INVESTIGATION OF THE FLOW AND NOISE DISTRIBUTION ON A BLOWER VIA INTEGRATION OF SIMULATION AND EXPERIMENTS

Y. D. Kuan* **J. M. Huang** **J. H. Wong** **C. Y. Chen**

Department of Refrigeration, Air Conditioning and Energy Engineering
National Chin-Yi University of Technology
Taichung, Taiwan

S. M. Lee

Department of Aerospace Engineering
Tamkang University
New Taipei, Taiwan

C. N. Hsu

CHC Fans and Blowers Company
Taichung, Taiwan

ABSTRACT

As the consciousness of energy saving and carbon reduction and comfortable environment is paid increasing attention to, the common objective of various countries with decreasing energy is to develop and popularize high efficiency and low running noise blowers. This study uses CFD to calculate the flow field and performance of a blower and compare with the experimental measurement. The characteristic curve of blower shows that the simulated and experimental values are close to each other, the difference between the values is only 0.4%. This analysis result proofs the CFD package is a highly reliable tool for the future blower design improvement. In addition, this study discusses the noise distribution of blower flow field, the periodic pressure output value calculated by CFD is used in the sound source input of sound pressure field, so as to simulate and analyze the aerodynamic noise reading of the flow field around the blower. The result shows that the simulated value of flow field around the fan has as high as 80.5 dB(A) ~ 81.5 dB(A) noise level and is agree with measurement (82 dB(A)). The noise level is low but has a sharp noise. According to the numerical results, designer of the blower modify the tongue geometry and remove the sharp noise.

Keywords: Blower, Computational Fluid Dynamics, Performance curve, Noise.

1. INTRODUCTION

Centrifugal blowers are widely appeared in many industrial applications, such as combustion air, fluid bed aeration, cooling, pneumatic conveying systems, liquid agitation, glass blowing, cooling, water blow-off, air pollution control systems, exhausting and gas boosting. Almost all the blowers have operating range. Outside this range, the blowers will surge or power overloading. As the operating flow rate is not very clear in some of these applications, users wish a blower without surge and without power overloading. Besides, as the energy conservation and green environment are paid increasing attention to, a blower should also be high efficiency and low noise. The blower is a machine which generates differential pressure to make the air flow. The blades of blower apply work to the air to increase its static pressure value and dynamic pressure. The design static pressure should have the same flow rate with the best efficiency point. This is a big challenge to a blower designer. The earlier blower designs and developments were mostly based on the measured data of wind-tunnel experiment [1], and the manpower, time, materials and instruments consumed in the experimental process were very considerable. Therefore, the advance of computer technology contributes to continuous innovation and

* Corresponding author (ydkuan@ncut.edu.tw)

DOI : 10.1017/jmech.2017.36

development of numerical analysis method, the time cost of calculation decreases as the computing efficiency increases.

The Computational Fluid Dynamics (CFD) technology will be indispensable to the design of blower and flow field analysis [2, 3] in the future. However, the accuracy of computational results is very important or it will mislead the designers.

As the government has made many environmental legislations for energy saving, it is required to face the competitive pressure from foreign manufacturers, as well as to follow the environmental regulations made by home government. Making optimum high performance low noise blowers is one of the future trends [4].

Taiwan green building label EEWH evaluation system published by the Ministry of the Interior in 1999 has become a country-level green building certification specification. The environmental considerations and popularization of urban building greening can abate the pollution of environmental noise according to investigation. Therefore, in terms of environmental quality, the decibel of noise is one of important factors to be improved for energy saving and carbon reduction.

2. RESEARCH METHOD AND PROCESS

2.1 Questions and Discussion

Generally, the flow field of blower is analyzed by MRF (Multi Reference Frame) and steady-state model. However, when the impeller in the blower rotates, it is a dynamic interaction between the air flow and blower housing. Therefore, the simulation result of steady state approximates to a mean, but the effect of the interaction of air flow in the actuation duration cannot be analyzed.

