International Journal of Heat and Fluid Flow 000 (2016) 1–12

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Direct numerical simulation of the flow around a wing section at moderate Reynolds number

S.M. Hosseini[a], R. Vinuesa[a], P. Schlatter[a,*], A. Hanifi[a,b], D.S. Henningson[a]

[a] KTH Royal Institute of Technology, Department of Mechanics, Linné FLOW Centre, Swedish e-Science Research Centre (SeRC), SE-100 44 Stockholm, Sweden
[b] Swedish Defence Research Agency, FOI, SE-164 90 Stockholm, Sweden

A R T I C L E I N F O

Article history:
Available online xxx

Keywords:
Turbulent boundary layer
Vortex shedding
Wake
Incipient separation
Pressure gradient
NACA4412

A B S T R A C T

A three-dimensional direct numerical simulation has been performed to study the turbulent flow around the asymmetric NACA4412 wing section at a moderate chord Reynolds number of $Re_c = 400,000$, with an angle of attack of $AoA = 5^\circ$. The mesh was optimized to properly resolve all relevant scales in the flow, and comprises around 3.2 billion grid points. The incompressible spectral-element Navier–Stokes solver Nek5000 was used to carry out the simulation. An unsteady volume force is used to trip the flow to turbulence on both sides of the wing at 10% of the chord. Full turbulence statistics are computed in addition to collection of time history data in selected regions. The Reynolds numbers on the suction side reach $Re_\tau \simeq 373$ and $Re_\theta = 2,800$ with the pressure-gradient parameter ranging from $\beta \approx 0.0$ to $\beta \approx 85$. Similarly, on the pressure side, the Reynolds numbers reach $Re_\tau \approx 346$ and $Re_\theta = 818$ while β changes from $\beta \approx 0.0$ to $\beta \approx -0.25$. The effect of adverse pressure gradients on the mean flow is consistent with previous observations, namely a steeper incipient log law, a more prominent wake region and a lower friction. The turbulence kinetic energy profiles show a progressively larger inner peak for increasing pressure gradient, as well as the emergence and development of an outer peak with stronger APGs. The present simulation shows the potential of high-order (spectral) methods in simulating complex external flows at moderately high Reynolds numbers.

1. Introduction

A clear evolutionary path can be observed looking back at the aircraft from early days and comparing them to the state of the art modern civil aircraft. This is owed to our deeper knowledge of different flow phenomena around such bodies, *i.e., laminar-turbulent transition, wall-bounded turbulence under pressure gradients, flow separation, turbulent wake flow*, etc. Despite the conspicuous advances, there still remain major challenges in terms of understanding the complex flow phenomena and a design procedure that can efficiently exploit the interacting features on an airplane. Traditionally such procedures relied heavily on experimental findings. Recent advances in supercomputers and massive parallelization have enabled designers to determine more and more parameters in aircraft design via computational approaches in the early stages of the design, effectively minimizing costs. Recently NASA issued a report by Slotnick et al. (2014) laying out a number of findings and recommendations regarding the role of computational fluid dynamics (CFD) in aircraft design. The main revelations point out the necessity of accurate prediction of turbulent flows with significantly separated flow regions, more robust, fast and reliable mesh generation tools and development of more multidisciplinary simulations for both analysis and design optimization procedures. Johnson et al. (2005) also discuss the changing role of CFD from a mere curiosity to a significant role in efficient designs. For instance, in a *Boeing–777* significant part of high-speed wing design and propulsion/airframe has been done via CFD simulations.

Despite numerous efforts in keeping the flow laminar on aerodynamic surfaces, turbulence flow dominates over a large portion of modern aircraft. In particular pressure gradients (PG) on different surfaces play a major role in the development of turbulent boundary layers (TBL). Some of the early numerical studies of PG TBLs on flat plates using direct numerical simulations (DNS) were the work by Spalart and Watmuff (1993), and Skote et al. (1998), which focused on the role of adverse pressure gradients (APG). This problem has attracted growing attention over the last decade in the recent works by Lee and Sung (2008), Gungor et al. (2014) and Bobke et al. (2016) . Favorable pressure gradients (FPG) have also been recently studied numerically by Piomelli and Yuan (2013).

