LARGE EDDY SIMULATION OF COMPRESSIBLE FLOW AROUND A-AIRFOIL

Anne Cadiou, Christian Tenaud, and Loc Ta Phuoc

LIMSI-CNRS BP 133 F-19 403 Orsay Cedex, France
e-mail: cadiou@limsi.fr, web page: http://www.limsi.fr/

Key words: Compressible Navier-Stokes equations, large eddy simulation, boundary layer separation

Abstract. The prediction of aerodynamic coefficients is important for design. When an airfoil is at high incidence, for instance during landing and taking-off, unsteady and transitional mechanisms occur in the flow due to the separation of the boundary layers. The aim of this work is to study the ability of large-eddy simulation to predict the unsteady turbulent compressible flow around an airfoil near stall. The configuration retained is the A-airfoil with the angle of attack of $\alpha = 13^\circ 3$, a freestream Mach number of $M_\infty = 0.15$ and a chord Reynolds number of $Re_\infty = 2.1 \times 10^4$. 

1 INTRODUCTION

The prediction of aerodynamic coefficients is important for design. When an airfoil is at high incidence, for instance during landing and taking-off, unsteady and transitional mechanisms occur in the flow due to the separation of the boundary layers. The aim of this work is to study the ability of large-eddy simulation to predict the unsteady turbulent compressible flow around an airfoil near stall.

The configuration retained is the A-airfoil. Measurements performed at the ONERA show that for an angle of attack of $\alpha = 13^\circ 3$, a freestream Mach number of $M_\infty = 0.15$ and a chord Reynolds number of $Re_\infty = 2.1 \times 10^6$, the flow remains subsonic and three-dimensional effects can be considered as weak. The flow exhibits transition to turbulence followed by a trailing edge detachment on the suction side. At such a high Reynolds number, the correct prediction of near wall dynamics is still a challenge for large eddy simulations. Two approaches can be considered [2]. One consists in reducing the Reynolds number of the calculated flow, so that the near wall flow becomes resolved, and the other consists in imposing the wall shear stresses through the use of a wall law. To validate the present numerical approach, the first method has been investigated.

The solution obtained at the moderate Reynolds number of $Re_\infty = 2.1 \times 10^4$ is compared with the solution obtained at the same Reynolds number from the resolution of the incompressible Navier-Stokes equations, based on a velocity-vorticity formulation [6]. This incompressible solution will be the reference in this study. At $Re_\infty = 2.1 \times 10^6$ the incompressible solution agrees fairly well to the measurements, leading to a good prediction of the pressure and friction coefficient around the airfoil [6].

2 GOVERNING EQUATIONS

2.1 Filtered Navier-Stokes equations

In large eddy simulations, a flow quantity $\phi$ is decomposed into a large scale part $\overline{\phi}$ and a small scale part $\phi'' = \phi - \overline{\phi}$ using a spatial filter. This filter is supposed to commute with time and spatial derivatives. For compressible flow, a density-weighted filter, denoted as $\phi = \rho \overline{\phi}/\overline{\rho}$, is used in order to simplify the equations. The conservative form of the filtered Navier-Stokes equations can be expressed as

$$Q_{,t} + (F_{k}'' + F_{k}')_{,k} = 0$$

where $Q$ denotes the conservative flow variables, and $F_{k}''$ and $F_{k}'$ respectively the convective and viscous fluxes. They are defined as

$$Q = \begin{pmatrix} \overline{\rho} \\ \overline{\rho} \overline{u} \\ \overline{\rho} \overline{v} \\ \overline{\rho} \overline{w} \\ \overline{\rho} \overline{e} \end{pmatrix}, \quad F_{k}'' = \begin{pmatrix} \overline{\rho} \overline{u}_{k} \\ \overline{\rho} \overline{u}_{k} \overline{u} + \overline{\rho} \delta_{k1} \\ \overline{\rho} \overline{u}_{k} \overline{v} + \overline{\rho} \delta_{k2} \\ \overline{\rho} \overline{u}_{k} \overline{w} + \overline{\rho} \delta_{k3} \\ \overline{u}_{k} (\overline{\rho} \overline{e} + \overline{\rho}) \end{pmatrix}, \quad F_{k}' = \begin{pmatrix} 0 \\ \overline{\sigma}_{k1} + \overline{\tau}_{k1} \\ \overline{\sigma}_{k2} + \overline{\tau}_{k2} \\ \overline{\sigma}_{k3} + \overline{\tau}_{k3} \\ \overline{\sigma}_{n_k} (\overline{\tau}_{n_k} + \overline{\sigma}_{n_k}) + \overline{\sigma}_{k} + \overline{\sigma}_{k}' \end{pmatrix}$$
where $\tilde{\sigma}_{ij}$ is the filtered viscous tensor and $\tilde{q}_i^f$ the resolved heat flux. The subgrid stress tensor is defined by