In view of this, the purpose of this study is to use the dynamic grid technology of CFD for transient analysis, to analyze the flow field of blower, so as to analyze and discuss the mean of a period of time after stabilization and the measured experimental data value of blower according to the periodic oscillation of airflow with time in the case of fixed outlet-inlet pressure difference [5].

The simulated oscillation period result and the measured values of blower are discussed, and the improvement scheme is discussed and the simulation is validated according to the result. In addition, the torque on the rotating bearing resulted from the rotation of blower impeller is analyzed, and the periodic oscillation is observed, so as to be analyzed and compared with the measured values of blower.

2.2 Solid Modeling

The picture of the blower studied in this paper is shown in Fig. 1. The CAD model is drawn by using Solid Work 3D drawing software before simulation. Then the CAD model is imported to the FlowVision CFD software. A 2m (L) × 1.04m (W) × 0.47m (H) model of the FlowVision software is shown in Fig. 2. The drawn Solid Work file is saved as STP file, and then the STP file is exported as finer MESH file, the file is imported

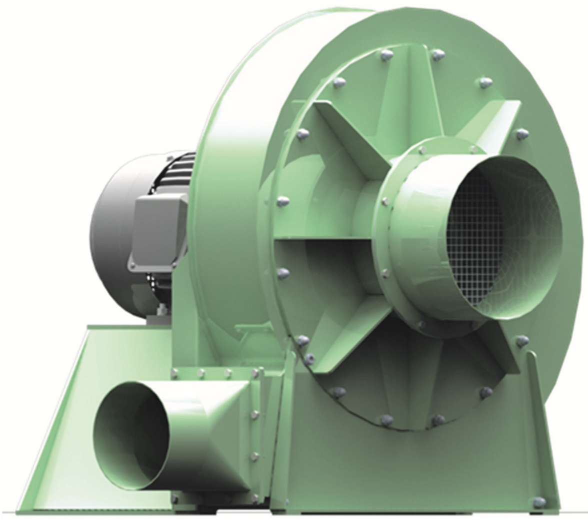

Fig. 1 Picture of the blower studied in this paper.

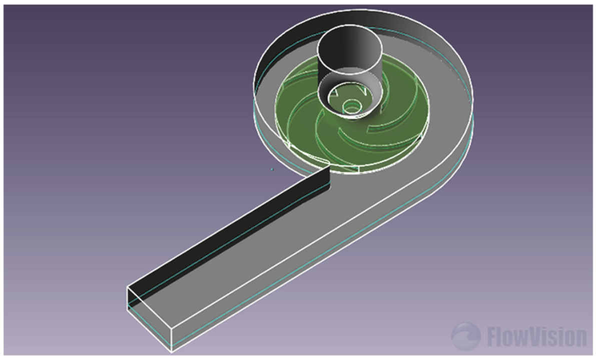

Fig. 2 Model construction in the CFD simulation.

into CFD simulation, the conditions are set according to the actual state of wind turbine.

2.3 Software Application

The simulation software CFD used in this study, FlowVision, is a professional general-purpose CFD software package from Russia. It can solve complex fluid dynamics problems, such as complex three-dimensional laminar flow, turbulent flow, combustion and electrochemistry. It supports the latest C++ syntax, providing a modular and flexible processing mode for complex CFD problems. Its particular sub-grid geometry resolution makes it easier to be combined with CAD and finite element mesh. Figure 3 shows the software architecture. The CFD package is an integrated software which integrates preprocessing, solver and post processing into the same system. The user can set fluid model, physical and methodological parameters, initial and boundary conditions (preprocessing); execute and control calculation (solver); and display results (post processing) in the same window. The user can stop calculation at any time, and change setup parameters and continue or restart calculation [6].

