Noise Prediction of a Subsonic Turbulent Round Jet using the Lattice-Boltzmann Method

Phoi-Tack Lew∗, Anastasios Lyrintzis†

School of Aeronautics & Astronautics, Purdue University, West Lafayette, IN 47907, USA

Bernd Crouse‡, Gana Balasubramanian§, David Freed¶

EXA Corporation, 3 Burlington Woods Dr., Burlington, MA 01803, USA

Luc Mongeau∥

Department of Mechanical Engineering, McGill University, Montreal, QC H3A 2K6, Canada

The Lattice-Boltzmann Method (LBM) was used to study the far-field noise generated from a Mach, \( M_j = 0.4 \), unheated turbulent axisymmetric jet. A commercial code based on the LBM kernel was used to simulate the turbulent flow exhausting from a pipe which is 10 jet radii in length. Near-field flow results such as jet centerline velocity decay rates and turbulence intensities were in agreement with experimental results and results from comparable LES studies. The predicted far field sound pressure levels were within 2 dB from published experimental results. Weak unphysical tones were present at high frequency in the computed radiated sound pressure spectra. These tones are believed to be due to spurious sound wave reflections at boundaries between regions of varying voxel resolution. These “VR tones” did not appear to bias the underlying broadband noise spectrum, and they did not affect the overall levels significantly. The LBM appears to be a viable approach, comparable in accuracy to large Eddy simulations, for the problem considered. The main advantages of this approach over Navier-Stokes based finite difference schemes may be a reduced computational cost, ease of including the nozzle in the computational domain, and ease of investigating nozzles with complex shapes.

Nomenclature

Roman Symbols

\( a_j \)  Jet centerline speed of sound
\( c_i \)  Particle speed
\( f \)  Frequency (Hz)
\( f_i \)  Distribution function
\( \mathcal{F}_i \)  Equilibrium distribution function
\( D_j \)  Jet diameter
\( M_j \)  Mach number = \( U_j/a_j \)
\( OASPL \)  Overall sound pressure level
\( p \)  Pressure
\( r_o \)  Initial jet radius
\( \mathcal{R} \)  Radial arc length from centerline jet exit
\( Re_D \)  Jet Reynolds number = \( U_j D_j/\nu_j \)
\( St \)  Strouhal number = \( f D_j/U_j \)

∗Graduate Research Assistant, Student Member AIAA.
†Professor, Associate Fellow AIAA.
‡Senior Aeroacoustics Engineer, EXA Corporation, Junior AIAA Member.
§Aeroacoustics Engineer, EXA Corporation, Junior AIAA Member.
¶Director for Aeroacoustics Applications, EXA Corporation.
∥Professor, Senior Member AIAA.

Copyright © 2007 by P. Lew, A. Lyrintzis, B. Crouse, G. Balasubramanian, D. Freed and L. Mongeau. All other rights are reserved by the copyright owner.
Here is a need to better understand the noise generation mechanisms in turbulent sub-sonic jets. Over the past several years, airports have implemented stricter regulations for aircraft with high jet noise emissions. Aircraft noise may be detrimental to the communities surrounding airports. For example Hygge et al.\textsuperscript{1} reported that primary schoolchildren who live in the vicinity of airports and are routinely exposed to aircraft noise may exhibit deficits in reading perception, long-term memory and speech perception. The same study reports a decrease in impairment when noise is lowered below a threshold value, or when the airport is closed. Aviation is a vital part of the development and expansion of the economy. In the United States, NASA stated the goal in 1997 of reducing the perceived noise levels of future subsonic aircraft by a factor of two (10 EPNdB) by 2007, and by a factor of four (20 EPNdB) by 2022.\textsuperscript{2} This goal is challenging due to the fact that the underlying mechanisms/sources that cause aircraft noise are always well understood and, therefore, cannot yet be fully controlled or optimized. Jet noise is believed to be the dominant contributor to aircraft noise at takeoff. Noise reduction at the source requires a deep understanding of the turbulent flow processes responsible for the generation of sound radiated in the surrounding environment. Jet noise, however, remains one of the most elusive problems in aeroacoustics due to the complexity of the flow-generated sound processes.

Despite significant advances in experimental methods such as PIV, sufficiently detailed experiments to characterize turbulent shear stresses over the entire plume of an aircraft engine jet may not be possible in the foreseeable future. Computational simulations have thus been the primary tool for recent jet noise sound generation studies, despite notable limitations for practical aircraft jet noise problems. Most numerical simulations methods involve the solution of some form of the basic equations of motion using finite difference schemes. With the continuous improvements in computing power, the application of Direct Numerical Simulation (DNS) methods to jet noise prediction is now feasible in some cases.\textsuperscript{3,4} The approach involves the simulation of the flow dynamics for all the relevant turbulence length scales; it requires no turbulence model. The wide range of time and length scales present in turbulent flows and the limitations of current computational resources hampers the use of DNS for high Reynolds number flows. Large Eddy Simulation (LES) involve direct computations of the large scales, in conjunction with subgrid scale models. It is assumed that the large scales in turbulence are generally more energetic compared to the small scales and are affected by the boundary conditions directly. In contrast, the small scales are more dissipative, weaker, and tend to be more universal in nature. Most turbulent jet flows that occur in experimental or industrial settings are at high Reynolds numbers, usually greater than 100,000. LES methods for high Reynolds number flows require a fraction

\begin{itemize}
  \item \( T \) Lattice temperature
  \item \( t \) Time
  \item \( U_j \) Jet streamwise centerline velocity
  \item \( w_i \) Weight parameter for lattice model
  \item \( x, y, z \) Cartesian coordinates
  \item \( x \) Cartesian coordinate of fluid particle
  \item \( x_c/r_o \) Potential core length
\end{itemize}

