k$ by turbulence, and $D$ the transport of dissipation by turbulence; $P_\epsilon$ and $\Phi_\epsilon$ are production and destruction of turbulent dissipation terms. The final form of the transport equations for the turbulent kinetic energy $k$ and the turbulent time scale $\tau$ are:

$$\frac{Dk}{Dt} = \tau_{ij} \frac{\partial \tau_{ij}}{\partial x_j} - k \frac{\partial \tau}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right]$$  \hspace{1cm} (13)

$$\frac{D\tau}{Dt} = (1 - C_{\epsilon 1}) \frac{\tau}{k} \tau_{ij} \frac{\partial \tau_{ij}}{\partial x_j} + (C_{\epsilon 2} f_2 - 1) +$$

$$+ \frac{2}{k} \left( \nu + \frac{\nu_t}{\sigma_{\epsilon 1}} \right) \frac{\partial k}{\partial x_i} \frac{\partial \tau}{\partial x_i} - \frac{2}{\tau} \left( \nu + \frac{\nu_t}{\sigma_{\epsilon 2}} \right) \frac{\partial \tau}{\partial x_i} \frac{\partial \tau}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ \left( \nu + \frac{\nu_t}{\sigma_{\epsilon 2}} \right) \frac{\partial \tau}{\partial x_i} \right]$$  \hspace{1cm} (14)

The full set of ancillary equations, constants and damping functions are given in [21].

### 5.3 Adaptive $k - \tau$ Model

Though they can be adapted for unsteady simulations, Reynolds Averaged Navier Stokes (RANS) models are designed for reproducing the time-averaged mean quantities of the flow, and the constants are tuned based on time-averaged measurements obtained in stationary flows. These models assume that turbulent transport is dictated by the large eddy motion. In unsteady calculations, a fraction of the large scale unsteadiness is resolved by the simulations; the turbulence model should account only for the unresolved part of the turbulence. In well resolved Large Eddy Simulations (LES), only a small part of the turbulence dynamics (high wave number) needs to be modelled using a relatively simple subgrid scale model (see. e.g. [5]). However, in order to obtain simulations that are well within the intended use of the subgrid-scale models, a fine grid resolution is required. In addition to the difficulty in achieving proper resolution at higher Reynolds numbers, another issue in LES is the coupling between the filter and the grid size (cf. e.g. [22]).

Magagnato and Gabi [25] have proposed a new adaptive $k - \tau$ model that can be used for any grid resolution in the unsteady case. The model reduces to a standard two equation model in steady state and adapts to resolvable fluctuations based on grid size in unsteady calculations. The model was initially designed for highly unsteady and three dimensional turbo machinery flows, where RANS models are unable to adequately predict pressure losses and for which DNS and LES are too expensive due to the high Reynolds numbers. The adaptive $k - \tau$ model is a compromise between URANS and LES methods. Algebraic subgrid scale models are used to model the unresolved turbulence in LES. By using a non-linear two equation turbulence model based on the model of Craft/Lauder/Suga [23,24] rather than an algebraic model, the adaptive $k - \tau$ model can model a broader range of subgrid scale turbulence. Accordingly, computational costs may be reduced by using a coarser mesh.

In the limit of a very coarse grid, the adaptive $k - \tau$ model approaches a RANS simulation. In the limit of a very fine grid in three dimensions with cell Reynolds number length scales approaching the Kolmogorov length scales, the adaptive model simulations approach DNS [25].

The filtering process in LES and RANS is in principle different; LES filtering is based on a spatial scale determined by the grid (implicit) while RANS filtering is obtained by time averaging. The adaptive $k - \tau$ model assumes that the implicit filtering in LES becomes, in the limit of high cell Reynolds numbers, similar to the Reynolds time averaging process [25], and breaks the turbulent kinetic energy $k$ and turbulent time scale $\tau$ into resolved and unresolved parts:

$$\tau = \tau' + \overline{\tau}, \quad k = k' + \overline{k}$$  \hspace{1cm} (15)
The resolved part of the turbulent kinetic energy $\overline{k}$ is obtained as part of the numerical solution. The unresolved part $k'$ is modelled with an additional transport equation. The resolved time scale is obtained \textit{aposteriori} using a model for isotropic high Reynolds number flows where $\overline{k}$ is determined by the unresolved turbulent kinetic energy $k'$ and the filter length scale $L_{\Delta}$:

$$\overline{k} = \frac{L_{\Delta}}{\sqrt{k'}}$$ (16)