One of the first structure-resolving numerical studies of the flow around a wing was the large-eddy simulation (LES) performed

* Corresponding author. Tel.: +46 87907176.
E-mail address: pschlatt@mech.kth.se (P. Schlatter).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2016.02.001

by Jansen (1996), who considered a NACA4412 profile at a Reynolds number based on freestream velocity U_∞ and chord length c of $Re_c = 1.64 \times 10^6$. The idea behind that study was to compare his results with three different available experimental campaigns of the same flow configuration; those experiments were carried out by Coles and Wadcock (1979), Hastings and Williams (1987) and Wadcock (1987). The flow around the symmetric NACA0012 wing profile was studied by means of DNS by Shan et al. (2005), who considered a Reynolds number of $Re_c = 100,000$, and an angle of attack of $AoA = 4^\circ$. They found that the backward effect of the disturbed flow on the separated region may be connected to the self-sustained turbulent flow and the self-excited vortex shedding on the suction side of the wing. In addition to this, they found that the vortex shedding from the separated free shear layer was due to a Kelvin–Helmholtz instability. At lower Reynolds number, $Re_c = 50,000$, and larger angles of attack of 9.25° and 12°, the same case was analyzed through DNS by Rodríguez et al. (2013). They used a second-order conservative scheme, and found that a combination of leading-edge and trailing-edge stall caused the massive separation on the suction side of the wing. Besides, they also reported that the vortices formed after the initial shear layer undergoes transition are shed forming a von Kármán vortex street in the wake. It should be put in perspective that this range of Reynolds number is significantly smaller than a conventional passenger plane cruising at $Re \approx 10^8$. Note that the resolution requirements increase with $Re^{37/14}$ (*cf.* Choi and Moin, 2012). For instance, increasing the Reynolds number with a factor of 10^3 dictates a resolution of approximately 80 million times higher. Therefore despite the invaluable information gained from performing well-resolved DNSs at higher Reynolds numbers, they are accompanied by numerous challenges, such as mesh generation, computational resources, and data handling.

Here we investigate the turbulent flow around the asymmetric NACA4412 airfoil, with a chord Reynolds number of $Re_c = 400,000$ and $AoA = 5^\circ$. The airfoil geometry includes a sharp trailing edge, obtained by modifying the last coefficient in the NACA airfoil equation to −0.1036 (see for instance Abbot and von Doenhoff, 1959). This is a big leap in Reynolds number compared to similar previous studies. The Reynolds number corresponds to that of a small cruising glider. Moreover, the use of high-order methods (as opposed to other numerical studies) ensures high-fidelity simulations, especially at the moderate Re under consideration where a significant scale separation starts to emerge. This paper describes the different steps required in performing such large-scale simulations. In addition to full turbulence statistics, time history data is also collected to monitor the different transient behavior of turbulent structures in different regions of the flow.