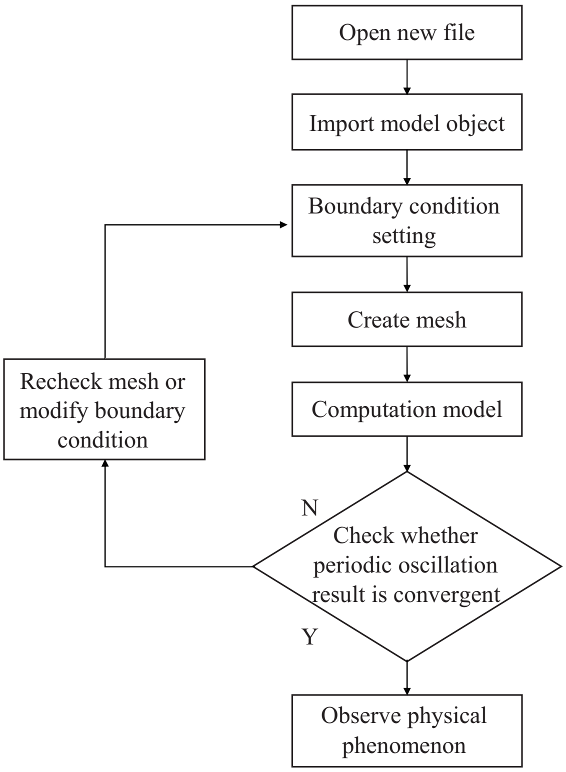

Fig. 3 Software architecture for the CFD simulation.

The solver uses finite volume method and fully implicit N-S equation for computation. The finite volume method is used to discretize governing equation [7]. The implicit velocity-pressure separation algorithm is used to process Navier-Stokes equations. The governing equation system consists of various condition equations. In addition, the solver adopts conservative velocities obtained at the given time step for solving the Navier-Stokes equations and the other convection-diffusion equations entering the mathematical model. The method is compatible with technologies of moving boundaries.

The diffusion- convection scheme adopted in this paper is called as "smooth reconstruction scheme, and the methodology was fully developed and comprehensive test in a commercial CFD package, CAPVIDIA FlowVision CFD [8, 9].

The computer system for numerical simulation in this paper is an Acer workstation with two CPUs (Intel(R) Xeon(R) CPU E5-2687W 0 @ 3.10GHz), one GPU (NVIDIA Quadro K5000), 128G RAM and Windows 7 64bit.

The following assumptions are made in the CFD simulation:

1. The air of fluid medium is incompressible Newtonian fluid.
2. The turbulent flow is simulated by standard k-e model.
3. The dynamic grid technology is used for transient analysis of flow field.
4. Set the object outlet and inlet conditions.
5. Set angular velocity of rotation of central impeller.
6. Monitor and record average velocity at blower air outlet and rotary torque of central impeller.

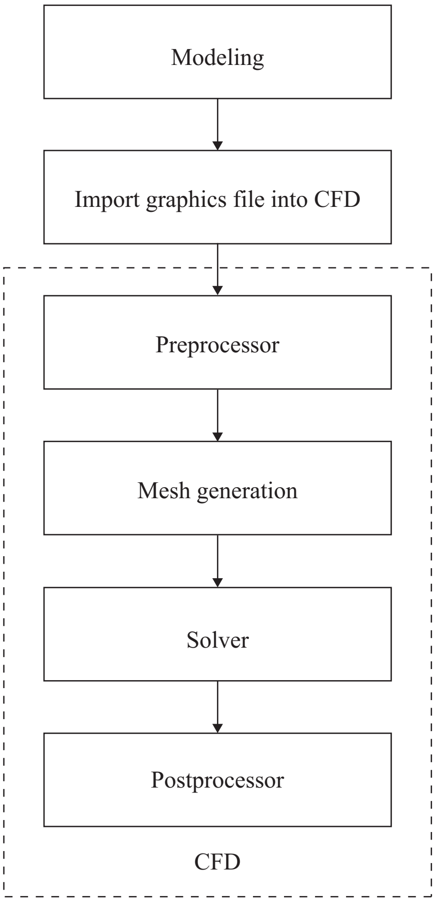

Fig. 4 Flow chart of the CFD simulation analysis.