\textbf{Greek Symbols}
\begin{itemize}
  \item \( \Delta t \) Time resolution
  \item \( \Delta r^+ \) First grid point oﬀ the wall in wall units
  \item \( \epsilon \) Dissipation
  \item \( \Theta \) Observer angle relative to centerline jet axis
  \item \( \kappa \) Turbulent kinetic energy
  \item \( v_j \) Jet centerline kinematic viscosity
  \item \( \rho(x,t) \) Zeroth order moment or density of fluid
  \item \( \rho u(x,t) \) First order moment of velocity of fluid
  \item \( \tau \) Relaxation time
\end{itemize}

\textbf{Subscripts}
\begin{itemize}
  \item \( i \) Direction of particle in a lattice model
  \item \( j \) Jet exit condition
\end{itemize}
of the cost of DNS. One of the first uses of LES as an investigative tool for jet noise prediction was carried out by Mankbadi et al. They performed a simulation of a low Reynolds number supersonic jet and applied Lighthill’s analogy to calculate the far-field noise. Lyrintzis and Mankbadi have used Kirchhoff’s method with LES to compute the far-field noise. Other numerical studies (e.g. References 8–13) were then carried out by investigators at higher Reynolds numbers. In general, these have been found to be accurate, and in good agreement with experimental results. However, the aforementioned simulations lack the inclusion of a nozzle in the computational domain, which does not allow possible dipole contributions from the nozzle surfaces to be included. Instead, ad hoc inflow conditions which typically include random Gaussian or pipe flow simulation output data as forcing are specified to mimic the nozzle exit plume. Although the exclusion of the nozzle reduces computational costs, inflow forcing tends to result in higher noise levels in the far-field compared to experiments. The inclusion of the nozzles in LES simulations is rather recent and the works of Andersson et al., Paliath and Morris, Schur et al. and Uzun and Hussaini are the most notable. The simulation results obtained following the inclusion of the nozzle geometry do improve the far-field noise prediction but at the expense of computational cost. Even if the computational expense accrued with the addition of the nozzle is acceptable, the setup for these simulations includes the arduous task of body-fitted meshing for complex geometries. Thus, despite recent progress in computational aeroacoustics, detailed LES studies remain largely confined to academic jet configurations, for Reynolds number values that are low relative to that of the actual flows of interest. It is also worth noting that computational cost is exacerbated for the case of low Mach number flows due to a smaller time steps requirement.

In general, numerical simulations rely on solutions of the macroscopic Navier-Stokes equations. Recent advances have been made in kinetic based methodologies such as the lattice-Boltzmann method (LBM). These methods have been shown to be accurate for the simulation of complex fluid phenomena. While the Navier-Stokes equations solve the macroscopic properties of the fluid explicitly, LBM’s involve the solution of the time history lattice-Boltzmann equation (LBE) by explicitly tracking the development of particle distribution functions either at the mesoscopic or the microscopic scale. Through the use of the Chapman-Enskog expansion, the LBE has been shown to recover the compressible Navier-Stokes equation at the hydrodynamic limit. The conserved variables such as density, momentum and internal energy are obtained by performing a local integration of the particle distribution. The LBM has recently been applied to a number of canonical problems, including flows over airfoils and cylinders, flow over rectangular cavities, and most recently flows in the micro-scale regime.

The question posed in the present study is whether or not LBM is amenable to the problem of sound radiation from turbulent jets. The use of LBM for acoustics is rather recent, with most applications limited to Mach numbers below $M = 0.2$. Among the recent works pertaining to aeroacoustics are by Ricot et al. and Crouse et al. on fundamental acoustic studies, Crouse et al. on automobile interior noise, and Seror et al. on the noise generated by a detailed-geometry aircraft landing gear. Yu and Girimaji applied an LBM-LES technique to study several low aspect-ratio rectangular turbulent jets. They reported good agreement with experimental measurements in terms of flow statistics, but their study did not include far-field sound. A cursory survey of the literature suggests that the use of LBM has not yet been attempted for the study of jet noise.

The aim of this study was to investigate the far-field noise of a nearly compressible turbulent jet ($M_j = 0.3$ to $M_j = 0.4$) using the lattice-Boltzmann method. The near-field flow physics and far-field noise simulations were performed using a commercial code PowerFLOW 4.0c, which is based on the LBM kernel. This paper is arranged as follows: Section II provides a brief description of the LBM methodology; Section III summarizes the grid setup and flow conditions used for LBM and LES; Section IV provides results for both the near and far-field flow variables and finally some closing remarks and future work are included in Section V.

