Lattice Boltzmann Simulation of Industrial Axial Fan Noise

K Hayashi¹, S Nakaye² and S Nishimura³

¹ Research Manager, Vibration Research Department, Research and Innovation Center, Mitsubishi Heavy Industries, Ltd.
² Engineer, Vibration Research Department, Research and Innovation Center, Mitsubishi Heavy Industries, Ltd.
³ Engineer, Fluid Dynamics Research Department, Research and Innovation Center, Mitsubishi Heavy Industries, Ltd.

E-mail: kentaro1_hayashi@mhi.co.jp

Abstract. Noise simulation of an industrial axial fan has been conducted by implicit large-eddy simulation with Lattice Boltzmann simulation which is based on Cumulant Lattice Boltzmann method with wall model and diffuse-interface method for a moving boundary condition. Simulation results are compared with experimental results of a factory test of an industrial fan for a thermal power generation. The result shows a good agreement and overall value is within 3dB error.

1. Introduction
Currently, Noise control of industrial machines becomes more important for natural environment. Thermal power plants have many noise sources, and an industrial axial fan is one of the noise sources of power plants. Therefore acoustic design of the fan duct should be adequate. On the other hand, noise generation from the axial fan is complex, and the simulation of fan noise needs large scale unsteady CFD with low numerical dissipation because sound pressure is quite small compared to a pressure variation of the flow field. Therefore it is important for the accurate aeroacoustic simulation to develop high resolution CFD code with low numerical dissipation.

LBM (Lattice Boltzmann Method) is based on kinetic theory to simulate various hydrodynamic systems. It is suitable for aeroacoustic analyses involving complex objects because of its low dissipation properties and simple algorithms. Recently, LBM has been applied to aeroacoustic problems and unsteady turbulent phenomena, and its effectiveness has been demonstrated [1][2]. However, from the viewpoint of industrial use, LBM still has some problems in applications to wall-bounded high Reynolds number flow and rotating machinery.

Based on the above circumstances, in-house Lattice Boltzmann Method was developed and validated with various applications in the previous study [3]. The code is based on Cumulant Lattice Boltzmann Method with wall model and diffuse-interface method for moving boundary condition. Especially, there are very few studies about diffuse-interface method for rotating machinery at high Reynolds number. In this study, newly developed in-house Lattice Boltzmann Method has been applied to the noise simulation of an industrial axial fan.
2. Computational Approach
A detail of computational approach is shown in previous paper [3]. In this paper, a summary of computational method is shown as follows.

2.1. Lattice Boltzmann Method
D3Q27 model, which can satisfy solutions in axis-symmetry problems, is used. Evolution of distribution functions $f_j$ is discretized as

$$f_j(x + c_j \Delta x, t + \Delta t) = f_j(x, t) + \Omega_j(x, t) \quad j = 1, \ldots, 27$$

(1)

Where $\Omega_j$ refers to a collision operator, $c_j$ is a discrete set of velocities.

Macroscopic variables, such as density $\rho$ and velocity vector $\mathbf{u}$, can be derived from a moment of $f_j$.

$$\rho = \sum_{j=1}^{27} f_j, \quad \rho \mathbf{u} = \sum_{j=1}^{27} f_j c_j$$

(2)

2.2. Computational mesh
The governing equation of LBM was discretized in Cartesian mesh. Therefore, it is easy to mesh for complex geometry, which means low computational costs are required. In our code, BCM (Building Cube Method) framework was applied [4][5][6]. In BCM, flow field is divided into sub-domains, called ‘Cube’ as shown in Figure 1. In each cube, an equally-spacing Cartesian grid is used in Figure 1.

The data communication among cubes is through overlapping cells in Figure 2. In the present code, the number of overlapping cells is 3. A detail is shown in previous paper [3].

![Figure 1 Computational mesh in BCM (Left: Cube, Right: Cell)](image1)

![Figure 2 Stencil for space interpolation in two dimensional case](image2)
2.3. Cumulant collision model
Besides its popularity, the LBM still presents some difficulties that reside in the collision operation $\Omega_j$. Recently, Geier proposed cumulant LBM [7]. It has significantly improved stability and also accuracy compared to MRT method. The cumulant LBM is Galilean invariant and hyper-viscosity is reduced to the same level or lower compared to that in the SRT.

In the cumulant LBM, moments are relaxed toward their equilibrium in cumulant space. Each relaxation process can be modeled for the cumulant with an independent rate of $\omega_j$ [7].

$$C'_j = \omega_j C_{eq} + (1 - \omega_j)C_j$$  \hspace{1cm} (3)

Where $C'_j$ and $C_{eq}$ is the post-collision and the equilibrium state of cumulant. In this paper, we chose all relaxation rates to be unity except for those acting on shear viscosity in all models. Obviously, this choice is not necessarily a good one with respect to the stability and the accuracy of the methods [8][9][10].

It should be noted that simulations were conducted with cumulant collision model as Implicit LES without any explicit turbulence model [8][9][10].