2. Direct numerical simulation

2.1. Nek5000

Direct numerical simulations were performed using the incompressible Navier–Stokes solver 'Nek5000' by Fischer et al. (2008), which uses the spectral-element method proposed (SEM) by Patera (1984). Enabling the geometrical flexibility characteristic of finite-element methods combined with the accuracy provided by spectral methods are the main advantages of using SEM. The equations are cast into weak form. The spatial discretization is then done by means of the Galerkin approximation, following the $\mathbb{P}_N - \mathbb{P}_{N-2}$ formulation. The solution to the Navier–Stokes equations is approximated element-wise as a weighted sum of Lagrange interpolants defined by an orthogonal basis of Legendre polynomials up to degree N (of degree $N-2$ in the case of the pressure). Inside each element, Gauss–Lobatto–Legendre (GLL) quadrature points are used, and extended to 3D in a tensor-product formulation. The geometry is mapped isoparametrically from the reference element to the actual grid. The nonlinear terms are treated explicitly by third-order extrapolation (EXT3), whereas the viscous terms are treated implicitly by a third-order backward differentiation scheme (BDF3). In order to avoid quadrature errors, the nonlinear terms are oversampled by a factor of 3/2 in each direction. Furthermore, a low amplitude filter of the highest expansion coefficient ($Amp_{filter} = 1\%$) is used in order to avoid errors from potential geometrical aliasing (*cf.* Fischer and Mullen, 2001). Nek5000 has previously been used to study turbulent flow in a pipe by El Khoury et al. (2013) and flow around a wall-mounted square cylinder by Vinuesa et al. (2015) at moderately high Reynolds-numbers. The following results have been obtained using comparably high order, $N = 11$. The present SEM code is optimized for modern computers with thousands of processors Tufo and Fischer (2001). Here, we have performed parallel computations on 16,384 processors. Scaling tests performed on the Cray-XC40 computer *Beskow* at PDC (KTH) are shown in Fig. 1, for the case under consideration in the present study (left), which has a total of 3.2 billion grid points (and around 1.85 million spectral elements), and for a similar configuration with a total of 120 million grid points (right). In order to establish a good measure of scaling, we report the time required to perform one GMRES (generalized minimal residual method) iteration for the pressure solve. Doing so, we can characterize code performance by isolating it from other factors contributing to the total time per time-step, such as I/O operations, different tolerances etc. Note that minor superlinear scaling is observed in Fig. 1 (left) up to 32,768 cores. This can be explained by the fact that Nek5000 exhibits best parallel performance when the number of cores is a power of 2, which was not the case for the simulation with the lowest number of processes (12,000 cores) due to the required memory. Interestingly, the scaling tests shown in Fig. 1 (right), which were performed on numbers of cores which were powers of 2, exhibit linear scaling up to 8192 cores. It is also interesting to note that in the case under consideration in this study the best parallel efficiency is achieved when running on 32,768 cores (with around 100,000 grid points per core), and in the smaller test case the best performance is observed on 4,096 cores, with a total of 30,000 grid points per core. A total of 35 million core-hours were spent to collect full turbulence statistics, time history data, and flow snapshots for visualization purposes. The flow was initially run for three flow-over times with $N = 5$, and after this point the polynomial order was progressively increased up to the final value of $N = 11$.

2.2. Boundary conditions, mesh design and simulation procedure

Initially a RANS simulation was performed by means of EDGE code developed at FOI, the Swedish Defence Research Agency (Eliasson, 2002). The code uses the explicit algebraic Reynolds stress model (EARSM) by Wallin and Johansson (2000). The RANS domain extends up to 200c in every direction. The solution from the RANS simulation is used to extract an accurate velocity distribution in the near field corresponding to the averaged flow at a given angle of attack. Consequently these values are imposed as Dirichlet boundary conditions on the DNS domain. Note that the natural stress-free condition was used at the outlet, and periodicity was imposed in the spanwise direction. We considered a C-mesh topology of radius c centered at the leading edge of the airfoil, with total domain lengths of 6.2c in the horizontal (x), 2c in the vertical (y) and 0.1c in the spanwise (z) directions, see Fig. 2. The resulting spectral-element mesh (without the GLL points) and the computational domain are shown in Fig. 3. In this figure it can be observed that a local smoothing was applied to a very small region close to the sharp trailing edge.

Please cite this article as: S.M. Hosseini et al., Direct numerical simulation of the flow around a wing section at the moderate Reynolds number, International Journal of Heat and Fluid Flow (2016), http://dx.doi.org/10.1016/j.ijheatfluidflow.2016.02.001