Numerical Investigation of Flow-Induced Noise Generation at the Nozzle End of Jet Engines

Andreas Babucke, Markus Kloker, and Ulrich Rist
Institut für Aerodynamik und Gasdynamik, Universität Stuttgart
Pfaffenwaldring 21, D-70550 Stuttgart, Germany
babucke@iag.uni-stuttgart.de
http://www.iag.uni-stuttgart.de

Summary

Sound generation downstream the nozzle end of a subsonic laminar jet has been investigated using two-dimensional direct numerical simulations (DNS). The nozzle end is modeled by a finite flat plate with Mach numbers of $Ma_I = 0.8$ above and $Ma_{II} = 0.2$ below the splitter plate. Behind the nozzle end, a combination of a wake and mixing layer develops. Due to the high amplification rates, disturbances saturate before a pure mixing layer occurs. Non-linear generation mechanisms produce higher harmonic disturbances in the upper boundary layer, resulting in an only quasi periodic solution. The main acoustic sources correspond to the positions of vortex pairing. Broadband noise is emitted instead of tonal noise, known from the pure mixing layer without a splitter plate [4].

1 Introduction

Noise reduction is an important issue for a wide range of technical problems but the mechanisms of flow-induced sound are not yet understood properly. The current investigation focuses on jet noise as it is a major noise source of aircrafts. Aero-acoustic simulations are a relatively new field in computational fluid dynamics demanding high requirements in computational performance and the accuracy of the computational scheme itself. On the one hand, a high resolution is needed to compute the noise sources accurately, on the other hand, it demands a large computational domain to obtain the relevant portions of the acoustic far-field. As the acoustic amplitudes are small compared to the flow-field disturbances, boundary conditions have to be chosen carefully, in order not to spoil the acoustic field with reflections.

Previous investigations of jet noise have been focusing on either pure mixing layers [3, 4] or low Reynolds number jets [5], where a S-shaped velocity profile is used at the inflow. In this two-dimensional DNS, we include the nozzle end, modeled by a thin flat plate with two different free-stream velocities on top and below. This allows to investigate the influence of the wake formed by the two boundary layers. Additionally, the splitter plate gives the possibility to model real actuators for future noise reduction instead of unrealistic volume forcing.
2 Computational Configuration

2.1 Numerical Method

The two-dimensional simulation is a first step and has been performed using the NS3D code [2]. It solves the unsteady compressible Navier-Stokes equations. The code is written in conservative formulation, solving for density $\rho$, the momentum densities $\rho u$, $\rho v$ and the total energy per volume $E$. The velocity components are normalized by the inflow velocity $\overline{U}_\infty$, the pressure with $\overline{p} \cdot \overline{U}_\infty^2$ and all other quantities by their inflow values of the upper boundary layer (see figure 1), marked with the subscript $\infty$. Length scales are made dimensionless with a reference length $\overline{L}$ and the time with $\overline{L}/\overline{U}_\infty$, where the overbar denotes dimensional quantities. The viscosity is modeled using the Sutherland law with a reference viscosity $\overline{\mu}(\overline{T}_\infty = 280K) = 1.735 \cdot 10^{-5} \text{ kg/(ms)}$. Since we can assume a weak temperature dependence, the Prandtl number $Pr = 0.71$ and the ratio of specific heats $\kappa = 1.4$ are taken constant.

In streamwise ($x$) and normal ($y$) direction, the flow-field is discretized by $6^{th}$-order compact finite differences. Alternating up- and downwind-biased differences [7] are applied to convective terms for de-aliasing. Second derivatives are computed directly instead of applying the first derivative twice. This leads to better resolved viscous terms and improves the stability of the code [1]. Grid transformation in the $x$-$y$ plane is implemented by mapping the physical grid on an equidistant computational $\xi$-$\eta$ grid. The equations are integrated in time using the standard $4^{th}$-order Runge-Kutta scheme. The domain decomposition in the $x$-$y$ plane is not only used for parallelization, it also allows to define neighbours or specific boundary conditions for each domain. Therefore, the nozzle end can be included easily without any special treatment of corner points.

At the free stream boundaries, a one-dimensional characteristic boundary condition [6] is used. An additional damping zone forces the flow variables smoothly to a steady state solution. Prescribing amplitude and phase distributions from linear stability theory allows to introduce defined disturbances at the subsonic inflow with characteristic boundary conditions [6]. The outflow is the most crucial part as one has to avoid large structures passing the boundary and contaminating the acoustic field. Therefore, a combination of grid stretching and spatial low-pass filtering is applied in the sponge region. Disturbances become increasingly badly resolved as they propagate through the sponge region. As the spatial filter depends on the step size in $x$-direction, perturbations are smoothly dissipated before they reach the outflow boundary. This procedure shows very low reflections and has been already applied by Colonius et al. [4].

2.2 Flow Parameters

For the current investigation, an isothermal subsonic jet with the Mach numbers $Ma_I = 0.8$ for the upper and $Ma_{II} = 0.2$ for the lower stream has been selected. As both temperatures are equal ($T_1 = T_2 = 280K$), the ratio of the streamwise