2.4 Simulation Model Building

2.4.1 Simulation Analysis Process

For simulation in this study, the object is built by using Solid Work drawing software, and then the file is imported into simulation software. The working medium of fluid is set, and then the boundary conditions of physical properties, air inlet/outlet conditions and angular velocity of rotation are set. A simulation process is derived from flow field analysis according to the main architecture of software, as shown in Fig. 4.

2.4.2 Condition Setting

This simulation environment temperature is set as 20°C, the atmospheric pressure is 101325 kPa, it is Newtonian fluid motion, standard k-e turbulent model, impeller rotation speed is set as 3500 rpm.

2.4.3 Mesh Generation

The grid construction is one of key jobs of numerical analysis, the number and quality of grids can influence the analysis time and simulation result. In the finite volume method, a grid can represent a fixed value of many physical characteristics, such as velocity, pressure, temperature and so on. If the grids are too loose, it is difficult to work out the average value in volume. Therefore, the boundary setting shall be correct in numerical calculation, and the quality and quantity of grids are of great importance.

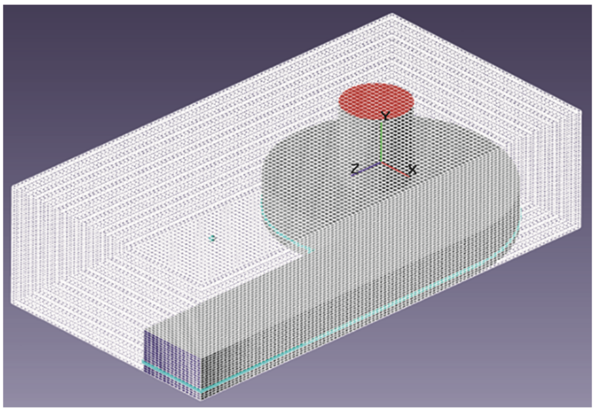

Fig. 5 Mesh construction in the CFD simulation.

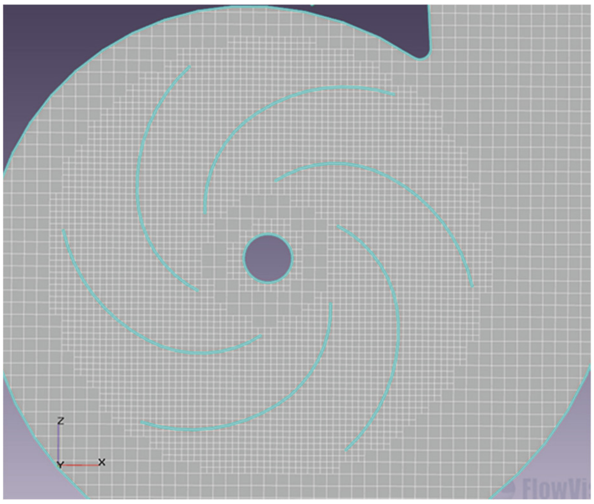

Fig. 6 Local refinement mesh.

In the blower simulation experiment, the unstructured hexahedral mesh is used, as shown in Fig. 5 and the local refinement is used in the region of the blades, as shown in Fig. 6. The surface mesh generation is free from simplifying geometry and the mesh is created directly by CAD and finite element system. This simulation is set as 50 × 50 × 100 grid points, the mesh is generated by sub-grid geometry resolution (SGGR) [10], the running time for each case is about 24 ~ 48 hours.