The filter length scale ($L_{\Delta}$) is determined by taking the maximum of the grid length scale ($L_g$) and the time step or temporal length scale ($L_t$):

$$L_g = 2 \cdot \sqrt{\Delta x \cdot \Delta y \cdot \Delta z}, \quad L_t = |u| \cdot \Delta t$$ (17)

$$L_{\Delta} = \max(L_g, L_t)$$ (18)

The subgrid scale stresses are obtained as follows:

$$-\rho u_i' u_j' = \rho c_{\mu} k' \tau' S_{ij} - \frac{2}{3} \rho k' \delta_{ij} - \rho v_i' v_j' + \frac{2}{3} \rho k' \delta_{ij}$$ (19)

where $v_i'$ are random velocities estimated at each time step using a Langevin-type equation. An important feature of the adaptive model is that it accounts for backscatter (second half of the RHS of equation 19).

The transport equations for the unresolved turbulent kinetic energy $k'$ and time scale $\tau'$ are obtained by recasting the model of [23]. Further details on the adaptive model can be found in [25].

6 Computational Procedure

The research code SPARC (Structured PArallel Research Code) developed at the University of Karlsruhe, Germany [25], is used in this study. The modular structure of SPARC is particularly suited to multi-team developments and collaboration, and the code is being integrated into a fluid-structure interaction framework at UVic.

SPARC is a finite-volume, structured, parallel multi-block code. In the simulations presented here, the full compressible form of the RANS equations are solved for the Spalart Allmamras and Speziale $k-\tau$ turbulence models [26], and the full weakly compressible Navier-Stokes equations with implicit filtering set by the grid resolution are solved for the adaptive $k-\tau$ model. A two stage semi-discrete method process is used. The first stage is a central difference (cell centered) finite volume discretization in space, and the second stage is an explicit Runge-Kutta discretization in time. The space discretization reduces the partial differential equations to a set of ordinary differential equations continuous in time.

A numerical dissipation term is introduced in the central differencing scheme to account for high frequency cascading. The dissipation term is a blend of second and fourth order differences where the second order differences are designed to prevent spurious oscillations, particularly around shocks in compressible flow, and the fourth order terms are designed to improve stability and steady state convergence [25].

An efficient full multigrid solver with implicit averaging technique is used. The multigrid implementation in SPARC has been found to yield up to tenfold convergence acceleration [25].

7 Simulation Results

Prior to discussing the results it should be noted that the choice of $Re_c = 3 \times 10^5$ for the simulations was dictated by computational costs as well as a focus for the eventual application of this work to investigate low-intermediate Reynolds number fluid-structure interaction problems in flapping propulsion [14] and for small aerial vehicles such as gliders and very high altitude aircraft (greater than 30,000 m) operating in this regime [27,28]. Also low Reynolds number airfoil data is of interest since little experimental data is available at $Re_c = 10^5$, particularly in stall conditions.

For the NACA 0012 airfoil at low angles of attack ($\alpha < 10^\circ$) there is no appreciable separation of the mean flow and the time averaged shear stress at every point on the airfoil surface is positive. There are of course small scale velocity fluctuations in the turbulent boundary layer and some negative instantaneous velocities, but their time average remains positive.

As the angle of attack increases above 13°, the adverse pressure gradient on the upper side of the airfoil leads to flow separation. The mean velocity close to the wall becomes negative in certain locations. Large vortices whose diameters are similar in scale to the chord length are shed; there is significant variation in the coefficients of lift ($C_l$) and drag ($C_d$) with time. The physics of such separated flows become much more complex and are therefore much more challenging for the turbulence models [12,29].