$$\tilde{\tau}_{ij} = \rho \tilde{u}_i \tilde{u}_j - \overline{\rho \tilde{u}_i \tilde{u}_j} \equiv \rho \overline{u'_i u'_j}$$

(2)

and the subgrid heat flux $\tilde{q}_i^f$ by

$$\tilde{q}_i^f = c_p \left[ \rho \overline{u_i T} - \overline{\rho \tilde{u}_i T} \right] \equiv c_p \rho \overline{u'_i T'}$$

(3)

These are the two terms that need to be modeled.

### 2.2 Subgrid scale models

The subgrid stress tensor is expressed in gradients of the resolved scales via a Boussinesq hypothesis:

$$\tilde{\tau}_{ij} = \frac{1}{3} \tilde{\tau}_{kk} \delta_{ij} - \mu_t \left( \tilde{u}_{i,j} + \tilde{u}_{j,i} - \frac{2}{3} \tilde{u}_{k,k} \delta_{i,j} \right)$$

(4)

where $\mu_t$ is the subgrid viscosity and $\tilde{\tau}_{kk}$ represents the subgrid scale kinetic energy. The subgrid viscosity $[5] [12]$ is expressed as a weighted average of a Smagorinsky-like model $[10]$ and a model based on the energy at the cutoff scale, like $[8]$:

$$\mu_t = \mu_{t,\omega} \mu_{tg}^{1-\alpha}$$

(5)

where $\alpha$ is the weighting coefficient, taken as $\alpha = 0.5$. The Smagorinsky model, associated with the large scales, is based on the resolved vorticity $\tilde{\omega} = \nabla \times \tilde{u}$.

$$\mu_{t,\omega} = \overline{\rho} C_s^2 \Delta^2 |\tilde{\omega}|$$

(6)

where $C_s = 0.02$ is the Smagorinsky constant and $\Delta$ characterizes the cutoff length scale. The model based on the energy cutoff writes

$$\mu_{t,g} = \overline{\rho} C_o^2 \Delta \sqrt{q_c}$$

(7)

where $C_o = 0.126$ and $q_c$ is the energy cutoff. The latter is evaluated using a test filter with a cutoff length larger than $\Delta$:

$$q_c = \frac{1}{2} \langle \tilde{u}_i' \rangle' \langle \tilde{u}_i' \rangle'$$

(8)

The energy decreases in the vicinity of the wall so that no damping function is needed for this model. The model of the subgrid scale kinetic energy is also based on the kinetic energy at the cutoff $[1]$:

$$\tilde{\tau}_{kk} = 2 \overline{\rho} q_c$$

(9)

The subgrid scale heat flux is closed using a gradient approach

$$\tilde{q}_i^f = -k_t \tilde{T}_{i,i}$$

(10)
where \( k_t \) is the subgrid scale thermal conductivity, estimated with a constant turbulent Prandtl number hypothesis \( P_{rt} = 0.6 \).

\[
    k_t = \frac{\mu_t c_p}{P_{rt}}
\]  

(11)

To take into account the anisotropy of the filter, the correction proposed by Scotti and Meneveau [9] for the estimation of the cutoff length \( \Delta \) is used.

3 NUMERICAL METHOD

3.1 General description

The compressible Navier-Stokes equations are discretized using a cell-centered finite volume technique. The convective terms are discretized by a fifth order shock capturing WENO scheme [3]. Viscous and thermal fluxes are treated with a second order centered scheme. Time integration is performed explicitly by a third order Runge-Kutta method. The time step is limited by the stability of the explicit Runge-Kutta scheme and is therefore calculated from a CFL \((< 1)\) and diffusion number \((< 0.5)\) condition.

