46

Sound Generation by Vortex-Blade Interactions

G.S. DJAMBAZOV[1], C.-H. LAI[2], & K.A. PERICLEOUS[3]

Introduction

Aerodynamic sound generation is a result of the interactions of vortex structures that arise in viscous flows. A full aeroacoustic simulation therefore should include the generation of these vortex structures themselves. The present work concentrates on the second stage of the sound generation process, namely the actual production of acoustic waves by vortices hitting a solid object.

Computational Fluid Dynamics (CFD) codes can be employed to resolve the aerodynamic *sources* of sound if a proper software coupling is implemented with an acoustic (linearised Euler) solver. This coupling is necessary because the fast and robust numerical schemes that make CFD codes valuable are not suitable for wave propagation simulations [DLP98a]. It is most natural that the Domain Decomposition ideas should be applied in such circumstances.

The physical problem of aerodynamic sound can be decomposed in two sub-problems: airflow with sound generation, and sound propagation in non-uniformly moving media. When external noise problems are considered, there is no possibility of acoustic resonance, and any feedback from the propagating waves to the flow can be completely ignored because of the extremely small magnitude of the acoustic perturbations [Lig52].

Mathematically, decomposition is applied to the variables of the fluid motion:

---

1 School of Computing & Mathematical Sciences, University of Greenwich, Wellington Street, Woolwich, London SE18 6PF, U.K. Email: G.Djambazov@gre.ac.uk
2 Address same as above. Email: C.H.Lai@gre.ac.uk
3 Address same as above. Email: K.Pericleous@gre.ac.uk

Eleventh International Conference on Domain Decomposition Methods
Editors Choi-Hong Lai, Petter E. Bjørstad, Mark Cross and Olof B. Widlund

density, pressure and velocity as they are separated into flow and acoustic parts [DLP97a, Har93]. At each time step the flow part is resolved first. Information about the generated sound is extracted from the time-dependent CFD solution and is transfered into the acoustic solution via the source terms of the linearised Euler equations. This method was validated against analytical one-dimensional solutions, and two-dimensional results with simple geometry were shown to be physically correct [DLP98a].

In this paper the coupling technique employing the linearised Euler source term is considered for sound generation at solid surfaces with realistic geometry.

The PHYSICA Package as a CFD Code

The coupling between this software package and an acoustic software module will be used for the simulation of aerodynamic sound generated by vortex-blade interactions in *realistic* geometries. PHYSICA [CPC95] was selected for its flexibility with complex shapes and different numerical algorithms, and also, because it is being developed at the University of Greenwich.

The PHYSICA package has a flexible modular structure which allows various modelling procedures of various physical phenomena to be accessed in a single numerical simulation. New modules or new features of existing modules can be added to the package at any time. Currently, the following modules are available: heat transfer, fluid flow, solidification, and elastic/visco-plastic solid mechanics module. In the present study only the fluid flow module is used.

The solution algorithms in PHYSICA are based on *unstructured* meshes which can be comprised of cells of various types and shapes. This makes the modelling of curved solid boundaries very easy. With the fluid flow variables there is no staggering of the grids: the velocity vector components are stored at the centres of the cells together with the pressure and density values. In this way only one computational mesh is used during the whole simulation rather than four separate meshes needed with the staggered approach.

Second order schemes with the flow solution

Most CFD implementations provide as a default option the stable upwind scheme [VM95]. It ensures that during the iterative solution, an increase of a quantity at a given location will *always* be followed by an increase and not by a decrease at the neighbouring points. Unfortunately, this restricts the approximation of the variables to piecewise constant in both space and time.

For the *flow perturbations* which generate aerodynamic sound the QUICK differencing scheme [Leo79] is an alternative to increase the accuracy of the solution. With it the cell-face values of fluxes are calculated by second order interpolation between the two neighbouring nodes and an upstream node. The scheme can be formulated in a standard way and in several alternative ways [VM95]. The alternative formulations where troublesome negative coefficients are placed in the source term (right-hand side) of the discretised equations are usually preferred for stability reasons [Cro98].

