C. R. Mecanique 346 (2018) 903–918

Contents lists available at ScienceDirect

Comptes Rendus Mecanique

www.sciencedirect.com

Jet noise modelling and control / Modélisation et contrôle du bruit de jet

A diagnostic tool for jet noise using a line-source approach and implicit large-eddy simulation data

Florent Margnat [a,*], Vasilis Ioannou [b], Sylvain Laizet [b]

[a] Institut Pprime, CNRS, Université de Poitiers, ISAE–ENSMA, 86962 Futuroscope Chasseneuil, France
[b] Department of Aeronautics, Imperial College London, London, SW7 2AZ, UK

ARTICLE INFO

Article history:
Received 6 September 2017
Accepted 1 February 2018
Available online 24 July 2018

Keywords:
Turbulence
Large-eddy simulations
Acoustic predictions

ABSTRACT

In this work, we propose a cost-effective approach allowing one to evaluate the acoustic field generated by a turbulent jet. A turbulence-resolving simulation of an incompressible turbulent round jet is performed for a Reynolds number equal to 460,000 thanks to the massively parallel high-order flow solver Incompact3d. Then a formulation of Lighthill's solution is derived, using an azimuthal Fourier series expansion and a compactness assumption in the radial direction. The formulation then reduces to a line source theory, which is cost-effective to implement and evaluate. The accuracy of the radial compactness assumption, however, depends on the Strouhal number, the Mach number, the observation elevation angle, and the radial extent of the source. Preliminary results are showing that the proposed method approaches the experimental overall sound pressure level by less than 4 dB for aft emission angles below 50°.

1. Introduction

The overall goal of the present study is to develop and validate a cost-effective hybrid jet noise prediction tool that could potentially offer a competitive trade-off between the accuracy of the experiments (in the same operating conditions) and the computational cost in terms of wall clock time and data storage. Such a tool could be critical for the efficient design of quieter turbulent jets and for quickly evaluating and comparing different noise mitigation strategies. For instance, to reduce the sound generated by aircraft engines, Boeing, General Electric, and NASA developed several years ago serrated edges called chevrons for engine exhaust nozzles [1,2]. The chevrons effectively reduce jet noise by controlling the way the air mixes after passing through and around the engine. However, the large number of parameters affecting the performance of such a control solution makes their development, testing and optimisation a very complicated task, with potentially numerous high-cost and time-consuming trial-and-error iterations. For this reason, cost-effective diagnostic tools are needed in order to quantify how the acoustic field is modified when a control solution is used, but more importantly to adjust and compare different parameters of the control solution in an effective fashion. The expectation for such a diagnostic tool may include quantitative and qualitative criteria, such as a correct estimation of the global sound levels without any empirical constant, and an ability to reveal significant changes in the directivity pattern and frequency content.

* Corresponding author.
E-mail addresses: florent.margnat@univ-poitiers.fr (F. Margnat), vasilis.ioannou14@imperial.ac.uk (V. Ioannou), s.laizet@imperial.ac.uk (S. Laizet).

https://doi.org/10.1016/j.crme.2018.07.007

Coupling compressible, large-eddy simulations (LES) with a wave extrapolation from a Kirchhoff surface is currently the most adopted strategy to quantify noise levels generated by turbulent jets. However, because the acoustic predictions are often carried out at the same time as the simulation, the potential for computational cost reduction is quite small. The very high cost of this direct approach, very successful to unlock the sound generation mechanisms for turbulent jets, is still a limiting factor to be used as a cost-effective diagnostic tool. Recent works have provided powerful insight into the ingredients of jet noise generation that might be helpful when trying to design a diagnostic tool. From the flow dynamic point of view, the boundary layer characteristics inside the nozzle [3] and the hydrodynamic instabilities at the initial stage of the shear layer have been identified as driving phenomena in the early stages of the development of turbulent jets [4,5]. From an acoustic point of view, the azimuthal mode decomposition of the acoustic field reveals that most of the energy is contained in the couple of first modes [6]. The near-field properties of those modes has been characterised by [7] through the streamwise evolution of instability waves. The higher modes are formally less efficient to radiate to the far-field. Another important feature of the acoustic field is its directivity, which needs an extended source to be reproduced [8], meaning that phase effects along the jet axis must be accounted for by any modelling. As for phase effects in the third direction, the transverse one, the lower the frequency and the radiation angle are, the weaker their influence is.