2.4. Wall model
Navier-Stokes based wall model has been widely researched and recently has had great success in improving their ability to predict flow separation [11][12]. However, wall modeling of LBM -LES is still in its very early development stages. In this research, the wall model by Spalding [13] is coupled with interpolated bounced back scheme. This implementation is similar to ones in Ref. [14] and Ref.[15]. A detail of wall model is shown in previous paper [3].

2.5. Diffuse-interface method for moving boundary condition
In the diffuse-interface method, it is assumed that a viscous fluid is filled in both inside and outside of a boundary. Then, body forces are applied on lattice points near the boundary in order to enforce the no-slip condition at the boundary. A scheme for determining the body forces is different between variations of the diffuse-interface method. In this study, Multi Direct Forcing Method is used [16][17].

Note that the moving boundary is represented by $N$ points, and the boundary Lagrangian points $X_k$ generally differ from the Eulerian grid points $x$. Formally, the fluid velocity $u$ at Lagrangian points is interpolated from the Eulerian grid as

$$u(X_k) = \sum_x u(x)W(x - X_k)dx$$  \hspace{1cm} (4)

Where $W$ represents the weight function and is given by

$$W(x, y, z) = \frac{1}{\Delta x} w\left(\frac{x}{\Delta x}\right) \frac{1}{\Delta y} w\left(\frac{y}{\Delta y}\right) \frac{1}{\Delta z} w\left(\frac{z}{\Delta z}\right)$$

$$w(r) = \begin{cases} \frac{1}{8} \left(3 - 2|\mathbf{r}| + \sqrt{1 + 4|\mathbf{r}| - 4r^2}\right), & (|\mathbf{r}| \leq 1) \\
\frac{1}{8} \left(5 - 2|\mathbf{r}| - \sqrt{-7 + 12|\mathbf{r}| - 4r^2}\right), & (1 < |\mathbf{r}| \leq 2) \\
0, & \text{otherwise.} \end{cases}$$  \hspace{1cm} (5)

The body force $g(x)$ is determined by a following iterative procedure. In this research, a number of iteration is 4, where accuracy of a boundary condition and computational efficiency are balanced well.
It should be noted that the body force $g(x)$ acting on the fluid is included in the cumulant collision model [3].

Step 0. Compute an initial value of a body force at boundary Lagrangian points by

$$
g(X_k) = \rho_f \frac{u_w(X_k) - u^*(X_k)}{\Delta t}$$

(6)

where $u^*(X_k)$ is a temporal velocity.

Step 1. Compute the body force at Eulerian grid points of $l$th iteration by

$$
g(x) = \sum_{k=1}^{N} g(X_k) W(x - X_k) \Delta V$$

(7)

Where the body force is not added to one boundary Lagrangian point but a small volume element whose volume is described as $\Delta V$. In this method, $\Delta V$ is a thin shell of thickness equal to one mesh width around each Lagrangian point [18].

Step 2. Correct the velocity at the Eulerian grid point by

$$
u(x) = u^*(x) + \frac{\Delta t}{\rho_f} g(x)$$

(8)

Step 3. Interpolate the velocity at the boundary Lagrangian point with

$$
u(X_k) = \sum_{x} u(x) W(x - X_k) \Delta x^3$$

(9)

Step 4. Correct the body force with

$$
g(X_k) = g(X_k) + \rho_f \frac{u_w(X_k) - u(X_k)}{\Delta t}$$

(10)

and go to Step 1.

2.6. Numerical setup
An axial fan in this study has 2-stage rotors and stators. Duct diameter is 2.4m on average and the rotational speed is 1800rpm. Tip Mach number is 0.5. Reynolds number based on duct diameter is $2.21 \times 10^7$. Schematic diagram is shown in Figure 3.

The computational mesh is shown in Figure 4. Minimum grid size is 4.8mm around the blades. A tip clearance is resolved with 2–3 cells, with which aeroacoustic effect at the tip can be roughly captured. The total simulation time is 0.323sec and the frequency resolution $\Delta f$ in this study is 8 Hz. OASPL (Overall sound pressure level) is the sum of narrow band levels from 1 Hz to 2 KHz.

A total pressure boundary condition is used for a far-field boundary and a mass flow boundary condition is used for a duct outlet boundary. To avoid the reflection from the boundary, a sponge layer is used at all boundaries. Non-rotational wall boundary condition is interpolated bounced back with a wall model and a rotational wall boundary condition is diffuse-interface method without wall model.
3. Results and Discussion

A flow field around axial fan is shown in Figure 5. It shows instantaneous Q value and nondimensional pressure. Small turbulence is observed and it interferes stators. A pressure field of the downstream shows a combination of acoustic modes. Figure 6 shows a flow velocity contour at cross section B. An error of a mass flow rate compared with experiment is 10%. Because a non axisymmetric intake exists, which is shown in Figure 3, it has a strong effect on its acoustic behavior. Figure 6 shows a speed difference between a left part and a right part of the flow field just in front of the first rotor blades. This drift flow makes a blade passing frequency noise.
A narrow band spectrum at cross-section B and the first blade passing frequency (BPF) noise level are shown in Figure 7. Locations of microphones are at 0, 90, 180, 270 degree of cross-section and the microphones are mounted on the wall surface. Evaluation points of simulation are the same as the measurement. The spectrum shows averaged spectrum of four measured points. Broad band noise and second blade passing frequency noise show good agreement with experiment. However, the first blade passing frequency noise has discrepancy about 5dB. The numerical grid in this study is only just the duct region near the axial fan, but actual duct has bend region in front of the axial fan and long duct at upstream. It will create stronger drift flow at upstream, and it will increase the blade passing frequency noise.