II. **Brief Description of the Lattice-Boltzmann Methodology**

The lattice-Boltzmann equation has the following form:

$$f_i(x + c_i \Delta t, t + \Delta t) - f_i(x, t) = \frac{\Delta t}{\tau} (f_i(x, t) - F_i(x, t)), \quad (1)$$

where the distribution function $f_i(x, t)$ yields the number density of kinetic particles at position, $x$, with a particle velocity $c_i$ in the $i$ direction at time $t$. The left hand side of (1) computes the particle advection from one center cell to another whereas the right hand side of (1), known as the collision operator, represents the relaxation of the particles. The Bhatnagar-Gross-Krook (BGK) approximation is used to relax the equilibrium distribution function $F_i(x, t)$. The relaxation time $\tau$ however, is related to the kinematic viscosity, $\nu$, such that $\tau = (\nu + \Delta t)/T$. This relation is also commonly referred to as Single Relaxation Time (SRT). The conservative macroscopic variables such as density and
momentum density, are obtained through the zeroth and first order moments of the distribution function, i.e.

\[ \rho(x, t) = \sum_i f_i(x, t), \quad \rho \mathbf{u}(x, t) = \sum_i c_i f_i(x, t). \]  

The pressure is obtained using the equation of state for an ideal gas with the assumption that the gas constant is taken to be unity. This can be expressed as \( p = \rho T \). In addition, the LBM approach recovers the compressible, viscous Navier-Stokes equation in the hydrodynamic limit for wavelengths \( \lambda \gg \Delta x \) and frequencies \( f \ll \Delta t/\tau \). To recover the macroscopic hydrodynamics, \( \mathcal{F}_i(x, t) \) must be chosen in such a way that the essential conservation laws are satisfied and the resulting macroscopic equations are Galilean invariant. In the three-dimensional situation, one of the common choices is the D3Q19 model and this is shown in Figure 1 with

\[ \mathcal{F}_i = \rho w_i \left[ 1 + \frac{c_i \cdot \mathbf{u}}{T} + \frac{(c_i \cdot \mathbf{u})^2}{2T^2} - \frac{\mathbf{u}^2}{2T} + \frac{(c_i \cdot \mathbf{u})^2}{6T^3} - \frac{c_i \cdot \mathbf{u}^2}{2T^2} \right], \]  

where \( w_i \) have the weighting parameters of 1/18 in the 6 coordinate directions, 1/36 in the 12 bi-diagonal directions and 1/3 for the ‘rest’ particle. \( T \) is the lattice temperature which is set to 1/3 for isothermal simulations. The LBM used in this study, has been shown to be both second order accurate in time and space. To account for the presence of solid boundaries in the simulation, the no-slip boundary condition by utilizing a simple particle bounce back and specular reflection process on a solid surface. In addition, an improved volumetric boundary scheme for arbitrary geometries has been devised and implemented to accurately control and govern the momentum flux across the boundary. Further details regarding the handling of solid geometries can be found in references [26, 35, 37].

To account for the unresolved turbulent scales, an eddy viscosity turbulence model was used. Specifically, PowerFLOW uses the two-equation \( k-\varepsilon \) Renormalization Group (RNG) turbulence model to compute the turbulence viscosity with the addition of a swirl corrector to model part of the large scale structures. This methodology is also commonly referred to as Very Large Eddy Simulation (VLES). Preliminary simulations with the turbulence model yielded a laminar jet, i.e. the plume exiting the pipe did not exhibit break-up beyond the potential core. Thus no turbulence model was used. The simulations were carried on using an under-resolved grid and no subgrid scale model. This procedure has been argued to be analogous to an LES.

The potential advantages of LBM over the conventional Navier-Stokes solvers include

1. Linearity of the convection operator (Equation (1)) due to the kinetic nature of the LBE method. In addition to particle convection and collisions, the use of multi-scale expansions allows the recovery of the nonlinear macroscopic advection process.
2. Direct obtention of the strain rate from the non-equilibrium distribution function. From the microscopic / mesoscopic point of view of LBM, the Poisson term (e.g. strain and rotation rate tensor) in the coarse grained Navier-Stokes equations often offer numerical difficulties in terms of accuracy for finite-difference based algorithms.
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 scapped 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.

The LBM approach used in this study is not without drawbacks. The most notable disadvantage is that the LBM does not recover flow physics correctly for cases with high Mach numbers (\( M > 0.4 \)). This limitation is due two main factors:

1. The discrete velocity model (D3Q19) is insufficient to span the particle phase space so that important moments that account for thermal energy transfer are not be correctly recovered.
2. The collision integral assumes a low-speed approximation.

Efforts are currently underway to extend the current LBM for higher Mach number jet flows. Recently, Sun and Hsu used an LBM technique to study a shock tube problem and obtained good results compared to the Reimann solution. Shan et al. and Chen et al. have laid a firm theoretical groundwork to efficiently extend the LBM to higher Mach numbers and arbitrary Knudsen numbers. Recently, Li et al. devised a modified Boltzmann equation and applied it to a 2-D aeroacoustic benchmark problem. They obtained good results and showed that their methodology is valid up to a Mach number of 0.9.
III. Computational Setup

A. Lattice-Boltzmann

A pipe with a length of \( L = 0.508 \) meters and a diameter of \( D_j = 0.0508 \) meters is considered as a part of the computational domain, starting at \( x = 0 \). The pipe diameter was chosen to match the jet diameter of the experiments of Tanna. The centerline of the pipe is along the \( x \)-axis and the \( y \) and \( z \) are lateral axes, respectively. The addition of a nozzle in the computational domain was intended to eliminate the need for an artificial forcing mechanism to trip the flow. Artificial forcing techniques used in many LES simulations (See References [47, 48]) may cause spurious far-field sound pressure levels (SPL). As discussed later, however, computational resources did not allow a very fine grid within the pipe.

