ICA 2013 Montreal  
Montreal, Canada  
2 - 7 June 2013  

Noise  
Session 3aNSb: Aviation, Aviation Engines, and Flow Noise  

3aNSb1. A robust numerical approach for the prediction of turbofan engine noise  
Kaveh Habibi*, Hao Gong and Luc Mongeau  

*Corresponding author’s address: Mechanical Engineering, McGill University, 817 Sherbrooke St. West, Montreal, H3A 0C3, Quebec, Canada, kaveh.habibi@mail.mcgill.ca  

Noise from aircraft jet engines remains a dominant sound source in takeoff conditions. Government regulations and community noise criteria are getting more stringent, creating significant challenges for aircraft manufacturers to meet noise requirements. The increased costs of experimental studies as well as access to powerful computers have given new motivation for computational studies. In this paper, an efficient and robust numerical scheme, namely the Lattice Boltzmann Method, was used to simulate the sound created by typical internal mixing nozzles with forced mixers. The simulation captured the time-resolved flow characteristics and large scale turbulent structures. The sub grid scales were modeled using the renormalization group (RNG) forms of the k-ε equations. Two cold flow test cases, conducted by NASA, were selected for computational setup and validation purposes. The far field sound was predicted using a surface integral method. The near-field simulation results such as the internal velocity profiles as well as far-field sound were qualitatively in agreement with experimental results. The far-field sound analysis suggested significant low-frequency noise reduction for the lobed mixers, as well as significant reduction in overall sound pressure levels (OASPL) in comparison with the confluent nozzle configurations.
INTRODUCTION

The prediction of jet noise from jet flows through complex nozzle configurations continues to be a problem for industry despite decades of research in flow-induced aerodynamic noise. The lack of reliable noise prediction models for the complex nozzle geometries used in modern gas turbine engines prevents noise to be included as a design factor for new mixer geometries. As a result, expensive experimental tests are required to assess the noise from new internal mixing nozzles to determine whether they meet government noise regulations.

Mixers are used to mix the core and bypass flows internally before they exit through the nozzle\textsuperscript{1,2}. Better uniformity of the flow velocity profile at the nozzle exit plane tends to reduce noise levels, as well as produce better cruise thrust efficiency. This benefit needs to be traded off against pressure losses, which increase with enhanced mixing. The net performance gain depends on the balance between improved mixing and the pressure losses. Noise reduction devices include chevrons, injection (i.e. fluidic chevrons), vortex generators, exhaust tabs, diverters and lobed mixers. Lobed mixers have been found to yield significantly enhanced mixing, with acceptable pressure losses. The main application of lobed mixers is for internal-mixing engines. A lobed mixer is basically a splitter plate with a convoluted trailing edge which alternately diverts the upper and lower streams towards the opposite stream. The key parameters of a lobed mixer nozzle include lobe number, lobe penetration, scalloping depth (if any), perimeter of the trailing edge and mixing length. For external mixing designs, chevron configurations are preferred.

Several studies have investigated the mechanisms of enhanced mixing by lobed mixer. Measurements performed by Patterson\textsuperscript{3} have quantified the radial outflow in the core region and the radial inflow in the fan region at the lobe exit plane. The results revealed the existence of large-scale streamwise vortices which were suggested to be responsible for the enhanced mixing. Werle et al.\textsuperscript{4} and Eckerle et al.\textsuperscript{5} suggested a three-step mechanism. Streamwise vortices were postulated to form, intensify, and then break down. This suggested that the high turbulence resulting from vortex breakdown improved the overall mixing process. Manning\textsuperscript{6} reported the separated role of the streamwise vorticity and the increased interfacial area on the enhanced mixing, and reached the conclusion that the mixing enhancements of the lobed mixers increased with velocity ratio. Elliot et al.\textsuperscript{7} reported three primary contributors to the mixing process in lobed mixing flows, namely spanwise vortices due to the Kelvin–Helmholtz instability, increased interfacial contact area due to the trailing edge shape, and streamwise vortices produced by the geometry. More recently, Mengle and Dalton\textsuperscript{8} and Mengle et al.\textsuperscript{9} published comprehensive data on the effects of scalloping, lobe number, lobe penetration and mixer-nozzle configuration on radiated noise for wide range of engine operating conditions.