3.2 Boundary conditions

The wall is assumed to be adiabatic and a no-slip boundary condition is imposed. Periodic conditions are applied in the spanwise direction. At the external boundary, non reflecting conditions are imposed through the characteristic method.

3.3 Flow initialization

Two kinds of initialization have been applied, one obtained from an incompressible solution, the other generated by a uniform field with compressible boundary layers on the wall [11]. The resolution of the incompressible Navier-Stokes equations is done using the velocity-vorticity formulation. The convective terms are discretized on a staggered grid by a second order Quick scheme whereas the viscous terms are treated with a second order centered scheme. Time integration is performed implicitly with a second order backward Euler scheme. The equations are solved within conformal transformation [4].

3.4 Mesh

An O-mesh of 257 \( \times 513 \times 3 \) is generated by conform transformation in order to preserve orthogonality of the grid. The computational domain extends to 10 chord lengths upstream and downstream. The wall normal resolution reaches \( \Delta n^+ \sim 2 \) on the pressure side and \( \Delta n^+ \sim 4 \) on the suction side. The repartition of the 257 points around the profile leads to a streamwise resolution of \( \Delta t^+ \sim 10 \). The periodic conditions applied in the spanwise direction allow a crude resolution in this direction so that the fictive plans are located at \( \Delta s^+ \sim 100 \). The mesh resolution usually required in large eddy simulation [7] of plane channel flow \((\Delta n^+ \sim 2 \text{ and } \Delta t^+ \sim 100)\) in bidimensional calculation, with
\( \Delta s^+ \sim 20 \) in threedimensional calculation) is therefore almost reached in the bidimensional configuration retained here.

4 NUMERICAL RESULTS

4.1 Available measurements

Measurements performed at ONERA for \( Re_\infty = 2.1 \times 10^6 \) provide pressure and friction coefficients on the profile. In addition, time averaged velocity profiles and Reynolds tensor components are available, measured in the direction normal to the airfoil wall (see figure 1) and in the direction normal to the incident velocity (see figure 2). The experiments cover the detachment region on the suction side and the wake flow.

![Figure 1: Measures normals to the airfoil](image)

![Figure 2: Measures normals to the incident velocity](image)

4.2 Time evolution

A convective time based on the chord length and the velocity far upstream is introduced, \( t = t^* c / V_\infty \). When the flow is initialized using the incompressible solution, the calculation exhibits a transient phase during which the pressure field adapts to the compressible conditions. Pressure waves are generated at the trailing edge and at the leading edge, propagating towards the outer domain. These waves interfere and eventually propa-

![Figure 3: Instantaneous pressure contours for the incompressible initialization \( t^* = 2.5 \)](image)

![Figure 4: Instantaneous pressure contours for the compressible initialization \( t^* = 2.5 \)](image)
Oscillating behaviour, generated by the initialization, is observed on the instantaneous pressure field at $t^* = 2.5$ in figure 3. Results for the uniform field initialization are shown in figure 4. The presence of those perturbations delayed the time where compressible statistics can be constructed and might lead to instability. The initialization with the uniform field has therefore been preferred.

### 4.3 Instantaneous field

The evolution of the solution obtained from the compressible calculation is presented below by the contours of instantaneous vorticity field. The initial flow field is uniform except around the profile where a compressible boundary layer connects the outer flow with the no-slip condition at the wall.

![Instantaneous vorticity fields](image)

$t^* = 0.5$  
$t^* = 1.0$  
$t^* = 1.5$

The impulsive start of the flow creates a vortex at the trailing edge, detaching from the profile. After a time of $t^* = 1$, the vortex is convected downstream over a distance of about one chord length. The wake is formed. At the same time, the boundary layer thickness on the suction side grows, generating vortices near the leading edge. Those small vortices propagates downstream along the wall.