Since the Reynolds number is $Re_c = 10^5$, the mesh
resolution required normal to the wall is not too demanding. The first grid point from the wall is again located at $y^+ \approx 1$. The grid is refined in the $x$ direction along the upper surface of the airfoil, and at the leading and trailing edges. The domain size remains the same, with far field boundaries applied everywhere except at the airfoil wall (solid boundary). Rather than apply an angle of attack by changing the flow direction of the free stream, the airfoil grid is tilted; this ensures sufficient downstream length before the outlet. Convergence problems were encountered for the case $\alpha = 20^\circ$ using free stream tilting rather than tilting the grid; the wake reached the far field boundaries too quickly (see figure 3).

The grid has 329 grid point along the airfoil surface in the $x$ direction, 209 of which are on the upper surface. There are 121 points in the cross stream ($y$) direction and 225 points in the downstream ($x$) direction. The outlet flares in the cross stream direction to reduce cell aspect ratios at $x = 0, y > 15$. Convergence difficulties were encountered for aspect ratios greater than 400.

A systematic grid study is not undertaken for the unsteady simulations. It would be difficult to achieve due to the computation time required and the unsteady nature of the results. An extensive set of simulations were performed using the steady state solver at low angles of attack. By using a grid similar to that used in the steady state simulations, with wall resolution at $y^+ \approx 1$ and more points along the upper airfoil surface, reasonable unsteady results are expected. The interest for the unsteady simulations is focused on a qualitative view of the vortex shedding phenomena and the impact of the turbulence model rather than an exact prediction of the lift and drag.
7.1 Spalart Allmaras

The Spalart Allmaras model did not produce an unsteady flow pattern at \( \alpha = 0^\circ \) and \( Re_c = 10^5 \) with the free stream eddy viscosity set to 1. At a low free stream eddy viscosity of \( 10^{-6} \), oscillations were present in the lift and drag signals but there were large eddy viscosity gradients near the domain boundaries. It was not possible to converge the solution at each time step. To overcome the problems at the boundaries and to stabilize the solution, the eddy viscosity was set to 1. For the sake of consistency, this matches the boundary conditions applied to the adaptive \( k-\tau \) and Speziale \( k-\tau \) models. It should also be noted that the Spalart Allmaras model implemented in SPARC has been tuned for steady state solutions; it appears that applying it to unsteady problem would require at least adjustment of the model constants to avoid excessive damping. The poor ability of this model in capturing the dynamics of recirculating flows was also noted recently by [6]. To allow comparison, contour plots of the Spalart Allmaras results at \( \alpha = 20^\circ \) and \( Re_c = 10^5 \) are presented with time averaged results for the Speziale \( k-\tau \) and adaptive \( k-\tau \) models.

7.2 Speziale \( k-\tau \)

Figure 4 shows the lift and drag signals for the KTS model at \( Re_c = 10^5 \) and \( \alpha = 20^\circ \). The frequency and phase of the lift and drag are identical; the signal is periodic. The period is approximately 0.1 seconds which corresponds to a Strouhal number of \( St = 0.55 \) based on the chord length \( c \), the free stream velocity \( U_\infty = 18.125 m/s \) and the shedding frequency. The mean values of the lift and drag coefficients are \( C_l \approx 0.85 \) and \( C_d \approx 0.33 \). The time step for these computations is 0.005 seconds; this corresponds to 20 time steps per shedding cycle.

Streamlines for the vortex shedding cycle are shown in Figure 5 with time increments of 0.2\( T \) where \( T \) is the cycle period. The large separated clockwise rotating vortex in the \( 0T \) frame gradually moves downstream. It detaches from the airfoil at approximately 0.4\( T \). Note that a counterclockwise rotating vortex immediately downstream of the clockwise rotating one has already separated. At 0.6\( T \) a counterclockwise rotating trailing edge vortex edge appears. It is shed at 0.8\( T \) and a clockwise rotating vortex is re-established at the leading edge. Two vortices are shed in each cycle, one generated at the leading edge and the other at the trailing edge.

Figure 6 shows the pressure contours. Note that there is a low pressure region at the centre of each vortex.

Fig.10: Unsteady Lift and Drag Signals for Adaptive \( k-\tau \) Model at \( \alpha = 20^\circ \) and \( Re_c = 10^5 \).