The computational domain is partitioned into several variable resolution (VR) regions, in order to tailor the grid as needed to resolve the flow details, and reduce computational costs. This methodology is similar to grid stretching techniques typically employed in traditional CFD. Figure 2 shows a side view of the entire computational domain used in this study. Successive VR regions are concentric and cylindrical, as shown in Figure 3. The outer most bounding region is rectangular. From Figure 2, it is important to provide ample spacing between each VR region radially and in the streamwise direction. An initial simulation with no streamwise spacing between VR regions showed that huge “VR tones” were present in the far-field pressure spectra. These tones were as high as 15 dB and appeared to bias the overall spectral makeup. Hence, providing ample streamwise spacing between VR regions actually eliminated these tones in the lower frequency portion but are somewhat present in the high frequency range. Each grid cell is called a “voxel”.

Hence, each VR region represents one grid resolution level and the VRs cascade outwards from the fine resolution region towards the coarse resolution region. The voxel cell size between each successive VR region differs by a factor of 2. The domain includes a total of 20 million voxels. The smallest voxels, of size approximately \( 8 \times 10^{-4} \) meters (0.8 mm), are located in the shear layer region of the pipe. The voxel size corresponds to approximately \( \Delta r/D_j \simeq 0.016 \) which is considered very coarse for wall bounded flow studies. The ratio needed to resolve the duct boundary layers is at least one order of magnitude less without the implementation of a wall model. This was deemed prohibitively expensive. The first grid point off the wall is at \( \Delta r^+=7.7 \). Although the adopted cell size did not resolve the boundary layer details, it was sufficiently small to supply physical jet inflow conditions without the need for artificial forcing techniques.

The entire simulation domain includes a total of 11 VR regions with a domain size set of \((x, y, z)=(250 \ D_j, \pm 125 \ D_j, \pm 125 \ D_j)\). These values, which may seem large at first, were chosen to minimize reflections of the propagating sound waves and convected jet plume flow structures back into the near-field. To the same end, coarse VR regions further away from the jet dissipate the outgoing traveling waves and thus act as ‘sponge’ zones. Non-reflecting type boundary conditions are applied at the inlet lateral and outlet boundaries. The region where the physical properties are measured and stored is roughly \((x, y, z)=(27.5 D_j, \pm 5 D_j, \pm 5 D_j)\). This region has a funnel type shape, as shown in Figure 4. The inner most funnel shaped VR region is the measurement region for near-field flow statistics. In the core VR region, virtual probes are placed on the jet axis centerline and nozzle lip-line along the streamwise direction for the eventual calculations of turbulent spectral content. The location of the probes is indicated in Figure 4. Additional probes are placed sufficiently far away from the non-linear turbulent region. These probes are used to compute far-field acoustics. The far-field acoustic measurement probes are placed at \( 15r_s \) above the centerline jet axis. Figure 5 shows the voxel setup for the entire domain. Note the very coarse voxels in the outer VR region are larger than the pipe dimensions. Figure 6 shows a close-up view of the voxel distribution in the pipe region whereas Figure 7 shows the voxel distribution at the pipe exit.

We consider no mean density or temperature gradients in the nozzle compared to ambient conditions. To initialize the velocity at the entrance of the pipe, a top-hat velocity profile is specified with \( U_j = 100 \) m/s as the jet centerline velocity. When the simulation is progressed, the jet centerline velocity at the exit of the pipe reaches approximately \( 130 \) m/s or \( \frac{M_j}{a_j} = 0.4 \) due to the formation of a thick boundary layer. As previously mentioned, no turbulence model was used for this test case and an under-resolved DNS (uDNS) was assumed. In order to reasonably resolve the turbulent scales with the available voxel resolution, the kinematic viscosity had to be adjusted. In this case, the Reynolds number achieved at the exit of the pipe is roughly \( Re_p = (U_j D_j)/\nu_j \simeq 6,000 \), implying a kinematic viscosity of around \( 1.1 \times 10^{-3} \) m²/s.

B. Large-Eddy Simulation

The 3-D LES code used for this study, developed by Uzun et al., uses either the classical or a localized dynamic Smagorinsky (DSM) subgrid-scale model. The modeling of the subgrid-scale stress tensor still raises some
fundamental issues as discussed by Bogey & Bailly\textsuperscript{38,39} and Uzun \textit{et al.}\textsuperscript{53} Eddy-viscosity modelings such as the classical Smagorinsky subgrid-scale model\textsuperscript{51} and the localized dynamic subgrid-scale model (DSM)\textsuperscript{52,54} might dissipate the turbulent energy through a wide range of scales up to the larger ones, which should be dissipation free at sufficiently high Reynolds numbers.\textsuperscript{55} In addition, since the eddy-viscosity has the same functional form as the molecular viscosity the effective Reynolds number is reduced in the simulated flows.\textsuperscript{56} See References [38] and [53] for a thorough analysis and discussion on the shortcomings of eddy viscosity subgrid-scale model on jet flows. An alternative to the use of an explicit eddy-viscosity model is the use of spatial filtering for modeling the effects of the subgrid-scales. Using this alternative, the turbulent energy is only dissipated when it is transferred from the larger scales to the smaller scales discretized by the mesh grid.\textsuperscript{38} For the isothermal jet simulated here, spatial filtering was used as an implicit subgrid-scale model.