![Instantaneous vorticity fields](image)

$t^* = 2.0$  
$t^* = 2.5$  
$t^* = 3.0$

About $t^* = 2.0$ the wake starts to oscillates as a result of the instabilities triggered by the vortices generated on the suction side wall. A small mixing layer develops at
the trailing edge. This phenomenon is also observed in experimental visualization. The vortices created at the leading edge grow and detach, followed by other vortices.

\[ t^* = 4.0 \quad t^* = 4.5 \quad t^* = 5.0 \]

While propagating downstream, those vortices become larger. They are involving strong pressure gradient which are pumping flow from the pressure side with a reversal speed. This vortice interact back with the pressure side vortice, a part of which is ejected in the wake and an other part is forced to agglomerate with the next pressure side swirl, leading to an solely swirl more intense. This phenomena can also be observed in the incompressible calculation, in a less intense way however. The depression created by the swirl coming from the leading edge at the trailing edge also attracts fluid from the pressure side, but the backward flow does not reach one third of the chord like here so that the primary vortex is simply ejected from the profile, rolling up with the trailing edge vortex.

\[ t^* = 5.5 \quad t^* = 6.0 \quad t^* = 6.5 \]

The second swirl is ejected away from the wall and, when it reaches the trailing edge, interferes again with vortex coming from the pressure side.
$t^* = 7.0$  
$t^* = 7.5$  
$t^* = 8.0$

This phenomena repeats itself but the evacuation of the trailing edge vortex is more simple. The evolution of the incompressible flow is quite similar to the behaviour of the compressible solution. The influence of pressure gradient in the compressible calculation seems however more pronounced than in the incompressible calculations.

### 4.4 Statistics

As observed with the time evolution of the vorticity field, the flow evolves towards a very unsteady production of vortices on the suction side of the profile. Up to a convective time about $t^* = 2.5$, the suction side boundary layer grows, while the pressure distribution around the profile reaches its maximum on the suction side. The pressure coefficients obtained from the compressible and the incompressible calculations are plotted in figure 5. The peak is about two times smaller for the moderate Reynolds number flow of $Re_\infty = 2.1 \times 10^4$ than for the experimental Reynolds number flow of $Re_\infty = 2.1 \times 10^6$.

![Figure 5: Time-averaged pressure coefficient between $t^* = 0$ and $t^* = 2.5$](image)

![Figure 6: Time-averaged friction coefficient between $t^* = 0$ and $t^* = 2.5$](image)

Results obtained using the incompressible and the compressible initialization are very similar. Both calculations predict a detachment of the suction side boundary layer at about 30% of the chord at $Re_\infty = 2.1 \times 10^4$. At $Re_\infty = 2.1 \times 10^6$, the flow exhibits a laminar detachment at this location, followed by a transition to turbulence. The suction side boundary layer detaches then near the trailing edge.
The boundary layer development at $Re_\infty = 2.1 \times 10^4$ can be compared to the one at $Re_\infty = 2.1 \times 10^6$ before $t^* = 2.5$. Time-averaged streamwise velocity profiles in the direction normal to the wall, obtained using incompressible and compressible initialization, are shown in the following figures. These are time-averaged quantities between convective times of $t^* = 0$ and $t^* = 2.5$ for both calculations.

Figure 7: Streamwise velocity in the direction normal to the wall $\bar{u}/U_\infty$

Results obtained using the two initializations are quite similar at $t^* = 2.5$. As a consequence of the pressure oscillations produced after the initialization done using the
incompressible solution, oscillations can be observed in boundary layer profile obtained using the incompressible initialization. The convergence of the statistics of the spanwise velocity component is slower than the convergence of the streamwise velocity component. The next figures show the time-averaged spanwise velocity profile in the wake obtained using both initialization.

![Graphs showing incompressible and compressible calculations](image)

**Figure 8:** Spanwise velocity component normal to the wake $\tilde{v}/U_{\infty}$

The formation of the wake is observed. The position of the accumulated wake is shifted with respect to the experimental profiles. This might be explained by misinterpretation of the reference point of the experimental results. Results obtained from the incompressible and compressible (using uniform initialization) large eddy simulations are very similar at $t^* = 2.5$. Both predict a suction side boundary layer detachment at 30% of the chord. After $t^* = 2.5$, both simulations predict massive flow detachment and unsteady vortex production on the suction side.