3. ACOUSTIC SIMULATION AND RESEARCH METHOD

3.1 Software Application

The acoustic simulation software used in this study, LMS Sysnoise, is an acoustic platform for three-dimensional analysis. It can integrate the structure, multi-body motion, vibration, acoustic and fatigue analyses, and can import the analysis results of other analytical software, such as ANSYS, ABAQUS, MSC. Nastran, and so on. Figure 7 shows the acoustic software architecture.

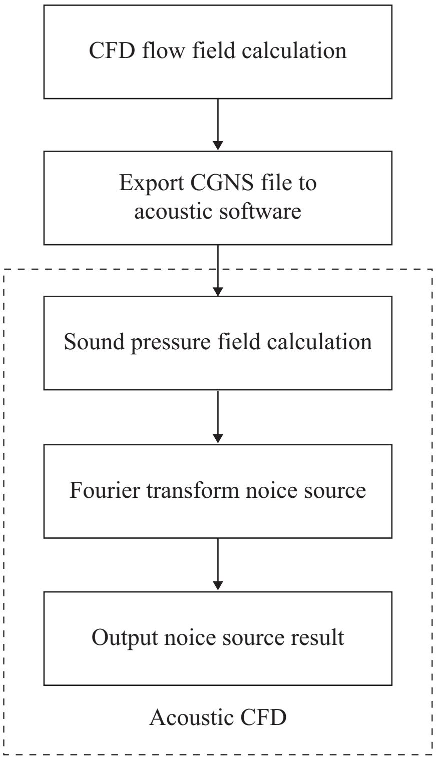

Fig. 7 Software architecture for the acoustic simulation.

The acoustic operation can use FEM and BEM to calculate various acoustic problems. With advanced acoustic computing technique, such as ATV, TPA, IFEM, Inverse Acoustics, Engine Acoustics and so on, the aeroacoustics can be calculated. With the CFD software, such as FlowVision, STAR CD, Fluent and CFX5, the audio response resulted from flow field change can be calculated.

3.2 Acoustic Simulation Process

In terms of the setup of experimental simulation in this study, the CFD implements simulation calculation of the spatial flow field, when the calculation converges stably, the CGNS file of stable time interval is extracted, and imported into the acoustic simulation software for boundary condition setting. The acoustic software simulation process is shown in Fig. 8.

The CGNS represents CFD universal symbol system, applicable to general aspects of storage and retrieval, transportable and extensible standard CFD analysis data. The CGNS design facilitates the data exchange between applications, and helps filing of stable aerodynamic data and the format is a conceptual entity of file for production record. Finally, the noise source after Fourier transform is analyzed and the fan surrounding is discussed and calculated.

4. RESULTS AND DISCUSSION

This study analyzes and discusses the experiment and simulation of blower performance curve, and analyzes

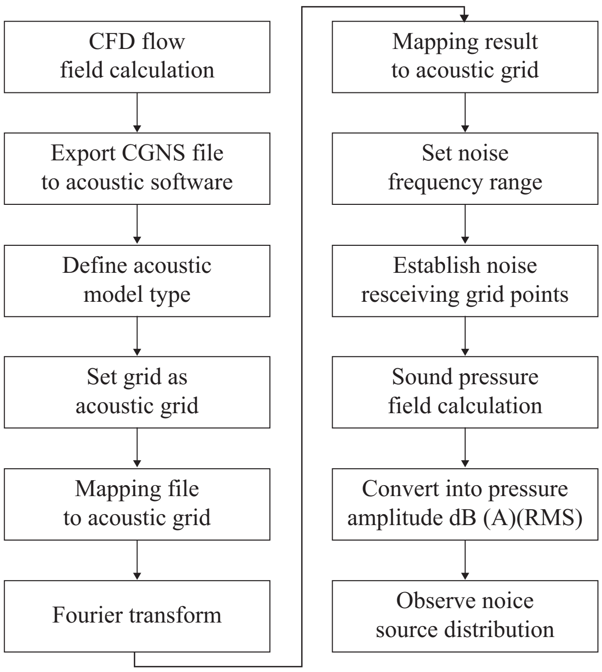

Fig. 8 Flow chart of the acoustic simulation analysis.