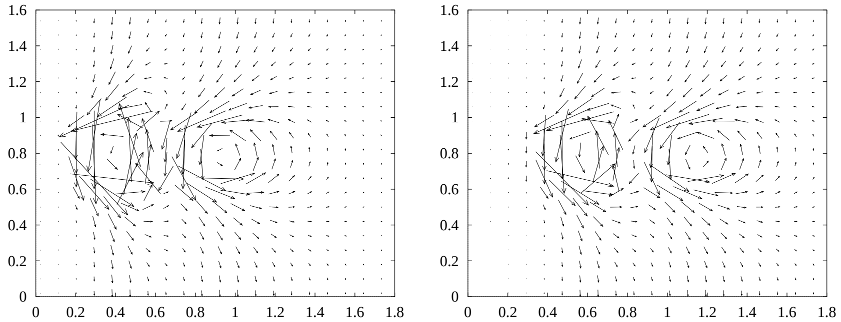

Figure 1 PHYSICA: Vortex convection with 2nd order (QUICK) scheme. Perturbation velocity vectors shown after subtracting the mean velocity from left to right (scale: 4 m/s to 0.1 m)

For the PHYSICA unstructured meshes a special procedure has been developed for finding the second upwind node [Cro98]. This implementation was tested with the inviscid example of vortices carried by a mean flow of 160 m/s (Figure 1).

The Acoustic Module

Since most CFD codes with which the acoustic module has to communicate, use finite volume methods, initially a finite volume algorithm of extended accuracy (to third order) was developed for the numerical solution of the linearised Euler equations [DLP97b]. However, optimised *finite difference* numerical schemes of higher order turned out to be less complex to implement in three dimensions and also exhibited better accuracy, so they were finally selected as a basis of the acoustic code.

The acoustic algorithm implemented solves the 3D linearised Euler equations (1) and (2) in a time-accurate way and exhibits the following features:

- Fully-staggered storage, with respect to pressure, of the velocity components in the three spatial directions and in time;
- Optimised forth order finite difference schemes:
  - — non-staggered convection terms [TW93]
  - — staggered propagation terms [DLP98b]
- Regular Cartesian grids with stepwise representation of solid boundaries;
- Mirroring of variables at solid walls for the missing values of the numerical scheme;
- Acoustic radiation boundary conditions.

The code can make direct numerical simulation of the *sound field* in an efficient way by solving the linearised Euler equations (1) and (2) given the sound sources (S, F_i) and the mean flow quantities: $\overline{v_j}$, $\overline{\rho}$, $c^2 = 1.4\overline{p}/\overline{\rho}$.

$$\frac{\partial p}{\partial t} + \overline{v_j}\frac{\partial p}{\partial x_j} \quad + \quad \overline{\rho}c^2\frac{\partial v_j}{\partial x_j} = S \tag{1}$$

$$\frac{\partial v_i}{\partial t} + \overline{v_j}\frac{\partial v_i}{\partial x_j} \quad + \quad \frac{1}{\overline{\rho}}\frac{\partial p}{\partial x_i} = F_i \tag{2}$$

The acoustic perturbation parts of the pressure and the velocity components are denoted by p and v_i respectively. The right-hand sides S and F_i accommodate, along with any external sources and forces, all the small nonlinear terms that arise when the equations of motion for the full variables $(\overline{\rho} + \rho)$, $(\overline{v_i} + v_i)$, and $(\overline{p} + p)$ are expanded [DLP97a]. Within the acoustic solver the right-hand sides are considered as known functions of x_i and time t. In some rare cases when long-distance or resonant nonlinear sound propagation effects have to be taken into account, the terms S and F_i can be evaluated iteratively.

The acoustic software module was validated against benchmark solutions [DLP98b]. It has two aspects of application. First, it can be used on its own with known mean flow and sound sources. Second, the module can be coupled with a CFD package to study the time-dependent noise generation by oscillating formations in the flow.

Sound sources on overlapping meshes

With the CFD simulation finite volume meshes are used, and the flow domain is divided into computational cells. If the assumption is made that the cells which become sources of sound are known in advance, the source term S (1) can be used to transfer the information about the generation of sound from the CFD code to the acoustic solver.

Closer examinations of the time history of test solutions obtained from *CFD* codes showed that the pressure at the *first* node next to the source of sound is resolved with sufficient accuracy. This suggests that when the source nodes are known, the time dependent CFD pressure at these nodes may be used to calculate the necessary source term S of the acoustic equations.

The following *assumption* has to be made: the CFD code resolves the *full* physical pressure (comprised of mean flow and acoustic components) in the first layer of computational cells next to a solid surface or in any other cells that have been identified as sound sources. The term 'resolves' is used here to denote that the CFD pressure is a good approximation of the true pressure signal in these selected cells.

Since the CFD pressure signal contains a mean-flow component, it cannot be fed directly into the acoustic code; the time dependent component has to be separated first. This can be done if a preliminary *steady* CFD solution is obtained in the same geometry, and the time dependent simulation is started with this initial condition. Then the difference between the time dependent and the steady pressure is the signal that has to enter the acoustic simulation at the prescribed source nodes.

