Computers & Fluids 86 (2013) 1–13

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

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

A CFD/CAA coupling method applied to jet noise prediction

O. Labbé *, C. Peyret, G. Rahier, M. Huet

CFD and Aeroacoustics Department, ONERA – The French Aerospace Lab, Châtillon F-92322, France

A R T I C L E   I N F O

Article history:
Received 2 August 2012
Received in revised form 12 June 2013
Accepted 10 July 2013
Available online 17 July 2013

Keywords:
Turbulence
Large-Eddy Simulation (LES)
Subsonic jet noise
Computational aeroacoustics
Coupling method

A B S T R A C T

A new strategy regarding the simulation of sound generation and propagation is presented. A domain decomposition approach is used for the simulation of an aeroacoustic problem. The basic concept is to combine adapted numerical methods, equations, grids and even time steps for a greater efficiency. This aeroacoustic coupling is based on the splitting into noise sources generation and acoustic propagation in separate physical domains. The key idea of the present work is to limit, as much as possible, the CFD domain to the noise generation region that is often confined in a small part of the flow field, and to accurately propagate the acoustic waves with a CAA solver. Generally, such a reduction of the CFD domain requires coupling the CFD and CAA computations with an exchange boundary located within the turbulent flow. In the present paper, this splitting method is applied to a hot jet simulation. A LES based on the resolution of the Navier–Stokes equations with a Finite Volume Method on structured mesh is used to generate the acoustic sources, while an acoustic solver based on the resolution of Euler equations with a Nodal Discontinuous Galerkin Method on unstructured mesh propagates the acoustic waves. As a first step towards a full coupling, the present study deals with a one way coupling from LES to CAA.

1. Introduction

In the recent years the reduction of flow induced noise became an important problem in the development of new aircraft. There are many kinds of aerodynamic noise including turbine jet noise, jet/wing interactions, noise due to landing gears, wings, propellers, rotors, broadband noise due to inflow turbulence and boundary layer separated flow etc. Accurate prediction of noise mechanisms is essential in order to be able to control or reduce them to conform to noise regulations. Both theoretical and experimental studies are being conducted to understand the basic noise mechanisms. As the available computational power increases, numerical techniques are becoming more and more attractive and the prediction of aerodynamic noise sources and their propagation has led to the emergence of the Computational AeroAcoustics (CAA).

The numerical simulation of both generation and propagation of acoustic waves into the far field in one computation is actually a challenge due to the spatio-temporal scale disparities of the problem. The full time-dependent, compressible Navier–Stokes equations describe both aerodynamic and acoustic phenomena, but require a detailed numerical compressible flow simulation, using a grid fine enough to minimize the introduction of sound propagation errors. A large number of grid elements would be necessary to resolve all the scales accurately in an unsteady simulation. Moreover, the handling of long-distance sound propagation remains difficult with usual CFD solvers due to the numerical damping and dissipation. On the other hand, a single approach contains naturally the interaction of the acoustic perturbations with the flow field and with embedded geometries.

The computations of laboratory experiments with typical Reynolds number of about 10^5–10^6 are difficult to reproduce with a direct numerical simulation, Freund [1] for example, computed a jet at Mach number 0.9 and at Reynolds number 3600. Most of the unsteady computations of turbulent flows are using Large-Eddy Simulations solvers, except Lew et al. [2], who computed the far-field noise of an unheated jet at Mach number 0.4 at a Reynolds number of 6000 with a lattice-Boltzmann method and compared the results with those obtained with a Large-Eddy Simulation. In LES, only the large scales of the flow are resolved and the influence of the unresolved ones is modeled using a subgrid scale model [3]. By refining the mesh, the LES approaches the DNS, where all scales of the flow should be represented accurately. However, resolving only the larger scales significantly decreases the computational cost, making the LES more feasible for non-academic cases. The choice of the use of LES for sound predictions is also justified by the fact that large scales are more efficient than small ones in generating sound. Once the sound sources are predicted, several approaches can be used to describe the noise propagation. A simple strategy is to extend the computational domain for the full, nonlinear Navier–Stokes equations far enough to encompass the location where the sound must be calculated. In Constantinescu and Lele [4], the sound in the near field for jets at Mach 0.9 and at Reynolds numbers of 3600 and 72,000 are directly calculated from a LES. The peak of the near field noise spectra is accurately captured. The acoustic field can thus be obtained directly from the flow

---

* Corresponding author. Tel.: +33 1 46 73 42 50; fax: +33 1 46 73 41 66.
E-mail address: odile.labbe@onera.fr (O. Labbé).

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.07.013

simulation, but the sound waves are exposed to numerical dissipation which can lead to sound propagation errors.