From a computational view point, the design of the noise-suppressing nozzle design requires computational fluid dynamics (CFD) and computational aeroacoustics (CAA), dealing with large-scale turbulent flow physics. Computational studies using Navier-Stokes formulations only focused on the region downstream of the lobe exit\textsuperscript{10,11,12}. Barber et al.\textsuperscript{13} performed RANS simulations of jet flows with lobed mixers. Salman et al.\textsuperscript{14,15} used both structured and unstructured grids to study lobed mixers jet flows. Garrison\textsuperscript{16} carried out RANS calculations based on the WIND flow solver with a two-equation turbulence model. The results were found to capture many features of lobed mixer flows. Despite all these efforts, computational tools with high accuracy, high efficiency, stability, and relatively low cost are still needed to elucidate the flow and noise characteristics of lobed mixer flows. The Lattice Boltzmann Method (LBM) is proposed for this purpose.

The LBM is based on kinetic theory and considers a particle distribution function in a discrete lattice domain. By imposing streaming and collision laws governed by the Lattice-Boltzmann equation (LBE), transient macroscopic fluid properties are obtained. Through the Chapman-Enskog expansion\textsuperscript{17}, the LBE recovers the compressible Navier-Stokes equation at the hydrodynamic limit\textsuperscript{18,19,20}. In this method, conservative flow variables are replaced by particle density function obtained by solving the Lattice Boltzmann equation (i.e. discussed in next section). The potential advantages of LBM over the conventional Navier–Stokes solvers include (1) Linearity of the convection operator due to the kinetic nature of the LBE method.\textsuperscript{21} In addition to particle convection and collisions, the use of multi-scale expansions allows the recovery of the nonlinear macroscopic advection process. (2) Direct obtainment of the strain rate from the non-equilibrium distribution function. From the mesoscopic point of view of LBM, the Poisson term e.g., strain and rotation rate tensor in the coarse grained Navier–Stokes equations often cause numerical difficulties in terms of accuracy for finite-difference based algorithms.\textsuperscript{22} (3) Suitability for complex geometries, due to the absence of Jacobians to compute grid metrics. This feature is essentially an advantage when applied to jets with scalloped mixers and chevrons. It also facilitates the inclusion of nozzles in the computational domain. (4) Ease of parallelization for large to massive supercomputing architectures due to its simplicity in terms of form.
This paper aims to study the feasibility of using LBM for time-resolved simulation of dual-stream nozzles with complex-shaped mixers. The PowerFLOW 4.3d solver licensed by Exa Corporation was used for LBM simulations. Unlike the previous RANS studies reported in the literature, LBM captured instantaneous flow characteristics such as flow separation, vortex shedding, internal mixing layer formation, external shear layer development and local sound radiation from the jet plume. Standard confluent and 12-lobed NASA mixers were selected for case studies.

LATTICE BOLTZMANN METHOD

By considering $\Delta x$ as lattice unit length and $\Delta t$ as lattice unit time step, the lattice-Boltzmann equation may be written as:

$$f_i(\hat{x} + c_i \Delta x, t + \Delta t) - f_i(\hat{x}, t) = -\frac{\Delta t}{\tau} \left( f_i - f_i^{eq} \right),$$  

(1)

where the particle density distribution function $f_i(\hat{x}, t)$ can be interpreted as a typical histogram representing a frequency of occurrence at position $\hat{x}$ with particle velocity $c_i$ in the $i$ direction at time $t$. The function may be considered as a direction-specific fluid density. The propagation (i.e. convection) of particles through the discretized lattice domain is represented by the left hand side of Eq. (1), whereas the right hand side represents the collision operator adopted from the Bhatnagar-Gross-Krook (BGK) approximation. The relaxation time parameter, $\tau$, used in this model is related to the kinematic viscosity, $\nu$, such that $\tau = (\nu + \Delta t) / T$. The equilibrium distribution function ($f_i^{eq}$) relates the lattice Boltzmann model to hydrodynamics properties and is essential for local conservation criteria to be satisfied. In our study, $f_i^{eq}$ is adopted from the D3Q19 model. It can be shown that this model is second order in both time and space. The conservative fluid properties such as density and momentum are obtained by summing the moments along each velocity direction.