Figures 7 and 8 present the eddy viscosity ratio and turbulent kinetic energy ratio, also in increments of 0.2\( T \). Note that the maximum eddy viscosity occurs in the wake between \( x/c = 1 \) and \( x/c = 2 \). The maximum turbulent kinetic energy occurs at the trailing edge and in the near wake at \( x/c = 1 \) to \( x/c = 2 \) which also corresponds to the area of maximum eddy viscosity. There is a turbulent kinetic energy peak in the separated shear layer at the leading edge \( (x/c = 0.1, \ y/c = 0.35) \).

Vorticity contours are shown in Figure 9; a regular shedding pattern is clearly visible. Note that the contour scale is limited to values of \( \pm 100 \); values as high as \(-9000 \) and \(+4000 \) occur in the wall boundary layer but the interest here is in the vortex shedding.

7.3 Adaptive \( k-\tau \)

The unsteady results for the adaptive \( k-\tau \) (AKT) model are presented here. This model yields considerably more detail than the Speziale \( k-\tau \) model as can be seen in the contour plots and in the lift and drag signals of Figure 10. The \( C_l \) and \( C_d \) signals are stochastic and no longer perfectly periodic. The period and amplitude of vortex shedding events is similar but not identical. An approximate period for vortex shedding can be taken as \( T \approx 0.1 \) seconds from Figure 10. This yields a Strouhal number of \( St = 0.55 \) which was also obtained from the Speziale \( k-\tau \) model. The time averaged lift and drag coefficients are \( C_l \approx 1.05 \) and \( C_d \approx 0.45 \).

Velocity stream lines for the AKT model are shown in Figure 11. The snapshot time for each plot is shown in the upper right hand corner and is listed in sec-
Fig. 5: Velocity Streamlines for Speziale $k - \tau$ Model at $\alpha = 20^\circ$ and $Re_c = 10^5$; Time is given in fractions of cycle Period $T$.

Fig. 6: $C_p$ Contours for Speziale $k - \tau$ Model at $\alpha = 20^\circ$ and $Re_c = 10^5$. 
Fig. 7: Eddy Viscosity Ratio $\frac{\mu}{\mu_t}$ Contours for Speziale $k - \tau$ Model at $\alpha = 20^\circ$ and $Re_c = 10^5$.

Fig. 8: Turbulent Kinetic Energy Ratio (local to free stream) Contours for Speziale $k - \tau$ Model at $\alpha = 20^\circ$ and $Re_c = 10^5$. 
Several recirculation bubbles of various diameters are observed; see the stream line plot at $t = 0.16$ and $t = 0.28$.

Figures 13 and 14 show the eddy viscosity ratio and turbulent kinetic energy ratio contours. The eddy viscosity contours are much more complex than those obtained with the KTS model. Note however that the magnitude of the eddy viscosity is much smaller with a maximum in the near wake of $\mu_t \approx 60$. The lower eddy viscosities, obtained with the AKT model, indicate a less dissipative model and therefore allow richer dynamic features. The turbulent kinetic energy peaks at the leading edge at the separation point $x/c = 0.5$ and along the trailing edge from $x/c = 0.8$ to $x/c = 1.0$. The turbulent kinetic energy ratio has a maximum value of three in these regions; this is an order of magnitude less than the results obtained with the KTS model. Figure 15 shows the vorticity contours. The magnitude of the vorticity in the wake is similar to the KTS model. The very high vorticities (order $10^3$) in the boundary layer are hidden by the contour scale limits. Again, the interest is in the wake vortex structures.

### 7.4 Comparison of Lift and Drag Signals

Lift ($C_l$) and drag ($C_d$) signals for the Speziale $k - \tau$ (KTS), adaptive $k - \tau$ (AKT) and Spalart Allmaras (SA) models at $Re_c = 10^5$ and $\alpha = 20^\circ$ are presented in Figure 16. As discussed earlier, the high dissipation associated with the SA model yields a steady state solution only. The AKT model gives the largest signal amplitude ($0.55 < C_l < 1.45$), ($0.3 < C_d < 0.6$) compared to the KTS model with ($0.8 < C_l < 0.9$), ($0.32 < C_d < 0.34$).