The unsteady, Favre-filtered, compressible, non-dimensional LES equations were solved. Transformation from curvilinear coordinates to a uniform grid was done in computational space. The non-dissipative sixth-order compact scheme developed by Lele\textsuperscript{57} was used to compute the solution on the internal points. For the points on the boundaries, a third-order one-sided compact scheme was used. The points next to the boundaries were computed using a fourth-order compact central differencing technique. In order to eliminate numerical instabilities that can arise from the boundary conditions, grid non-uniformities and unresolved scales the sixth-order tri-diagonal spatial filter proposed by Visbal and Gaitonde\textsuperscript{58} was employed with the filter parameter set to \( \alpha_f = 0.47 \). For time advancement, the explicit fourth-order Runge-Kutta scheme was used. Tam and Dong’s 3-D radiation and outflow boundary conditions\textsuperscript{59} are implemented on the boundaries. In addition, a sponge zone\textsuperscript{60} is attached to the end of the computational domain to dissipate the vortices present in the flow field before they hit the outflow boundary. This minimizes unwanted reflections from the outflow boundary.

To excite or trip the flow, randomized perturbations in the form of induced velocities from a vortex ring proposed by Bogey & Bailly\textsuperscript{61} are added to the velocity profile at a short distance (approximately one jet radius) downstream from the inflow boundary (see Figure 8). This is done to ensure the break up of the potential core within a reasonable distance.

The Mach number used here is also \( M_j = 0.4 \). The total number of grid points used is roughly 5 million with a grid point allocation consisting of \( N_x = 292, N_y = 128 \) and \( N_z = 128 \) in the \( x, y \) and \( z \) directions, respectively. Figure 9 shows the cross sectional plane at \( z = 0 \) of the computational domain. The physical part of the computational domain extends to approximately \( 60\,r_o \) in the streamwise direction and \( -20\,r_o \) to \( 20\,r_o \) in the transverse \( y \) and \( z \) directions. Beyond the streamwise location of \( 60\,r_o \), the sponge zone is the sponge zone. The physical domain length of \( 60\,r_o \) was chosen for two reasons. Firstly, Uzun \textit{et al.}\textsuperscript{62} reported that the Reynolds stresses achieve their full asymptotic self-similar state if a domain length of at least \( 45r_o \) is used. Secondly, in order to capture the overall sound pressure levels (OASPL) adequately at shallow angles, a domain length of at least \( 55\,r_o \) is required based on the recommendations of Uzun \textit{et al.}\textsuperscript{62} and Shur \textit{et al.}\textsuperscript{63} Based on the minimum grid spacing and ambient Mach number, the time resolution was determined to be \( \Delta t = 0.01\,r_o/a_{\infty} \).

The Reynolds number for the LES test case was \( Re_p = \rho_j U_j D_j/\mu_j = 6,000 \), i.e. the same as for the LBM case. The vortex ring used included up to 16 azimuthal jet modes. Bogey and Bailly\textsuperscript{12} performed a simulation with all modes present and later removed the first four modes and found that the jet was quieter for the latter case, in better agreement with experimental results. Accordingly, the first four azimuthal modes of forcing were not included in the forcing. The forcing amplitude is \( \alpha = 0.007 \).

Since there is no nozzle-lip in the simulation domain, the hyperbolic tangent velocity profile was used, see Freund,\textsuperscript{63} on the inflow boundary given by

\[
\tilde{\rho}(r) = \frac{1}{2} U_j \left[ 1 - \tanh \left( b \left( \frac{r - r_o}{r_o} \right) \right) \right],
\]

where \( r = \sqrt{x^2 + z^2}, r_o = 1, \text{ and } U_j \) is the jet centerline velocity. The parameter that controls the thickness of the shear layer is \( b \). This parameter was \( b = 2.8 \). A higher value of \( b \) implies a thinner shear layer. For comparison, Freund\textsuperscript{63} used a value of \( b = 12.5 \) for his 3-D jet DNS. Hence, a thicker shear layer is expected here compared to Freund’s jet. For laboratory jets however, the measured value of \( b \) is usually an order of magnitude higher compared to that used in LES and DNS of jets. Considering an isothermal jet, the Crocco-Busemann density is accordingly specified as a function of the jet Mach number and inlet velocity profile.
IV. Results

The physical time scaling or time step for the LBM is $1.45 \times 10^{-6}$ seconds. A total of 400,000 time steps was required to achieve reasonably converged statistics for flow analysis. In terms of computational resources used, this test case took approximately two days of runtime using 128 processors in parallel on a Dell Xeon cluster (See Acknowledgements). In comparison, the LES simulations took approximately 3.5 days of run time using 128 processors on the same machine.