$$\rho = \sum_{i=1}^n f_i(\hat{x}, t), \quad \rho \hat{u}(x, t) = \sum_{i=1}^n f_i(\hat{x}, t).$$

(2)

A specific volumetric boundary scheme was used in order for LBM to achieve a particle bounce-back boundary conditions. This algorithm is well customized for complex geometries such as lobed mixers. Turbulent wall boundary conditions are applied by a generalized LBM slip algorithm and a modified wall-shear stress model significantly reducing the near wall grid resolution required for capturing turbulent structures close to the solid boundaries. More details on the application of LBM in jet flow simulations may be found in Lew et al. and Habibi et al.

In the VLES method, large scales are directly simulated. In order to account for subgrid turbulent fluctuations, the LBE is extended by replacing its molecular relaxation time scale with an effective turbulent relaxation time scale. The time scale is derived from a systematic renormalization group (RNG) procedure defined by Eq. 3.

$$\tau_{eff} = \tau + C_\mu \frac{k^2/\nu}{T \left(1 + \eta^2\right)^{3/2}},$$

(3)

where $C_\mu$ is constant and $\eta$ is a combination of a local strain parameter $(k |S_y| / \nu)$, local vorticity parameter $(k |\Omega_x| / \nu)$ and local helicity parameter. A modified two-equation “k-\epsilon” model based on the original RNG formulation describes the subgrid scale turbulence contributions. The turbulence model energy production and dissipation equations may be written as:

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\rho v_0}{\sigma_0} + \rho \frac{v_\epsilon}{\sigma_\epsilon} \right) \frac{\partial k}{\partial x_j} \right] + \tau_{\epsilon} S_{\epsilon} - \rho \epsilon,$$

(4)

$$\rho \frac{D\epsilon}{Dt} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\rho v_0}{\sigma_0} + \rho \frac{\epsilon}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{\epsilon 2} \tau_{\epsilon} S_{\epsilon} - \left[ C_{\epsilon 2} + C_\mu \frac{\eta^3}{1 + \beta \eta^2} \right] \frac{\rho \epsilon^2}{k},$$

(5)
where the parameter, $\nu_e = C_k \frac{k^3}{\varepsilon}$, is the eddy-viscosity in the RNG formulation, $\sigma_{c_0}, \sigma_{c_1}, \sigma_{c_2}, \sigma_{c_3}$, and $\beta$ are constants, either derived from the RNG procedure or tuned for internal and external flow configurations. The above equations were solved using a modified Lax-Wendroff explicit 2nd order finite difference scheme.

**COMPUTATIONAL SETUP AND TEST CASES**

Two sets of one-quarter scale nozzles, mixers and center-bodies were selected from NASA experiments. Mixer properties are listed in Table 1. Key geometrical parameters and tested models are shown in Fig. 1 and Fig. 2 respectively. Cold and hot test set points were extracted from NASA reports. Total pressure ratios, total temperature ratios, and mass flow rates were adjusted to achieve a fixed bypass ratio of 3 and peak exit Mach number less than 0.5 ($i.e. M_f \leq 0.5$). These conditions stay within the weak compressibility limit of the D3Q19 LBM model. Based on mean velocity and pressure magnitudes obtained from scaled NASA dataset for core and bypass flows, a hyperbolic tangent velocity profile and uniform pressure distribution were applied as inlet boundary conditions at the commencement of nozzle.