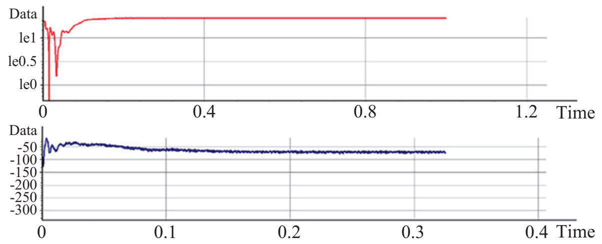

Fig. 9 Torque (down) and velocity (up) oscillations.

the noise reading of flow field around the blower calculated by acoustic simulation software, which can be compared with the experimentally measured noise in the future. Therefore, the performance curves of the experimental and simulated flow fields of fan are compared and analyzed before the simulation calculation of acoustic noise.

4.1 Flow Field Analysis Result and Discussion

This simulation aims to record the time-varying periodic oscillation of the flow rate at the blower outlet and the rotary torque of impeller under different inlet-outlet pressure differences of blower by monitoring its oscillation behavior. When the oscillation of values is stable for a period of time, the average value of relatively stable oscillation values is compared with the experimental value of blower. The oscillations are shown in Fig. 9. The Y+ over the surface of rotor at a certain time is shown in Fig. 10 and the range is about from 8 to 800, which is quite applicable to use logarithm law of wall function to calculate the flow velocity on first layer of grid near the wall surface.

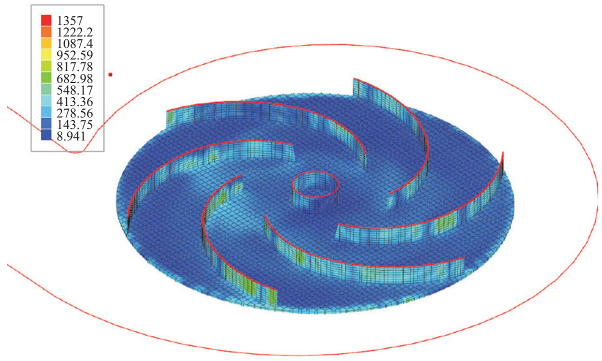

Fig. 10 The Y+ distribution on the rotor at certain time.

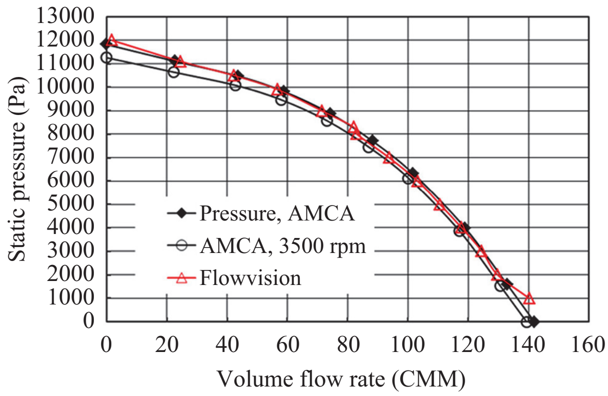

Fig. 11 Comparison of the performance curves for the blower between the simulation and experimental results.

Different pressure differentials (Pa) and volume flow rate (CMM) at all points in the performance curve are experimentally measured and the values of the blower studied in this paper are read, so as to work out the difference between the measured values and the simulation values. This blower was sent to Chicago, USA and its performance data was measured by AMCA (Air Movement and Control Association International, Inc.). Figure 11 shows the comparison between measurement and simulation. The two curves are close to each other, the difference between them is about 0.4% on an average. Therefore, the simulation data are relatively close to the measured data. The maximum static pressure value is almost 12,000 Pa, and the maximum air flow rate is nearly 140 CMM.