Time evolution of lift and drag coefficient obtained with the compressible calculation is shown figures 9 and 10.
The initialization has a great influence on the evolution of these coefficients up to a convective time about $t^* = 2.5$. Time-averaged statistics of incompressible and compressible calculations are compared over the similar period, between $t^* = 2.5$ and $t^* = 5.0$, corresponding to a vortex production cycle. Time-averaged pressure and friction coefficients are compared on the next pictures.

The pressure distribution differs on the suction side near the trailing edge. It can be seen on figure 11 that the pressure difference between the suction and pressure side is larger in the compressible calculation. This is consistent with the vortex ejection mechanism observed in the instantaneous vorticity evolution. The friction coefficient distribution is similar in both calculations. It is observed that the vorticity layer next to the wall develops from about 20% of the chord in both calculations.

Time-averaged streamwise velocity profiles in the boundary layer are plotted in the next figures. Results for the incompressible and compressible calculations exhibit the same tendency. Since $t^* = 2.5$, the boundary layer on the suction side has thickened.
Results of the normal velocity component in the direction normal to the wake are also quite similar in both calculations. The position of the wake is the same as well as its development downstream. Because of the massive detachment occurring at the suction side of the profile, the mean flow on the upper part of the wake has slowed down compared to the evolution before $t^* = 2.5$. 
Comparison of the time-averaged velocity correlations on the profile are shown in the next figures. The rms velocity fluctuations in streamwise and spanwise direction gives an indication about the turbulent intensity in those directions. Incompressible and compressible calculations predict about the same level of turbulence in streamwise and spanwise direction. It is observed that the streamwise correlations near the leading edge are larger than the spanwise correlations. This difference is more pronounced in the time-averaged correlations of the compressible calculation.
Figure 15: Time-averaged streamwise velocity fluctuations in the direction normal to the wall $\sqrt{\overline{u'v'}/U_{\infty}}$
Figure 16: Time-averaged spanwise velocity fluctuations in the direction normal to the wall $\sqrt{u'v'}/U_\infty$. 
Larger differences are be observed for the shear stress components, as predicted by the incompressible and the compressible simulations in the boundary layer.

Figure 17: Time-averaged shear stress component to the wall $\overline{uw}/U_\infty^2$
5 CONCLUSIONS

Large eddy simulations of a turbulent flow around an airfoil near stall has been performed for moderate Reynolds number $Re_{\infty} = 2.1 \times 10^4$ with an infinite Mach number of $M_{\infty} = 0.15$. Two calculations have been performed, one consists in the resolution of the incompressible Navier-Stokes equations based on a velocity-vorticity formulation, the other being based on the resolution of the compressible Navier-Stokes equations. The numerical implementation of these methods are very different, yet they predict a flow development rather similar. Two initialization have been tested for the compressible calculation. One is taken from an incompressible calculation, the other is a uniform field satisfying the no-slip boundary condition at the wall. The result obtained with the physical incompressible initialization show the development of pressure waves propagating in the domain. These oscillations express the adjustment of the incompressible variables to the compressible conditions and they persist during a rather long time period compared to the development of the flow. Despite the fact that they exit the domain and are progressively damped, they affect the entire flow domain and lead to oscillations of the variables during a long time. The initialization with a uniform flow does not suffer from those oscillation, allowing to a faster calculation of the flow statistics, and is therefore preferred. After $t^* = 2.5$ the boundary layer detachment produce an unsteady detachment over the entire suction side. The vorticity production process obtained with the incompressible calculation and the compressible calculation are quite similar. Statistical results obtained from one vortex production detachment cycle are presented. Both calculations exhibit a massive detachment from the leading edge. The flow at $Re_{\infty} = 2.1 \times 10^4$ is entirely turbulent, in opposite to the flow measured at $Re_{\infty} = 2.1 \times 10^6$ where a transition occurs at 30% of the chord. Accordingly, our simulations show large streamwise velocity fluctuations. It would be interesting to compare these results with three-dimensional calculations to validate the bidimensional hypothesis made in this study.
REFERENCES