**FIGURE 1.** Nozzle-Mixer characteristic Lengths.

**FIGURE 2.** Dual-stream mixing nozzles, a) Confluent and b) 12CL.

The selected geometry is a contoured convergent nozzle with a diameter of $D_j = 7.27'' (0.185 \text{ m})$ and a length of $L = 1.24 \times D_j$. The computational domain size was chosen to be $27 \times 22.8 \times 22.8 \times D_j$. To achieve a more realistic turbulent shear layer at the nozzle outlet, 4% velocity perturbation was forced at both the fan and core flow inlets. A random forcing function was chosen according to Bogey et al. to minimize spurious sound radiation to the far-field. The computational domain is divided into structured lattice arrays with variable resolution (VR). Six levels of VR zones were used in the numerical setup. VR zones acts as the grid refinement in numerical finite-difference schemes. The grid setup is shown in Figs 3-a and 3-b. Additional resolution was provided through smaller grid cells (voxels) close to solid boundaries and through the shear layer (Fig. 3-a). Lattice length conversion from one VR region to another is always a factor of two. This is necessary to keep the lattice velocity directions consistent at VR interfaces. The total amount of 78 Million voxels were used. The smallest voxels near solid boundaries and inside the shear layer were $3.7 \times 10^{-4} \text{ m}$ in size. A high-viscosity sponge zone was applied in the outermost ($i.e. \text{ coarsest}$) fluid region in order to damp the outgoing acoustic waves.
It was found that a sponge zone with thickness of 5D_j was sufficient to mimic anechoic conditions at outlet boundaries of computational domain. Test cases were selected for fixed bypass ratio 3 and adjusted pressure ratio to meet the stability limits of 19-atages LBM. Test cases are listed in Table 2.

### TABLE 1. Geometric specification of mixers.

<table>
<thead>
<tr>
<th>Description</th>
<th>Mixer ID</th>
<th>Penetration</th>
<th>Mixing Length</th>
<th>Area Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Confluent</td>
<td>CONF</td>
<td>N/A</td>
<td>0.88</td>
<td>2.34</td>
</tr>
<tr>
<td>12 lobe with low penetration</td>
<td>12 CL</td>
<td>0.48</td>
<td>0.79</td>
<td>2.34</td>
</tr>
</tbody>
</table>

### TABLE 2. Approximate operating conditions.

<table>
<thead>
<tr>
<th>Case number</th>
<th>M_J</th>
<th>NPRf</th>
<th>NPRc</th>
<th>NTR</th>
<th>BPR</th>
<th>Attributes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.45</td>
<td>1.22</td>
<td>1.18</td>
<td>1.01</td>
<td>3</td>
<td>Cold – CONF</td>
</tr>
<tr>
<td>2</td>
<td>0.45</td>
<td>1.22</td>
<td>1.18</td>
<td>1.01</td>
<td>3</td>
<td>Cold – 12CL</td>
</tr>
</tbody>
</table>

* Net total pressure ratio(P_Tf/P_a) for fan stream;
† Net total pressure ratio(P_Tc/P_a) for core stream;
‡ Net total temperature ratio(T_Tf/T_Tc).

### MEAN THRUST CALCULATION

The net average thrust was evaluated for two mixers in order to study the thrust increase potential of lobed mixers. Thrust values (T) were recast as thrust coefficients (C_T) in which the net propulsion force is normalized by dynamic pressure based on the mean outlet jet velocity. The effect of excess outlet pressure was considered as well.

\[
T = \iint \rho_j \overline{U_j} \overline{\vec{U}} \ dA + A_j (P_j - P_a),
\]

(6)

\[
C_T = \frac{2T}{\rho_j \overline{U_j}^2},
\]

(7)

where \( \rho_j \) is jet density, \( \overline{U_j} \overline{\vec{U}} \) is streamwise velocity normal to the nozzle cross section. \( A_j \) is the nozzle exit surface area. \( P_j \) and \( P_a \) are jet and ambient absolute pressure.
FAR-FIELD SOUND

