Compressible-Flow DNS with Application to Aeroacoustics

R.D. Sandberg

Aerodynamics and Flight Mechanics Research Group, University of Southampton

sandberg@soton.ac.uk

1 Introduction

Dramatic advances in available computing power over the last decades have enabled the use of high-fidelity flow simulation approaches for investigation of aerodynamically generated noise. The so-called computational aeroacoustics (CAA) approaches can broadly be classified into direct or hybrid approaches. In the latter case, flow simulations are conducted as a first step to provide the noise source field that serves as input for various types of acoustic analogies, used to propagate sound to the acoustic far field. In hybrid approaches, either compressible or, should the flow parameters allow, incompressible simulations of the hydrodynamically relevant region can be conducted. In contrast, in direct noise computations (DNC), compressible flow solvers are used to simulate the hydrodynamic and acoustic fields simultaneously. Moin & Mahesh (1998) described such computational aeroacoustics simulations as still being in their infancy, and highlighted that “the sound from a canonical flow such as a perfectly expanded supersonic jet has only just been computed…”.

However, over the last decade and a half, significant progress has been made applying DNS in the context of computational aeroacoustics, and the technique is becoming more popular as the computing power increases. Freund (2001) used DNS to investigate the noise sources in a low-Reynolds-number turbulent jet at Mach 0.9, and the first simulations of wall-bounded flows followed, with Hu et al. (2003) considering sound radiation in turbulent channels. A few years later, Sandberg & Sandham (2008) simulated turbulent flow over a trailing edge and the associated noise generation in order to investigate the accuracy of classical trailing edge theories, and direct noise simulations of full airfoil configurations have followed.

In the current contribution, current compressible direct numerical simulations (DNS) will be presented with application to aeroacoustics. The challenges in conducting such simulations, in terms of both computational effort and boundary conditions, will be discussed. A high-fidelity multi-block structured curvilinear compressible Navier-Stokes solver purposely developed for exploiting high-performance computing systems (HPC) will be introduced. The code has been designed with maximum efficiency in mind to allow cutting-edge high-fidelity aeroacoustics simulations on HPC systems. The performance of the code on HPC architectures will be presented. Results will be shown for i) turbulent flow over trailing edges, testing acoustic theories, ii) a canonical turbulent pipe-jet configuration, investigating the significance of the various noise sources present; iii) noise reduction strategies for airfoils.

2 Numerical Challenges

Directly solving the unsteady compressible Navier–Stokes equations to study aerodynamically generated noise introduces several computational challenges. Some of the challenges were pointed out by Crighton (1986), who stated that the extent of the acoustic field is considerably larger than the relevant flow field due to the much greater wave length of acoustic waves when compared to flow structures. This implies that large domains are needed to capture the acoustic field, while a fine resolution in the source region is required to resolve the noise-generating turbulence. The computational effort as a function of Reynolds and Mach numbers can be estimated as follows. The number of grid points $N^1$ required in one spatial direction to resolve the smallest scales of turbulence motion, the Kolmogorov scale $\eta_K$, over the length of the largest scales $\ell$, associated with the geometry of the flow, scales as $N^{1D} = \ell/\eta_K \propto Re_t^{3/4}$, with $Re_t$ denoting the turbulent Reynolds number and assuming isotropy of the turbulence.

Because turbulence is intrinsically a three-dimensional phenomenon, the number of grid points $N^{3D}$ that are required to resolve the turbulence flow field is thus proportional to $Re_t^{3/4}$. Including the acoustic wavelength in the scaling results in $N^{3D}_a \propto Re_t^{3/4}/M^3$. If one finally factors in the change in timestep required to resolve the Kolmogorov timescale for increasing Reynolds number while also running for sufficiently long times to capture the acoustic frequencies, the total computational effort is proportional to $Re_t^{3/4}/M^4$. While the adverse Reynolds number scaling can be somewhat remedied by introducing turbulence modelling, e.g. to $Re_t^2$ in large-eddy simulation, the high power of the Mach number in the denominator

1 Aeronautics and Flight Mechanics Research Group, University of Southampton

sandberg@soton.ac.uk
constitutes a severe constraint when considering small Mach numbers. In such situations, direct noise computations become computationally prohibitively expensive and one typically resorts to hybrid approaches using incompressible flow solvers. However, in cases where the acoustic field affects the flow field, e.g. in the presence of acoustic feedback mechanisms, hybrid approaches cannot be used due to their one-way fluid-to-acoustics coupling and very high computational expense is unavoidable.