The period of the KTS model is approximately 0.1 seconds. The period for the AKT model has greater variation but is also approximately 0.1 seconds. The Strouhal number for the models is calculated based on the projected length of the airfoil normal to the free stream, the shedding period $T$ and the free stream velocity $U_\infty$.

$$St = \frac{c \sin \alpha}{TU_\infty}$$  (20)
Fig. 11: Velocity Streamlines for Adaptive $k - \tau$ Model at $\alpha = 20^\circ$ and $Re_c = 10^5$. 
Fig. 12: Pressure Contours for Adaptive $k - \tau$ Model at $\alpha = 20^\circ$ and $Re_c = 10^5$. 
Fig. 13: Eddy Viscosity Ratio $\frac{\mu_t}{\mu}$ Contours for Adaptive $k-\tau$ Model at $\alpha = 20^\circ$ and $Re_c = 10^5$. 
Fig. 14: Turbulent Kinetic Energy Ratio (local to free stream) Contours for Adaptive $k - \tau$ Model at $\alpha = 20^\circ$ and $Re_c = 10^5$. 
Fig.15: Vorticity Contours for Adaptive $k - \tau$ Model at $\alpha = 20^\circ$ and $Re_c = 10^5$. 
6.1 Comparison of Unsteady Lift Signals for Spalart Allmaras, Speziale $k-\tau$ and Adaptive $k-\tau$ models at $\alpha = 20^\circ$ and $Re_c = 10^5$.

This yields a Strouhal number of 0.19 for the KTS model and 0.17 for the AKT model. Experimentally, bluff bodies, including airfoils at high angles of attack, have Strouhal numbers of approximately 0.21; this is slightly higher than predicted here.

6.5 Comparison of Time Averaged Contour Plots

Plots of the time averaged streamlines, velocity, pressure, eddy viscosity, turbulent time scale and turbulent kinetic energy are presented. Averaging was done over 2000 time steps; this removes the effect of the starting and stopping points in the vortex shedding cycle. Time averaging was done only for the KTS and AKT models; the steady results of the SA model are presented directly. Turbulent time scale and turbulent kinetic energy ratio plots are only available for the KTS and AKT models; the SA model does not use these quantities.

Time averaged velocity streamlines are shown in Figure 18. The SA model yields a large clockwise rotating vortex between $x/c = 0.2$ and $x/c = 1.4$. There is also a secondary counter-rotating vortex below the main vortex at the trailing edge at $x/c = 0.9$ to $x/c = 1.4$. The KTS model also yields two vortices, one clockwise rotating between $x/c = 0.1$ and $x/c = 1.1$ and the other counterclockwise rotating at $x/c = 1.0$ to $x/c = 1.4$. These vortices do not overlap as in the SA model; the larger vortex does not extend past the trailing edge. Multiple vortex structures are obtained with the AKT model. There is a large clockwise rotating vortex at the trailing edge between $x/c = 0.6$ and $x/c = 1.3$ followed by a smaller counterclockwise rotating vortex between $x/c = 1.2$ and $x/c = 1.6$. There are several smaller vortices near the leading edge including a small counterclockwise vortex on the airfoil surface between $x/c = 0.2$ and $x/c = 0.3$. There is a clockwise rotating vortex between $x/c = 0.1$ and $x/c = 0.5$ located above the

![Fig.16: Comparison of Unsteady Lift Signals for Spalart Allmaras, Speziale $k-\tau$ and Adaptive $k-\tau$ models at $\alpha = 20^\circ$ and $Re_c = 10^5$.](image1)

![Fig.17: Comparison of Unsteady Drag Signals for Spalart Allmaras, Speziale $k-\tau$ and Adaptive $k-\tau$ models at $\alpha = 20^\circ$ and $Re_c = 10^5$.](image2)

![Fig.18: Comparison of Time Averaged Velocity Streamlines for Spalart Allmaras (SA), Speziale $k-\tau$ (KTS) and Adaptive $k-\tau$ (AKT) models at $\alpha = 20^\circ$ and $Re_c = 10^5$.](image3)
Fig. 19: Comparison of Time Averaged $U/U_\infty$ Velocity for Spalart Allmaras (SA), Speziale $k-\tau$ (KTS) and Adaptive $k-\tau$ (AKT) models at $\alpha = 20^\circ$ and $Re_c = 10^5$.

surface vortex, and another counterclockwise rotating vortex downstream of it from $x/c = 0.8$ to $x/c = 0.7$.