However, one-third octave band spectrum is used to evaluate such like industrial axial fan noise. Figure 8 shows a comparison of the one-third octave band spectrum at cross-section B, it shows good agreement between the experiment and the numerical analysis. Its overall value shows good agreement within 3dB. Considering the design of silencers or other acoustic devices for industrial fans, this discrepancy is considered to be enough for an adequate design.
4. Conclusion

In the present study, the newly developed in-house Lattice Boltzmann code has been applied to the noise simulation of industrial axial fan. Simulation results are compared with experimental results of the factory test of the industrial fan for a thermal power generation. The result shows good agreement and overall value is within 3dB error.
References

[1] Lockard D P, Choudhari M M, O’Connell M D, Duda B M, and Fares E 2017 Noise Simulations of the High-Lift Common Research Model 23rd AIAA/CEAS Aeroacoustics Conference

[2] Casalino D, Hazir A, and Mann A 2018 Turbofan Broadband Noise Prediction using the Lattice Boltzmann Method AIAA Journal vol.56, pp.609-628

[3] Nishimura S, Hayashi K and Nakaye S 2019 Implicit Large-Eddy Simulation of rotating and non-rotating machinery with Cumulant Lattice Boltzmann method aiming for industrial applications AIAA Aviation 2019 Forum AIAA-paper 2019-3526

[4] Nakahashi K 2005 High-Density Mesh Flow Computations with Pre-/Post-Data Compressions 17th AIAA Computational Fluid Dynamics Conference

[5] Nakahashi K, Kitoh A, Sakurai Y and Meinke M 2006 Three-Dimensional Flow Computations Around an Airfoil by Building-Cube Method 44th AIAA Aerospace Sciences Meeting and Exhibit

[6] Sasaki D and Nakahashi K 2014 Rapid Large-scale Cartesian Meshing for Aerodynamic Computations Proceedings of 29th Congress of the International Council of the Aeronautical Sciences (ICAS)

[7] Geier M, Schönerr M, Pasquali A and Krafczyk M 2015 The cumulant lattice Boltzmann equation in three dimensions: Theory and validation Computers & Mathematics with Applications vol. 70, pp. 507–547.

[8] Pasquali A 2017 Enabling the Cumulant Lattice Boltzmann Method for Complex CFD Engineering Problems Ph.D. Dissertation, Technische Universität Braunschweig.

[9] Geier M, Pasquali A and Schönerr M 2017 Parametrization of the cumulant lattice Boltzmann method for fourth order accurate diffusion part I: Derivation and validation Journal of Computational Physics vol. 348, pp. 862–888.

[10] Geier, M, Pasquali A and Schönerr M 2017 Parametrization of the cumulant lattice Boltzmann method for fourth order accurate diffusion part II: Application to flow around a sphere at drag crisis Journal of Computational Physics vol. 348, pp. 889–898.

[11] Kawai S and Larsson J 2012 Wall-modeling in large eddy simulation: Length scales, grid resolution, and accuracy Physics of Fluids vol. 24, p. 015105.

[12] Larsson J, Kawai S, Bodart J and Bermejo-Moreno I 2016 Large eddy simulation with modeled wall-stress: recent progress and future directions Mechanical Engineering Reviews vol. 3

[13] Spalding D B 1961 A Single Formula for the ‘Law of the Wall’ Journal of Applied Mechanics vol. 28, p. 455.

[14] Malaspinas O and Sagaut P 2014 Wall model for large-eddy simulation based on the lattice Boltzmann method Journal of Computational Physics vol. 275, pp. 25–40.

[15] Schneider A 2015 A consistent large eddy approach for lattice boltzmann methods and its application to complex flows Ph.D. Dissertation, Technische Universität Kaiserslautern

[16] Suzuki K and Inamuro T 2011 Effect of internal mass in the simulation of a moving body by the immersed boundary method Computers & Fluids vol. 49, pp. 173–187.

[17] Wang Z, Fan J and Luo K 2008 Combined multi-direct forcing and immersed boundary method for simulating flows with moving particles International Journal of Multiphase Flow vol. 34, pp. 283–302.

[18] Ji C, Munjiza A and Williams J 2012 A novel iterative direct-forcing immersed boundary method and its finite volume applications Journal of Computational Physics vol. 231, pp. 1797–1821.