Far-field sound pressure levels were calculated using a modified porous Ffowcs Williams-Hawkings (FWH) surface integral acoustic method developed by Najafi-yazdi et al. This methodology, called Formulation 1C, is included in PowerACOUSTICS 2.0a package licensed by Exa Corporation. This method includes corrections for mean flow, moving sources and observers and is highly useful for cases with a mean free stream such as for engine simulations at flight conditions. A conical, open-ended control surface was surrounded the flow field such that there was no interaction between the jet shear layer and the FWH surface. A total of twenty five virtual microphone locations were defined for the FWH solver at a fixed radial distance of 14.6 m (i.e. $r=160 \ r_0$) from the nozzle exit, covering sufficient directional angles upstream and downstream of the nozzle exit plane (i.e. $45^\circ \leq \theta \leq 160^\circ$) as shown in Fig. 6. The sound pressure levels were calculated for different frequency bands and integrated from 100 Hz to the 8 kHz (i.e. grid cut-off frequency) to obtain overall sound pressure levels for selected directional angles.

RESULTS AND DISCUSSION

Instantaneous flow structures, flow separation and mixing layers inside the internal mixing nozzles were visualized and quantified. Fig. 4-a shows mean vorticity contours at a distance of $1D_j$ from the nozzle outlet. Streamwise vortices induced by the lobed mixer configuration are clearly visible. The instantaneous velocity profile in the streamwise view is depicted in Fig. 4-b). A thick shear layer forms as the jet plume evolves in the downstream direction. Turbulent coherent structures are formed in the vicinity of the nozzle outlet for 12CL mixers. Such structures are absent in the confluent configuration and seems to originate from rotational mixing induced by lobed geometry. Figure 5 shows the mean velocity profile at different locations inside the nozzle for both mixers. The lobed shape mixing zone in 12CL model led into more uniform mixing as could be observed from Fig. 5-a. This is unlike the confluent mixer which caused abrupt mixing and flow deceleration in the bypass channel followed by the acceleration in through converging main channel.

![FIGURE 4. a) Streamwise vorticity contours b) Streamwise instantaneous velocity contours for 12CL model.](image)

Thrust coefficients were obtained using Eqs. 6 and 7. For the case of bypass ratio 3, $C_T = 2.10$ for the 12CL and $C_T = 1.96$ for confluent model. A 7% enhancement in thrust was obtained when the lobed mixer was used. Far-field OASPL (Fig. 6) were calculated using FWH analysis. The directivity of the radiated sound shows a noise reduction (~ 2dB) for almost all directional angles when lobed mixers were used. The peak sound pressure level was shifted 2 degree towards upstream by using 12CL model. The directional angles were measured from nozzle inlet. The peak value around 140° is consistent with values previously observed in experimental studies.
FIGURE 5. Mean velocity profiles for different mixers along a) centerline (from tip of the center-body to exit plane) b) from inlet to outlet through the Nozzle lip-line for two mixer models (12CL —) and (Confluent — — —).

FIGURE 6. a) Sound pressure field in the vicinity of the nozzle b) the Overall sound pressure level at the distance of $R = 160 r_0$ from the nozzle exit for 12CL (solid red line) and confluent (solid blue line).

CONCLUSION

The sound produced by flow through internal mixing nozzles was simulated using LBM. In agreement with experimental data, the use of lobed mixers in comparison with confluent mixers enhanced mixing and improved the net thrust in takeoff condition. Moreover, lobed mixers decrease low frequency and increase high frequency noise levels. Good mixer design may lead to decrease far-field OASPL. Current LBM simulations are restricted to Mach number values smaller than 0.5. Simulations for higher and more practical Mach numbers will be considered in the future using modified high-order LBM algorithms.
ACKNOWLEDGMENTS

The authors would like to thank the Green Aviation Research and Development Network (GARDN), an industry-led National Center of Excellence in Canada and Pratt & Whitney Canada for financing the project. We extend our appreciation to Exa Corporation for providing academic licenses for PowerFLOW and for their continuing technical support. Computations were performed on Colosse and Mammouth supercomputing machines under the auspices of Calcul Québec and Compute Canada.

REFERENCES