One way of inserting this signal into the linearised Euler solver is to specify it as a fixed-value internal condition at the selected nodes. However, this will preclude the possibility of other acoustic waves (reflected from solid boundaries or generated by neighbouring source nodes) to propagate through the prescribed source layers.

The other option is to calculate the contribution of the CFD source to the local increment of pressure at the selected nodes over each time step. Since any transients associated with the establishment of the mean flow have been eliminated by starting

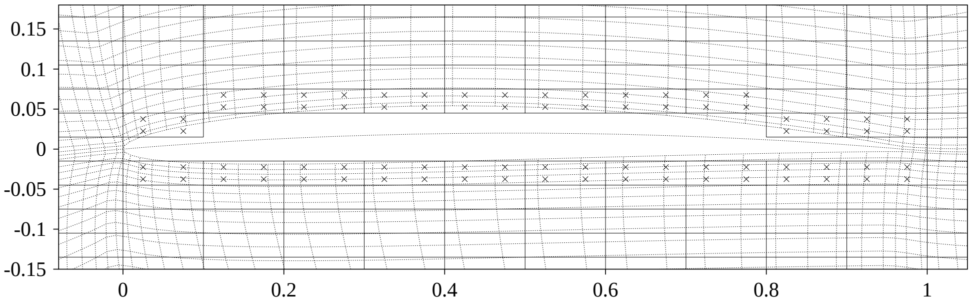

Figure 2 Overlapping CFD and CAA meshes and interpolation locations of the acoustic source cells

from a steady solution, this CFD contribution is simply the difference $\overline{p}(t) - \overline{p}(t - \Delta t)$ between the new and the old CFD pressure values.

If the CFD mesh and the acoustic mesh are the same, it is enough to add this difference to the other terms forming the acoustic pressure increment $p(t) - p(t - \Delta t)$ (see equation 1). However, most often this will not be the case because the CFD mesh is refined in the boundary layer while the acoustic mesh has to be coarse in order to obey the Courant limit. Therefore, it is best to express the CFD pressure contribution in terms of continuous quantities:

$$\overline{p}(t) - \overline{p}(t - \Delta t) = \frac{\partial \overline{p}}{\partial t}\Delta t. \tag{3}$$

Then the temporal derivative of the local pressure at the source nodes, calculated from the CFD solution, can be added to the source term S of the acoustic continuity equation (1):

$$S = \frac{\partial \overline{p}}{\partial t} + S_{vib}\,. \tag{4}$$

Here S_{vib} denotes sources external to the flow like vibrating solid objects.

The algorithm of the sequential coupling between the CFD code and the Computational Aeroacoustics (CAA) code based on the above definition of the linearised Euler source term was outlined in our previous communication [DLP98a].

The two codes (CFD and CAA) have separate meshes in overlapping domains. The CFD mesh must be body-fitted to represent smooth solid boundaries. The acoustic mesh is regular Cartesian to ensure high accuracy of the wave simulation, and the CAA domain can be larger than the flow domain because typically the acoustic wavelength is larger that the size of corresponding oscillating structures in the flow. In the latter case uniform mean flow is assumed outside the region of the CFD simulation.

Prior to the introduction into the acoustic simulation the flow quantities ($\overline{v_j}, \overline{\rho}, \overline{p}$, and c^2) have to be interpolated from the irregular CFD mesh. This mesh is usually finer than the acoustic mesh (in order to resolve vortices and boundary layers), and therefore, piecewise constant functions can be used for the interpolation. Also, averaging of the above flow quantities over each of the big acoustic cells has to be performed to ensure consistency of the communicated values. In Figure 2 the two overlapping meshes are shown, and for the designated acoustic source cells only, the interpolation locations are marked. The cell average values are then arithmetic averages of the interpolated values in these locations of each acoustic cell.

Simulation Results

As an example, the technique described above is applied to the sound generation due to vortex-blade interactions. The CFD body-fitted mesh around the section of the blade (aerofoil) can be seen in Figure 2. The production of sound by vortices hitting a blade is essentially an inviscid phenomenon since it is due to the inertial forces, and for this reason the boundary layer close to the aerofoil is not modelled. If the exact lift and drag are needed the mesh can be refined next to the aerofoil; this will not change the coupling technique in principle.