A. Near-field Flow Variables

Figure 10 shows a close-up view of the instantaneous velocity magnitude filled contours of the jet. This is a slice along the jet centerline or on the $z = 0$ plane. The shear layers appear to be relatively thick. The shear layer becomes unstable at a location approximately two jet diameters from the exit. Figure 11 shows a contour plot of the mean axial velocity on the $z = 0$ plane. The mean streamwise velocity on the jet centerline is plotted in Figure 12. The variation of the streamwise velocity with distance, $\frac{U_j}{U_{j,0}}$, for the LBM compares very well with the velocity decay data measured from Bridges & Wernet for a Mach 0.5 free jet. Here, $U_j$ is the jet exit velocity and $U_j(x)$ is the local mean streamwise velocity on the centerline. The LES data shows a slightly faster decay compared to the experiments of Bridges & Wernet but nonetheless reasonable. Figure 13 shows the inverse of Figure 12 and from here we see that the measured decay slope of 0.155 from the LBM is close to experimental correlation of 0.16 from Zaman. The LES calculation is 0.163 which seems to be closer to the experiments even without influence of a nozzle. Although not shown in the Figure 13, the measured decay rate of Bridges & Wernet’s experiment is close to 0.15. The potential core length, $x_c/r_o$, is defined when the jet mean centerline velocity reduces to 95% of the inflow jet velocity, $U_j(x_c) = 0.95U_j$. In the present simulation, a potential core length of 12$r_o$ and 11.4$r_o$ was obtained for the LBM and LES, respectively. This value is within the range of core lengths, between 10 and 14 jet radii, typically observed in laboratory experiments.

Figure 14 shows a filled contour plot of the mean axial turbulence intensity for the simulated jet. Note the small axial intensity near the nozzle lip, which indicates a nearly laminar exit shear layer. Figures 15 and 16 shows the axial turbulence intensities along the centerline and lip-line of the jet for both LBM and LES. The simulation results are also compared to recent experimental measurements of Laurendeau et al.66,67 for a $M_f = 0.3$ isothermal jet. Qualitatively, the trends of the axial turbulence intensities for this jet are consistent with those of an axisymmetric turbulent jet; the peak fluctuation for the centerline occurs earlier and is greater compared to the centerline peak fluctuation. The decay after the peak intensity is reached is shown to be slightly slower for the LBM compared to the LES. The peak value and the axial location of the computed lip-line turbulence intensity are in agreement with experimental observations. For the range $0 \leq x/r_o \leq 9$, the simulation results are much lower than the experimental data. The lower values may be due to the fact that the simulated exit shear layer is still laminar or transitional at best. The LES shows about a 2% turbulent intensity at the exit which could be a consequence of the vortex ring to excite the mean flow. Recall that there is no artificial excitation used in the LBM jet. The Reynolds number of the simulated jet, 6,000, is much smaller than the Reynolds number of the measured jet, which was $300,000$, due to the need for an increased viscosity. The experiments were performed using a converging nozzle rather than a straight pipe. Nonetheless, according to Laurendeau,68 the initial high turbulence intensities measured along the lip-line in the experiments could be due to the surrounding air not being seeded and a possible influence on how the boundary layer is tripped. But nevertheless the high intensity at the nozzle exit appears to be very plausible. A recent high resolution, well-resolved LES study performed by Uzun and Hussaini18 shows that the axial turbulence intensity at the jet exit could reach as high as 14% depending on how close one measures the intensity near the wall. The centerline mean axial turbulence intensity on the other hand shows a better agreement compared to the experiments of Laurendeau et al.67 This time the difference is not as pronounced as for the lip-line with a good agreement in the range of $10 \leq x/r_o \leq 15$. Again, note a slight under-prediction compared to the experiments from the exit to approximately $9r_o$. Note that unfortunately, no experimental data was collected beyond $x = 15r_o$ due to the limited size of the PIV window.

Figure 17 shows the axial velocity spectra at four different locations ($x = 20r_o, 25r_o, 30r_o$ and $35r_o$) downstream of the jet along the nozzle lip-line and centerline. Figure 17 shows that the jet development is indeed broadband. This is further substantiated by the fact that a portion of the spectra decays according to Kolmogorov’s well known -5/3 law, indicating that part of the spectra falls in the inertial subrange (equilibrated turbulence) before dropping-off at higher frequencies. Based on the spatial grid resolution, the maximum resolvable Strouhal number ($Sr = fD_j/U_j$) in this nonlinear region is approximately $5r = 3$ and this is also reflected in the simulations. Although not shown here, the axial velocity spectra for the station before the break-up of the potential core, $x = 5r_o$, did not exhibit the -5/3 decay. Instead, there was a concentration of energy in the low frequency region before the transfer of energy (sharp drop) to higher frequencies. From Figure 17(a), we observe that the LBM spectra within a reasonable range with the
LES calculations. The apparent sharp drop of the LES compared to the LBM is probably due to the relatively coarse grid of the LES. Nonetheless, in both simulations the inertial range is captured. Laboratory measurements of axial pressure spectra along an incompressible free shear layer performed by George et al.\textsuperscript{69} indicate the development of the turbulence-mean-shear contribution and decays according to -$11/3$ in the inertial subrange. Figure 18 shows the pressure spectrum in the shear layer at $x = 5r_o$, well before the break-up of the potential core. Here the -$11/3$ decay as measured experimentally by George et al. is reflected in the LBM simulation. The decay is shown quite clearly for the range $2 \leq Sr \leq 5$. However, there is a build-up of energy starting around $Sr = 6$ and then drops off again as shown in Figure 18. The -7/3 decay was also measured by George et al.\textsuperscript{69} and is once again reflected in this simulation shown in Figure 19. The near-field pressure spectra for the LES is also plotted alongside the LBM computations for station $x = 20r_o$ on the shear layer. The LES at around the range $0.5 \leq Sr \leq 0.8$ which is rather small does at least capture part of the inertial range. For the LBM, although $x = 10r_o$ is located slightly upstream of the potential core, it nonetheless indicates turbulence-turbulence interaction for a broad range of frequencies in the subrange compared to $x = 20r_o$ and $x = 30r_o$, respectively. From the simulation the sharpest roll-off seems to occur at around $Sr = 10$. Overall, based on the spectral characteristics of the near field velocity and pressure, the current simulation has many of the same features as an LES.