The energy of the acoustic pressure fluctuations being smaller than that of the hydrodynamic pressure fluctuations by \( O(M^4) \) constitutes another challenge for aeroacoustic simulations (Crighton, 1986). Firstly, very accurate numerical schemes are needed with minimal levels of dissipation. In addition, suitable free-space boundary conditions are required to avoid spurious reflections that can contaminate the simulation. For general reviews on the topic of computational aeroacoustics and the associated issues, see Tam (1995); Lele (1997), or, more recent, Colonius & Lele (2004); Wang et al. (2006).

3 Numerical Method

To meet the numerical challenges outlined in the previous section, direct noise simulations require numerical codes that can exploit modern high-performance computing (HPC) systems. One such code is the in-house multi-block structured curvilinear compressible Navier-Stokes solver HiPSTAR (High Performance Solver for Turbulence and Aeroacoustics Research). To minimize computation time for a given problem in order to make best use of the available resources, the numerical algorithm was designed to satisfy several stringent requirements. These include resolution of flow features with minimal amplitude and phase errors, efficiency of the scheme (i.e. high ratio of accuracy to computational cost) and high parallel efficiency on HPC systems. One of the most effective ways to increase the performance of a numerical code on bandwidth-limited systems is to reduce the memory requirement of a simulation for a given problem. In HiPSTAR, this is achieved by a) using a novel parallel wavenumber-optimized compact finite difference scheme (Kim & Sandberg, 2012) in conjunction with a spectral method (using the FFTW3 library) for the discretization of the span-wise direction to reduce the grid size needed for a given problem, b) employing a state-of-the-art Runge-Kutta scheme (Kennedy et al., 2000) that achieves fourth-order accuracy with only two registers of memory, and c) restricting the coordinate mapping to two-dimensions, resulting in a smaller number of metric terms that are two-dimensional only. When dealing with complex geometries where the metric terms at intersecting block boundaries become discontinuous characteristic interface conditions (CIC) are used for the coupling of the blocks. To increase the robustness of the numerical scheme for the DNS and to avoid aliasing errors of the cubically nonlinear convective terms within the discretized Navier-Stokes equations, a conditioning of the compressible Navier-Stokes equations was implemented (Kennedy & Gruber, 2008). This method employs skew-symmetric splitting of the non-linear terms, but, unlike previous methods, quadratic and cubic nonlinearities are split differently. To avoid spurious reflections from boundaries that are subject to passage of flow structures, a zonal characteristic boundary condition has been implemented (Sandberg & Sandham, 2006). To take full advantage of massively parallel HPC systems, the code was initially parallelized using MPI and has recently been extended to hybrid OMP/MPI parallelization through a CRAY Centre of Excellence project. In this project, HiPSTAR was adapted to use shared memory parallelism using the OpenMP API. The code has been thoroughly validated comparing data with, e.g., linear stability results and turbulent pipe flow (Wu & Moin, 2008). DNS of turbulent axisymmetric wakes have recently been conducted using the new code (Sandberg, 2012).