According to the algorithm outlined [DLP98a], first a **steady** solution of the airflow around the aerofoil was obtained. The aerofoil chord formed an angle of attack 7^0 with the free-stream velocity vector. No turbulence model was used, and inviscid flow was assumed instead, as explained above.

At the second stage of the aeroacoustic computation, the **time dependent** simulation of the flow is initialised with the steady solution, and with the inflow boundary conditions at the left end of the domain, a time dependent perturbation of the mean flow is specified in the perpendicular direction. It is sinusoidal with amplitude 7.5% of the mean velocity of 160 m/s and is applied to the inflow momentum in the middle part of the inflow boundary. In this way a series of vortex perturbations of the mean flow is generated. In a real aeroacoustic computation the flow perturbations (vortex structures) should not be prescribed but resolved within the CFD code. Due to the finer meshes and the use of turbulence models, the CFD part of the simulation is expected to be computationally much more expensive than the acoustic part.

The evolution of the flow field due to the vortex convection is illustrated by two snapshots in Figure 3 with arrows representing velocity vectors (scale: 4 m/s to 0.1 m). These vectors depict only the perturbation of the mean flow due to the passing vortices (the mean velocity vector has been subtracted before plotting).

At the third stage of this aerodynamic noise problem the special **acoustic** module (Section 46) is used. In order for the aerofoil to be better discretised on the rectangular acoustic mesh, the CFD mesh and velocity vectors have been rotated to the angle of attack around the leading edge of the aerofoil. The blocked cells forming the solid boundaries in the acoustic simulation have been omitted from the plot in Figure 2.

Although the flow solution does not contain acoustic waves, the sources of sound can be calculated from the pressure variations on the surface of the aerofoil. The pressure fluctuations (temporal derivatives) of the flow solution next to the solid surface are averaged over the rectangular cells which are neighbouring the blocked cells, and are imposed as the source term S of the linearised Euler equations. The aerofoil surface is assumed stationary ($S_{vib} = 0$). Since the linearised Euler solver uses explicit schemes, for stability reasons several acoustic time steps are needed to cover one flow time step.

In order to compare the mean convection and the sound propagation times, each of the instantaneous plots (Figure 3) shows the flow perturbation field and, superimposed on it, the resulting acoustic waves propagating away from the aerofoil. It can be