### B. Far-field Acoustics

A total of 15 probes are placed at approximately $15r_o$ from the centerline of the jet (as shown in Figure 4). Acoustic data were collected at every time step over a duration of 360,000 time steps. Based on the grid resolution at the probe location and assuming at least 6 voxels are needed to resolve one acoustic wavelength, the maximum resolvable frequency is roughly 8,000 Hz or about $Sr = 3.2$. The computed spectra are then extrapolated using a $1/R$ correction to the far-field distance of $R = 144r_o$ similar to the observation distance measured by Tanna et al.\textsuperscript{70} This far-field extrapolation methodology is crude but nonetheless yields a first cut approximation for the far-field noise. A more sophisticated methodology such which includes Doppler effects such the Ffowcs Williams-Hawkings (FWH)\textsuperscript{71,72} integral method is used to study the far-field noise. The integral method follows the description of Lyrintzis & Uzun\textsuperscript{73} and Lyrintzis.\textsuperscript{74} For simplicity, and to allow for a continuous stationary control surface around the turbulent jet is used. For details regarding the numerical implementation of the Ffowcs Williams-Hawkings method, the reader is referred to Uzun.\textsuperscript{82} The control surface starts about one jet radii downstream and is situated at approximately $7.5r_o$ above and below the jet at the inflow boundary in the $y$ and $z$ directions. It extends streamwise until the near end of the physical domain at which point the cross stream extent of the control surface is approximately $30r_o$. Hence, the total streamwise length of the control surface is $59r_o$. We show results for an open control surface. A open control surface here is defined where there is no surface at the end of the physical domain, i.e. $x = 60r_o$. We gather flow field data on our control surface at every 5 time steps for over a period of 55,000 time steps. Based on our grid resolution around our control surface and assuming that with our numerical method 6 points per wavelength are needed to accurately resolve an acoustic wave,\textsuperscript{82} the maximum frequency resolved corresponds to a Strouhal number of $Sr = fD_j/U_j \approx 1.8$. The overall sound pressure levels are computed along an arc with a distance of $R = 144r_o$ from the jet nozzle exit. The angle, $\Theta$, however, is measured relative to the centerline jet axis.

The directivity pattern along an observation arc at $R = 144r_o$ is shown in Figure 20, i.e. the overall sound pressure levels (OASPL) for our $M_j = 0.4$ jet. Our data is compared to the experimental measurements of Tanna et al. and the more recent data of Laurendeau et al.\textsuperscript{67} Here, SP02 simply refers to set point 2 in Tanna’s experimental test matrix. The results of Laurendeau are adjusted based on distance and Lighthill’s $U^3$ power law. In addition to the experimental data shown, we have also included the SAE ARP 876C\textsuperscript{75} database prediction for a jet operating at similar conditions as ours. This database prediction consists of actual engine jet noise measurements and can be used to predict overall sound pressure levels within a few dB at different jet operating conditions. Hence, this prediction is purely empirical. From Figure 20, the LBM computation agree reasonably well with the experimental data. The LES data show an over-
prediction of approximately 2 to 3 dB compared to Tanna’s experiment. Likewise, the LES data is higher compared to the LBM results. We believe that the vortex ring inflow forcing is the probable cause of the over-prediction of the LES compared to the laboratory jet. Nonetheless, there seem to be good agreement at some observation angles whereas at some angles the simulation either over-predicts or under-predicts the data by roughly 1 to 3 dB, depending on which experiments one chooses to observe. The computed far-field noise using LBM seems to yield encouraging results.

Figures 21 through 23 show the far-field spectra at a distance of $r = 144r_0$ with observer angles of $\Theta = 45^\circ$, $60^\circ$, and $75^\circ$, respectively. Tanna did not have spectra available for observation angle $\Theta = 30^\circ$. Again, the cut-off frequency based on spatial grid resolution is $Sr = 3$. As shown in Figures 21 through 23 there are weak but distinct tones present in the spectrum at $Sr = 2$ and $Sr = 4.5$. As for the axial pressure spectra shown in Figure 18, these weak VR tones do not appear to severely contaminate the overall spectrum. The computed spectra using LBM ignoring these spurious tones appear to be accurate. From the spectral plots, it appears that the LES-FWH methodology over-predict Tanna’s experiment in the low frequency region whereas the LBM result under-predicts the experiments in the high frequency region.