4 Parallel Performance

The performance of the DNS code has been evaluated on several different computing architectures, but for conciseness here only results obtained on the UK national supercomputing facilities HECToR, a CRAY XE6 architecture, and ARCHER, a CRAY XC30 system, are presented. Initial scaling tests were performed on ARCHER on a single-block test problem with 2,404 points in the streamwise and lateral directions, and 128 Fourier modes in the spanwise direction, resulting in a total number of collocation points of \( 1.3 \times 10^9 \). In a strong scaling test, i.e. core numbers are varied for a given problem size, the smallest number of cores that could be used was 288, which implies that at least \( 4.5 \times 10^9 \) grid points can be allocated per core. Figure 1 shows the excellent scaling of the code up to 36,864 cores, for which a real speed-up factor over 288 cores of 102 is achieved versus the ideal factor of 128. Weak scaling tests were also performed on the CRAY XE6 HECToR, i.e. the number of operations every core has to perform and the MPI messages every core has to send/receive is kept identical for increasing overall number of cores. When using 32\(^3\) points per core, very good scaling is obtained for all core numbers tested, with efficiency not dropping below 90%. The larger test case with 64\(^3\) points per core shows even better scaling, with efficiency remaining as high as 96% up to 65,536 cores (resulting in a case with \( 17.2 \times 10^9 \) grid points).

The scaling study with 64\(^3\) grid points/core was repeated with the parallel compact finite difference scheme. For moderate core counts, the efficiency drops slightly more than when using the standard difference scheme, e.g. 95% at 4,096 cores. However, at 65,536 cores, the efficiency increases back
to 96%. Using 65,536 cores, the time spent per full Runge-Kutta cycle, i.e. five evaluations of the right-hand-side of the governing equations, is 4.87s/step and 5.49s/step for the standard and compact finite difference schemes, respectively. This 13% increase in computational cost of the parallel compact scheme is more than acceptable considering that due to the significantly better wavenumber resolution characteristics of the compact scheme a coarser grid could be used for a given problem. The moderate increase in overall computational time despite the significantly higher algorithmic cost of the compact finite difference schemes can be explained by the fact that codes with better ratios of algorithmic operations (FLOPs) over communication fare better on current bandwidth-limited computing systems. In addition to the excellent parallel scaling of the code, the performance of the underlying algorithm also appears to be very good. On the CRAY XE6, a sustained performance of 1 GFLOP/s or 12.5% of peak performance in HiPSTAR’s core computation kernels was measured.

5 Results

Trailing-edge Noise

Direct numerical simulations (DNS) were conducted of turbulent flow passing an infinitely thin trailing edge (TE). The turbulent boundary layer on one side of the splitter plate reaches $Re_{d_{TE}} = 2,222$ at the trailing edge, whilst the boundary layer on the other side of the plate is laminar. The objective was to investigate the turbulent flow field in the vicinity of the TE and the associated broadband noise generation. The simulation was conducted using $10^6$ gridpoints. Full details about the grid and numerical set up are given in (Sandberg & Sandham, 2008). For a qualitative view of the flow field, instantaneous contours of dilatation are shown in figure 2.

The acoustic pressure obtained from the DNS were compared with predictions from two- and three-dimensional acoustic analogies and the classical trailing edge theory of Amiet (Amiet, 1976b). It is found that for low frequencies the single frequency two dimensional acoustic analogy, as derived in Sandberg & Sandham (2008), produces reasonable agreement with both DNS and the computationally more expensive three dimensional acoustic analogy. The reason for this is that the surface pressure is strongly correlated in the spanwise direction and the sound radiation is predominantly two dimensional. Once the spanwise coherence decreases decreases to smaller values, the sound radiation becomes three dimensional, necessitating the use of a full three dimensional theory.

The DNS data were also used to test some of the key assumptions invoked for the derivation of a classical trailing edge theory (Amiet, 1976b). Amiet’s theory assumes that the far field sound can be evaluated as a function of the total surface pressure difference, $\Delta p_t$. An accurate prediction of $\Delta p_t$ relies on a surface pressure jump transfer function $H_S$ used to determine $\Delta p_t$ as a function of the incident pressure $p_i$. The current set-up featuring turbulent and laminar boundary lay-
Total surface pressure difference \( \Delta p_t \); (c) span average of \( \Delta p_t = p_{top} - p_{bot} \) from DNS, (—) using Amiet’s theory with \( \Delta p_t = [1 + H_S] p_i \); low frequency \( f = 0.5 \) (top) and medium frequency \( f = 1.25 \) (bottom).