Averaged velocity contours in Figure 19 show that the SA and KTS models produce similar results. The AKT model predicts higher negative velocities in the trailing edge region ($U/U_\infty = -0.6$) from $x/c = 0.8$ to $x/c = 1.1$ compared to the KTS and SA models ($U/U_\infty = -0.2$). This is associated with the strong vortex at this location. There is a second smaller area of low velocity predicted by the AKT model at $x/c = 0.3$ to $x/c = 0.5$ that is missed by the SA and KTS models. A region of low velocity is associated with the centre of each of the major vortices.

Fig. 20: Comparison of Time Averaged Eddy Viscosity Ratio $\mu_t/\mu$ for Spalart Allmaras (SA), Speziale $k-\tau$ (KTS) and Adaptive $k-\tau$ (AKT) models at $\alpha = 20^\circ$ and $Re_c = 10^5$. Upper contour legend applies to AKT; lower contour legend applies to KTS and SA.

Fig. 20 shows eddy viscosity contours. The SA and KTS models use the same contour level scale; the AKT model has its own scale. Note that the AKT model yields eddy viscosities in the separated region and in the wake which are an order of magnitude lower (max of $\mu_t/\mu \approx 60$) than those of the KTS and SA models (max $\mu_t/\mu \approx 900$ and $\mu_t/\mu \approx 1000$ respectively). The lower eddy viscosities of the AKT model produce simulations akin to Large Eddy Simulations where the role of turbulence modelling is confined to the subgrid scale fluctuations. Indeed the AKT model produces richer dynamics and a broader range of vortices as illustrated in Figure 18. The KTS and SA models predict two vortex regions only. Since the AKT model resolves more of the turbulent structures directly, it is not necessary to account for their average effect through an increase in eddy viscosity.

The turbulent kinetic energy ratio plot for the AKT and KTS models is shown in Figure 21. The turbulent kinetic energy ratio is similar for both models at the leading edge in the adverse pressure gradient region ($x/c \approx 0.05$). From $x/c = 0.2$ and downstream in the wake, the turbulent kinetic energy is much higher for the KTS model (approximately 15). There is no appreciable difference between the free stream and wake turbulent kinetic energy in the AKT model for $x/c > 1.4$. The time average vorticity is shown in Figure 22; the KTS and SA models give similar results while the AKT model yields more complicated vortex structures in the separated region from $x/c = 0$ to $x/c = 1$. All models show negative vorticity (clockwise) originating from the leading edge and...
positive vorticity (counterclockwise) origination from the trailing edge.

8 Conclusions

Simulations of turbulent flow over a NACA 0012 airfoil at $Re_c = 10^5$ and $\alpha = 20^\circ$ have been conducted with an emphasis on assessing the performance of three turbulence models in unsteady simulations.

The Spalart-Allmaras model with the standard constants (tuned for steady flow) fails to yield unsteady flow. The Speziale $k - \tau$ model produces periodic lift and drag signals, with shedding of large counter rotating vortices at every cycle. Considerably richer flow structures are obtained with the adaptive $k - \tau$. This model, which tends asymptotically to DNS for high grid resolution and to RANS for unresolved fluctuations, yields non-periodic lift and drag signals and complex transient flow patterns with multiple separation bubbles. A key feature of this model is eddy viscosity levels in the separated region and the wake that are an order of magnitude lower than for the Spalart-Allmaras and $k - \tau$ models.

The predicted flow patterns and vorticity dynamics are particularly encouraging in view of the fact that massive flow separation is an inherently three dimensional phenomenon with significant spanwise interactions and structures. The next phase of this work will focus on investigating coupled flow-structure interactions involving separated flows.

REFERENCES