V. Closing Remarks and Future Work

A numerical simulation of a Mach 0.4 unheated jet with a pipe installed as a nozzle was performed using the lattice-Boltzmann method (LBM). Turbulent flow near-field statistics were in good agreement with experimental data and parallel LES results. The centerline mean axial turbulence intensity agree reasonably well with the recent experiments of Laurendeau. Although, the peak turbulence intensities that were measured agree well with the results of Laurendeau, the intensities in the shear layer region under-predict the experiments probably due to the laminar nature of the exiting boundary layer. To resolve this issue, a greater Reynolds number and finer grid resolution will be applied in a future study. It was also found that the current LBM methodology is indeed analogous to an LES based on the decay of the axial velocity and pressure spectra following Kolmogorov’s $-5/3$ and $-7/3$ (turbulence-turbulence interaction for pressure) law, respectively. The computed overall sound pressure levels in the far-field agreed with experimental results from Tanna et al. and Laurendeau et al.. The computed far-field spectra agree well with the experiments of Tanna. The LBM results were comparable to the LES-FWH computations. Weak but distinct VR tones appeared in the high frequency portion of the spectra. This anomaly needs be needs to be addressed and eliminated in future studies.

Of course, practical applications require to run LBM simulations at higher Mach numbers, i.e. Mach 0.6 to 2. There is an effort to extend the current lattice-Boltzmann methodology for high Mach number capability. Nonetheless, this study has shown that as a first step towards reaching that goal, the LBM indeed showed promising results for a near compressible jet. Hence, the use of LBM applied to the study of jet aeroacoustics appears to be a viable approach in the field of jet noise simulations.

Acknowledgments

The first author gratefully acknowledges the partial support of the Purdue Research Foundation (PRF) Special Incentive Research Grant (SIRG) and sponsorship from Exa Corporation. The jet simulations presented in this paper were carried out on a 1,152 processor Dell cluster Mammoth at Centre de Calcul Scientifique which is operated by Université de Sherbrooke, Quebec, Canada. We would also like to thank Professor Nourredine Atalla from Université de Sherbrooke for providing some of the resources on Mammoth. The first author would like to thank Dr. Estelle Laurendeau in providing the experimental data for the low speed jet. Finally, the first author would also like to thank Dr. Yanbing (Leo) Li for some useful discussions.

References


Figure 1. D3Q19 LBM model (Image scanned from Reference 25).

Figure 2. Variable resolution (VR) region setup for current test case.
Figure 3. A different view of the entire VR domain setup. Note that each VR region are concentric cylinders except for the outer most boundary.

Figure 4. A close-up view of the VR region \( z = 0 \) plane) where the jet pipe is installed. Cross like symbols indicate probes/microphones placed in the turbulent region and far-field region.
Figure 5. Voxel setup for entire computational region. Section taken on the \( z = 0 \) plane.

Figure 6. A close-up view in the \( z = 0 \) plane of the voxel/cell concentration near in the pipe region.
Figure 7. Voxel setup at the pipe exit. Section taken at $x = 0.0508$ m plane.

Figure 8. Boundary conditions used in the 3-D LES code.
Figure 9. The cross section of the computational grid for the LES simulation on the $z = 0$ plane. The physical extends to $60 r_o$ (Every 3rd grid point is shown).

Figure 10. Instantaneous snapshot of velocity magnitude flow field from LBM on the $z = 0$ plane.
Figure 11. Mean streamwise velocity contours from LBM.

Figure 12. Mean streamwise velocity decay along the jet centerline axis for numerical simulations compared to the experiments of Bridges & Wernet.
Figure 13. Mean streamwise velocity decay along the jet centerline axis for both LBM and LES.

Figure 14. Mean streamwise turbulence intensity contours from LBM.
Figure 15. Mean axial turbulence intensity along the jet centerline for both LBM and LES. Numerical results are compared to the recent experiments of Laurendeau.67

Figure 16. Mean axial turbulence intensity along the jet shear layer for both LBM and LES. Numerical results are compared to the recent experiments of Laurendeau.67
Figure 17. Streamwise velocity spectra at four different stations located along the shear and centerline of the jet. The LES data is only shown for station $x = 20r_o$. 

(a) $x = 20r_o$

(b) $x = 25r_o$

(c) $x = 30r_o$

(d) $x = 35r_o$
Figure 18. Spectral content of pressure along the shear layer at \(x = 5r_o\).

Figure 19. Spectral content of pressure along the shear layer for LBM at \(x = 10r_o, x = 20r_o\), and \(x = 30r_o\), respectively. The LES data is for location \(x = 20r_o\).
Figure 20. Overall sound pressure level directivity at $R = 144r_o$ with the observation angle $\Theta$ measured relative to the jet centerline axis. (*) indicate that the data from Laurendeau is adjusted based on distance and $U^5$ power law.

Figure 21. One third octave sound pressure level in the far-field at $\Theta = 45^\circ$, $R = 144r_o$.,
Figure 22. One third octave sound pressure level in the far-field at Θ = 60°, R = 144r_∗.

Figure 23. One third octave sound pressure level in the far-field at Θ = 75°, R = 144r_∗.