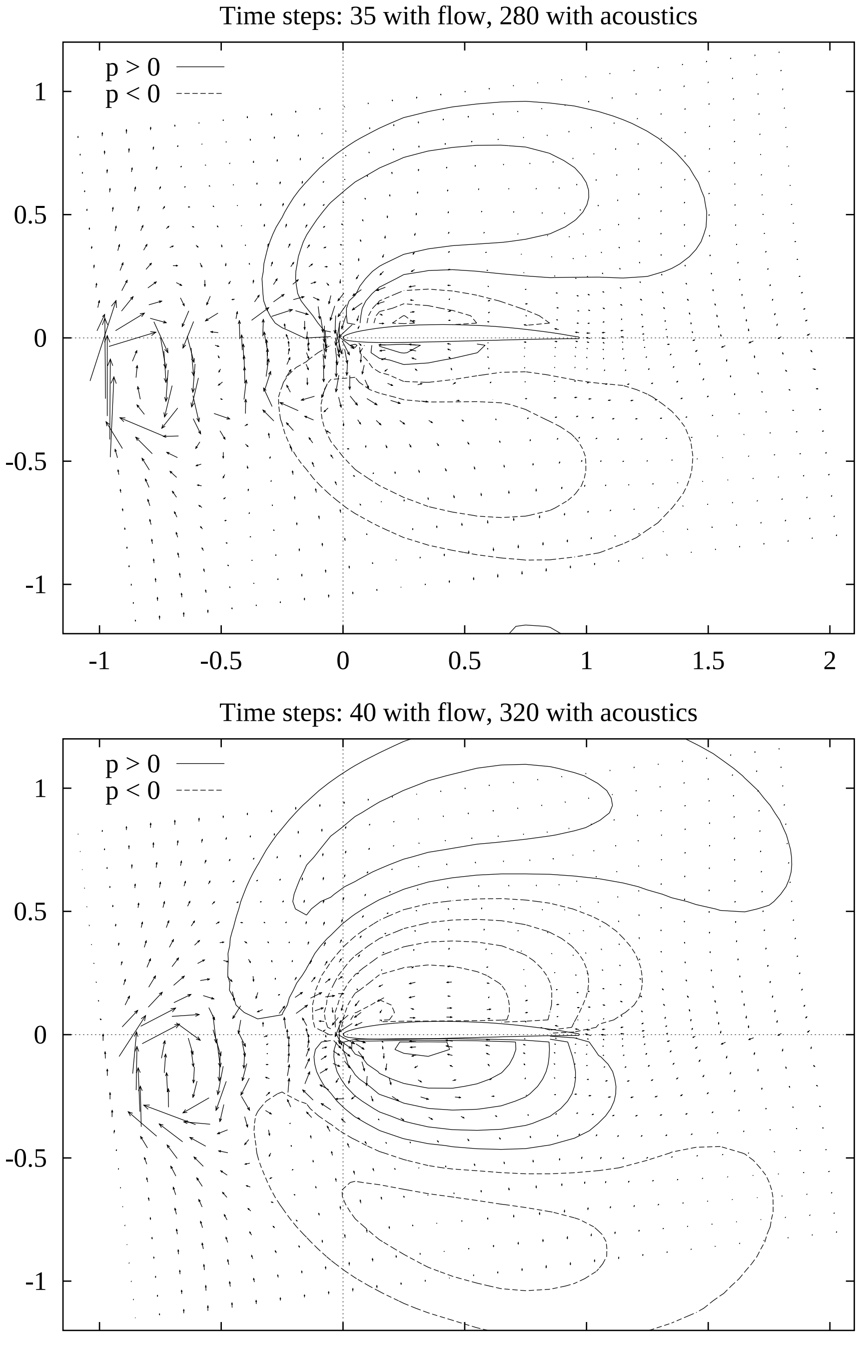

Figure 3 Flow and acoustic perturbation fields (showing superposition of mean-flow and acoustic domains)

seen that the mechanism of inviscid sound generation by perturbations in the flow has been captured by the combined simulation. The present implementation of the coupling technique does not account for sound generated in the wake downstream of the aerofoil.

Conclusions

The physical decomposition of the aerodynamic sound problem was implemented in two dimensions with realistic geometry using overlapping body-fitted and Cartesian meshes and two separate codes coupled in a mono-directional way. The technique was applied to vortex-generated sound at the surface of an aerofoil in sub-sonic conditions. There is no analytic validation but the results obtained are physically correct.

REFERENCES

[CPC95] Croft N., Pericleous K., and Cross M. (1995) PHYSICA: A multiphysics environment for complex flow processes. In Taylor C. *et al.* (eds) *Num. Meth. Laminar & Turbulent Flow '95*, volume 9, part 2, page 1269. Pineridge Press, U. K.

[Cro98] Croft N. (1998) *Unstructured Mesh - Finite Volume Algorithms for Swirling, Turbulent, Reacting Flows.* PhD thesis, University of Greenwich.

[DLP97a] Djambazov G., Lai C.-H., and Pericleous K. (1997) Development of a domain decomposition method for computational aeroacoustics. In *DD9 Proceedings.* John Wiley & Sons.

[DLP97b] Djambazov G., Lai C.-H., and Pericleous K. (1997) Testing a linear propagation module on some acoustic scattering problems. In Tam C. and Hardin J. (eds) *Second Computational Aeroacoustics (CAA) Workshop on Benchmark Problems*, number 3352 in Conference Publications, pages 221–229. NASA.

[DLP98a] Djambazov G., Lai C.-H., and Pericleous K. (1998) Efficient computation of aerodynamic noise. In *Contemporary Mathematics*, volume 218, pages 506–512. American Mathematical Society.

[DLP98b] Djambazov G., Lai C.-H., and Pericleous K. (1998) Staggered-mesh computation for aerodynamic sound. In *4th AIAA/CEAS Aeroacoustics Conference, part 1*, number 98-2219 in AIAA Papers, pages 97–103.

[Har93] Hardin J. C. (1993) Recent insights into computational aeroacoustics. In Mankbadi R. *et al.* (eds) *Computational Aero- and Hydro-acoustics*, volume FED-147, page 1. ASME.

[Leo79] Leonard B. (1979) A stable and accurate convective modelling procedure based on quadratic upstream interpolation. *Computer Methods in Applied Mechanics and Engineering* 19: 59–98.

[Lig52] Lighthill J. M. (1952) On sound generated aerodynamically: I. General theory. *Proceedings of the Royal Society* 211 A: 564–587.

[TW93] Tam C. and Webb J. (1993) Dispersion-relation-preserving finite difference schemes for computational acoustics. *Journal of Computational Physics* 107: 262–281.

[VM95] Versteeg H. and Malalasekera W. (1995) *An Introducion to Computational Fluid Dynamics. The finite volume method.* Longman.