The total pressure difference computed in this way can then be compared to the total surface pressure difference directly obtained directly from the DNS data, \( \Delta p_t = p_{top} - p_{bot} \). Hence, the surface pressure difference transfer function of Amiet (1976a), which is one of the key elements of the classical trailing edge theory, can be evaluated. A comparison between the predicted and the directly computed total surface pressure difference is shown in figure 3 over the length of the plate for two frequencies. It appears that the agreement between the predicted value of \( \Delta p_t \) and the DNS data improves for increasing frequency, although the predicted surface pressure difference is satisfactory even at lower frequencies. The good agreement with DNS data implies that obtaining the incident pressure field by adding the bottom surface pressure from the top surface pressure is a useful approach.

Other elements of Amiet’s trailing edge noise theory, such as the assumption of frozen turbulence spectra and an approximation of the spanwise correlation length were assessed using the DNS data and a detailed discussion is given in Sandberg & Sandham (2008). One aspect to be particularly aware of is that the use of periodic boundary conditions in the spanwise direction used for the simulations can artificially increase the amplitude of the noise measured in the simulations.

**Jet Noise**

Direct Numerical Simulations have been conducted of fully turbulent flow exiting a long pipe into a laminar co-flow. The inclusion of the pipe in the simulations with a fully turbulent flow inside ensures that all possible noise generation mechanisms are represented. The jet Reynolds number, based on pipe diameter and jet exit velocity at the axis, is \( Re_D = 7,500 \) and the Mach number of the jet and the coflow is varied. Here, only the case with jet Mach number \( M_j = 0.46 \) and coflow Mach number \( M_c = 0.11 \) is discussed. To fully resolve the pipe-jet configuration, \( 408 \times 10^6 \) grid points were required. The numerical set-up and further information about the flow parameters can be found in Sandberg et al. (2012). For a qualitative impression of the resulting acoustic fields, instantaneous contours of the dilatation field \( \partial u_i / \partial x_i \) in the entire domain are shown in figure 4. Sound waves can be observed emanating from the nozzle corner and from the jet core. Importantly, the resulting sound field appears to be very ‘clean’, i.e. no interference from boundary reflections can be detected. The sound appears to originate from the near-nozzle region with noise radiation showing the highest intensity at roughly \( \theta = 40^\circ \), where \( \theta \) is defined with respect to the streamwise direction. In addition, upstream radiating noise emanating from the nozzle lip, although more weakly, can be detected.
The noise of the turbulent jet generated by DNS was investigated with a phased array technique in order to understand the jet mixing noise and to identify any other sources of noise that might be present, e.g. nozzle lip noise. Although such techniques have been available for many years, successful application to LES and DNS data has often been prevented by a combination of two factors: the lack of far-field data and the lack of sufficient time history (record length) to obtain adequate accuracy in the required statistical quantities, such as the spectral density and cross-spectral density of the radiated unsteady pressure. In the current study, a polar array of virtual microphones was located at 30 jet nozzle diameters from the nozzle having ascertained that the field is obeying the inverse square law at this radius, without any significant reflections from the DNS domain boundaries. The DNS time history length permits the number of averages for the cross-spectral density to be of order 40 with a filter separation, expressed as a Strouhal number based on the nozzle diameter, of 0.1. A microphone spacing of 1 degree was used over an aperture of 140 degrees to 10 degrees to the jet axis (i.e. 40 to 170 degrees to the ‘intake’ axis), although much finer spacing is available if required. A detailed description of the phased-array technique can be found in (Tester & Gabard, 2012). Initial findings indicated the presence of a strong nozzle-based source over a wide range of frequencies. In order to investigate the jet mixing noise in more detail, the unsteady far-field pressure was decomposed into its constituent azimuthal modes \((m = 0, 1, 2, 3, 4, \ldots)\), so that for each mode \(m > 0\) the radiated pressure at frequencies below the cut-on frequency of that mode should be free of radiation from the internal source.