These dissipation problems generally lead to carry out hybrid methods, in which the computational domain is split into different regions, such that the governing flow field (source region) or acoustic field (acoustic region) can be solved with different equations, numerical techniques and computational grids. Various hybrid methodologies exist, differing from each other in the type of applied propagation equations or in the way the coupling between source and propagation regions is made. Bailly and Bogey [5] proposed a review of the progress in the computational aeroacoustics field and discussed connections between CFD and CAA using hybrid approaches. The coupling methods commonly used for hybrid CFD/CAA applications are divided into two categories: one based on equivalent source formulations and the other based on an acoustic continuation of source region simulation. For the first one, once the sound source is predicted, the approach to describe its propagation is the extension of near-field CFD results to the acoustic far-field with surface or volume integral methods [6–9]. Bodony and Lele [10] made a survey of numerical methods used to predict the noise of turbulent jets by LES. Mach numbers between 0.3 and 2.0 are considered. In the available simulations, three techniques are used to project the mid-acoustic field onto the acoustic far-field: a simple extrapolation in $1/r$, the Kirchhoff surface to solve the inviscid, linear wave equation beyond the LES domain and the third one is the Ffowcs Williams and Hawkings approach. The two latter are efficient but make the assumption of propagation in a uniform flow with no reflecting bodies. The second hybrid CFD/CAA approach does not present these limitations and solve the Acoustic Perturbation Equations [11–13] or the Linearized Euler Equations [14,15] to extend the CFD solutions to the far-field. These propagation solvers are generally high order accurate but necessitate a mean flow definition. Moreover, the coupling boundary between the different domains is located outside the turbulent flow. Generally these hybrid methods do not take into account any acoustic feedback, except for the domain decomposition performed by [16,17], where the coupling approach connects different classes of methods on structured and unstructured grids for the solution of Navier–Stokes, Euler and linearized Euler equations.

The approach presented in this paper is also a domain decomposition method as in [16,17], it is based on the coupling of different equations, grids and time steps, which allows a simulation of both flow and acoustics in one single calculation suitable for far field predictions with reflecting bodies. The CFD domain solving the Navier–Stokes equations is reduced to the region of viscous effects and initial turbulence development which generally accounts for a small part of the flow. The acoustic propagation is solved with the full non-linear Euler equations with a coupling boundary located in the turbulent flow. The acoustic solver is based on high order Discontinuous Galerkin schemes [18], which offer a high accuracy, low dispersion and low dissipation. This class of solvers is able to accurately propagate waves over large distances and allows using unstructured grids, which present significant advantages such as a highly flexible refinement even in complex geometries. In the first part of this paper, the CFD/CAA splitting approach is described with the different numerical tools and the coupling procedure, while the second part is devoted to the application of the method to the simulation of a hot jet.

2. CFD/CAA splitting approach

The coupling method presented in this paper consists in separating the whole computational domain into two complementary parts. The first domain is assumed to contain the region where turbulence develops and its size is reduced as much as possible. The flow in this domain is solved using the Navier–Stokes equations. The second domain is devoted to the propagation of the perturbations generated in the first one, in a flow which is not necessary uniform and may contain reflecting bodies. It may also include noise production, in which viscous effects can be negligible. The problem is solved using the full Euler equations. Because the equations in perturbations are not considered here, no mean flow has to be defined in the acoustic domain.

Such LES/CAA couplings pose, a priori, the problem of the continuity of the solution on both sides of the exchange surface. Indeed, the LES must be performed using sufficiently fine meshes and time steps small enough to correctly simulate the development of turbulence. On the other hand, to be efficient, the CAA must be carried out using meshes and time steps adapted to the acoustic scales which are generally much larger than the turbulence scales. So, the spatial and temporal scales may be very different on both sides of the coupling boundary.

The solution proposed in this study is to define a buffer zone between the aerodynamic and the acoustic domains which ensures a progressive transition of the space and time scales between both calculations. This buffer zone filters the small aerodynamic scales present on the coupling boundary that cannot be taken into account with the acoustic mesh and time step. In that way, this LES/CAA junction avoids performing an explicit filtering of these small aerodynamic scales in the LES data, contrary to the one-way CFD/CAA coupling procedure proposed by Cunha and Redonnet [13].

This buffer zone can be placed either in the aerodynamic domain or in the acoustic domain depending on the meshes and the methods used in each of them. In the present case, it is placed in the acoustic domain because, as it will be detailed further, the acoustic solver is based on a Nodal Discontinuous Galerkin Method with local time stepping and using unstructured meshes with adaptation techniques [19] which makes easier a smooth variation of the space and time discretizations.

As a first step towards a full coupling, the present study only deals with a one way coupling from LES to CAA and thus mainly focuses on the acoustic purpose. As a consequence, the coupled simulations are carried out without feedback from CAA to LES.

2.1. LES/CAA tools

The aerodynamic solver FUNk [20] developed at ONERA, is based on the compressible Navier–Stokes equations expressed in conservative form. The LES equations are obtained using Favre filtering and the filtered equations are closed by means of a subgrid scale viscosity and the Prandtl analogy. The model used to compute the subgrid viscosity is the selective mixed scale model introduced for compressible flows by Lenormand et al. [21]. The spatial discretization method is based on the cell-centered Finite Volume methodology (FVM) on structured grid. An upwind biased scheme, with a third-order MUSCL interpolation scheme of AUSM+(P) family without any shock capturing feature is used for the convective terms. A second-order-accurate centered scheme is used for viscous fluxes. The time integration is carried out by means of a third-order compact Runge–Kutta scheme. The whole process is detailed in Larchevêque et al. [22].

The CAA computation is carried out with the SPACE solver developed at ONERA [23,24]. In this study, the full non-linear Euler equations are used to solve the CAA domain with a Discontinuous Galerkin Method (DGM) which is well adapted to unstructured meshes and makes easier the access to high order schemes. The fluxes are computed with a local Lax Friedrich type scheme. A nodal DGM with a Lagrangian polynomial basis is used to solve the conservative form of Euler equations. The high order ability of