Taking into account all these considerations, a hybrid methodology is developed in the present study in order to evaluate sound levels generated by turbulent jets. The first step generates source data by an optimised turbulence-resolving simulation using an incompressible, massively parallel, high-order solver. The nozzle is included in the computational domain for a more realistic representation of the flow physics and an implicit large-eddy simulation (ILES) strategy is used to reach high Reynolds numbers while resolving a wide range of turbulence scales. Such a simulation can be performed in a few hours on a few thousands cores, contributing to an excellent balance between wall-clock time and accuracy. The second step is an azimuthally reduced, semi-compact formulation of Lighthill's solution: all effects are included in the axial direction, while only the five first azimuthal modes of the source quantity are propagated and a first-order compactness assumption is made in the radial direction. This leads to a line-source approach, for which the storage need and computational time are almost negligible with respect to the flow simulation.

The paper is organised as follows: in section 2, the numerical strategy for the flow simulation is described, and the velocity field is compared with experimental data [6,9,10] performed at the same Reynolds number. The acoustic methodology is presented in section 3, while the predicted acoustic field is validated in section 4 versus the experimental acoustic measurements [11] generated in the same facility as for the velocity field data. The range of validity of the radial compactness assumption is thus studied in terms of aft radiation angle and frequency. The full derivation of the present method as well as its limitations are presented in the appendix.

2. Flow simulation

The main goal of the flow simulation reported in this section is to provide the velocity field data necessary for the acoustic study.

2.1. Numerical methods

The governing equations are the forced incompressible Navier–Stokes equations

$$\frac{\partial \mathbf{u}}{\partial t} = -\nabla p - \frac{1}{2}\left[\nabla\,(\mathbf{u}\otimes\mathbf{u}) + (\mathbf{u}\cdot\nabla)\mathbf{u}\right] + \nu\nabla^2\mathbf{u} + \mathbf{f} \tag{1}$$

$$\nabla\cdot\mathbf{u} = 0 \tag{2}$$

where $p(\mathbf{x}, t)$ is the pressure field (for a fluid with a constant density) and $\mathbf{u}(\mathbf{x}, t)$ the velocity field. Note that convective terms are written in the skew-symmetric form as it allows the reduction of aliasing errors while remaining energy conserving [12]. In these forced Navier–Stokes equations, the forcing field $\mathbf{f}(\mathbf{x}, t)$ is used through a customized immersed boundary method (IBM) allowing us to include the nozzle inside the computational domain. Following the work of [13], the present methodology is based on an alternating direction forcing strategy for which a no-slip boundary condition is imposed at the wall of the nozzle in each spatial direction, while an artificial flow is introduced inside the nozzle to avoid any loss of continuity on the velocity field. From a practical point of view, this artificial expansion of the flow inside the nozzle, based on Lagrange polynomials, is performed in the direction where a spatial derivative is evaluated. As a consequence, a different expansion is generated, depending on the spatial direction of the computed derivatives. When combined with low-order schemes, a loss of continuity would only have a minor impact on the solution. However, when combined with high-order schemes, it can generate spurious oscillations on the derivatives at the wall of the nozzle. More details about this technique can be found in [13].

The governing equations are solved using the flow solver `Incompact3d` based on a Cartesian mesh, finite-difference sixth-order schemes for the spatial discretization and a conventional fourth-order Adams–Bashforth scheme for the time advancement. To treat the incompressibility condition, a fractional step method requires to deal with a Poisson equation, fully solved in spectral space via the use of relevant 3D Fast Fourier transforms. Combined with the concept of modified wave number [14], this direct (i.e. non-iterative) technique allows the implementation of the divergence-free condition up to machine accuracy. A partially staggered mesh is used where the pressure mesh is shifted by a half-mesh from the velocity