Source images are presented in a form illustrated in figure 5. The vertical axis depicts the Strouhal number, \(St_D\), covering a range 0.2 to 5, and the horizontal axis depicts the axial distance over \(-5\) to 15 diameters, with the jet nozzle at \(z/D = 0\). The images of the DNS data are calculated from the unsteady DNS pressures at the virtual microphone positions The horizontal lines correspond to the computed cut-on Strouhal number of the \(m > 0\) modes. For the axisymmetric mode \(m = 0\), the source region is localized to the nozzle \((z/D = 0)\) for all Strouhal numbers. When looking at the third azimuthal mode \(m = 3\), for frequencies below the pipe cut-on frequency, the image shows that the sources are located several diameters downstream of the nozzle. What appears to be jet mixing noise moves closer to the nozzle as the frequency is increased. Above the cut-on, the source is again closely tied to the nozzle. In summary, the source images show a strong nozzle-based source at all frequencies for \(m = 0, 1\), while for higher azimuthal modes the images can be split into two regimes: below the cut-on frequency of the first radial nozzle-acoustic mode for which jet mixing noise can be observed, and above cut-on for which the interior noise source dominates.

**Noise Reduction**

DNS of the flow around a NACA-0012 aerofoil at freestream Mach number 0.4 and chord-based Reynolds number \(Re_C = 50,000\) are conducted,
employing an immersed boundary method to represent flat-plate trailing-edge extensions with serrations. The geometry of the serrations is shown in figure 6, along with iso-surfaces of the second invariant of the velocity-gradient tensor to visualize the turbulent flow field in the vicinity of the trailing-edge modifications. The serrations are found to reduce the noise produced by the aerofoil, when compared with an equivalent unserrated case.

This noise reduction can be visualized by looking at contours of modulus of pressure, as shown in figure 7. The hydrodynamic field was not written to disk to reduce storage requirements; hence the near field region is cut out of the graphs. One-third octave averaging about the target frequency of $f = 7.75$ was performed to account for the broadband nature of the airfoil-noise. In previous studies it was found that this target frequency corresponds to the most amplified frequency of instabilities in the laminar-turbulent transition region and structures observed in the laminar-separated shear layer on the suction side of the airfoil are consequences of these instabilities. Furthermore, according to the analytical study by Howe (1991), significant noise reduction can be expected from serrations with the current dimensions at this frequency. As illustrated by figure 7 the trailing edge noise contribution below the airfoil indeed appears considerably weakened by the addition of TE serrations. Note that the noise reduction is not obvious on the upper surface due to the presence of additional noise sources associated with turbulent reattachment that cannot be expected to be affected by serrations.

From further analysis of the DNS data (Jones & Sandberg, 2012) it was found that neither the directivity or spanwise correlation levels of the acoustic radiation change significantly for frequencies at which significant noise reduction is achieved. In addition, Reynolds stress amplitudes, turbulent spectra and spanwise correlations are similar for the serrated and straight trailing-edge cases. This implies that changes in the sound radiation are caused mainly by changes to the scattering process itself.

6 Conclusions

The challenges associated with conducting high-fidelity aeroacoustic simulations have been discussed and a high-performance compressible Navier–Stokes solver purposely developed to exploit modern high-performance computing systems has been introduced. The code has been shown to perform well on a large-scale computing facility. Direct noise computations have been conducted for three different cases with the DNS data for every case being used in a different way. The data obtained from the DNS of turbulent flow over a trailing edge was mainly used to test assumptions made in commonly used trailing edge noise theories. Results from direct simulations of a turbulent pipe-jet configuration allow the use of a recently developed phased array code which revealed the dominance of nozzle-based noise sources. Finally, DNS were conducted of airfoils with serrated trailing edges to investigate the noise reduction mechanisms.

Acknowledgments

The author is grateful for the contributions of Professor N.D. Sandham and Drs B.J. Tester and L.E. Jones and for computing time provided through the UK Turbulence Consortium under EPSRC grant EP/G069581/1.

References