The performance curve via the experiment of the blower in Fig. 10 represents the original data measured by AMCA, the rotational speed of the blower changes slightly with flow rate. In order to obtain the approximate mean, the curve of “AMCA-3500 rpm” in Fig. 10 is obtained by modifying the P-Q curve using Fan-Law. The rotational speeds of various points on the curve “AMCA-3500 rpm” are all at 3,500 rpm, where the pressure is quadratically proportional to the number of revolutions and the volume flow rate is proportional to the number of revolutions.

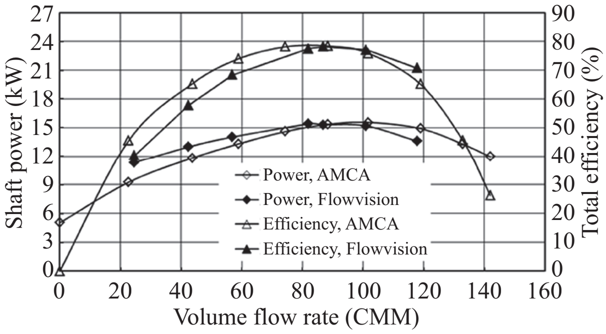

Fig. 12 Comparison of the power and efficiency curves between the simulation and experimental results.

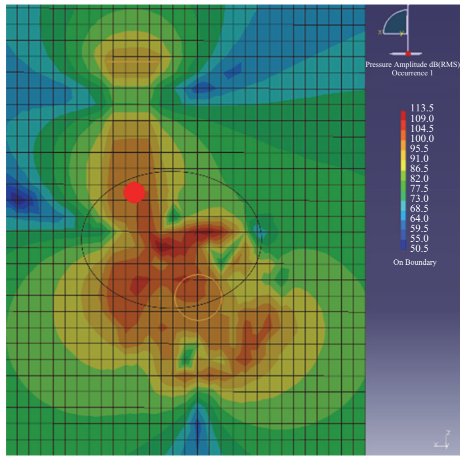

Fig. 13 Noise distribution at the highest efficiency flow rate.

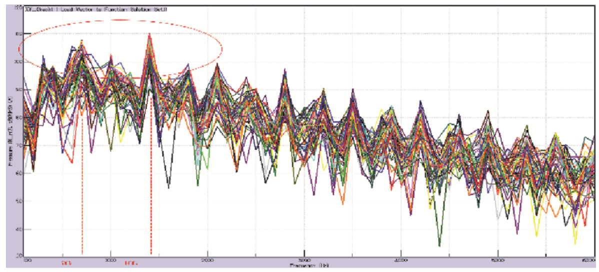

Fig. 14 Overall noise values analysis at different frequency.

The power and efficiency curves at different flow rates are shown in Fig. 11. The shaft power, Ws, at every flow rate is defined as

$$Ws = T \cdot \omega \tag{1}$$

where T is the measured and simulated torque value, ω is the angular speed. The efficiency is calculated as:

$$\eta = Wc / Ws \tag{2}$$

where Wc is the compression work at every flow rate which is the product of total pressure and volume flow rate.

From Fig. 12, the maximum efficiencies of experimental and numerical results are all about 78%, while the best efficiency flow rate of experimental and numerical results are all at 81 CMM. From the above discussions, all the static pressure, power and efficiency at different flow rate are good agree with the experimental data, it means that this CFD package is reliable on the flow field simulation of blowers. Because the noise of a blower is highly concerned with its flow field, an accurate flow field can be expected to have an accurate sound field.

4.2 Acoustic Simulation Result and Discussion

The sound power expresses the ability of fan to generate noise, and the sound pressure is the noise intensity perceived by human ears, the sound pressure varies with ambient conditions, such as temperature, medium and measurement distance and direction. Therefore, the sound level values are related to the location.

The major function of acoustic simulation is to analyze the flow field structure. It can analyze the function compiled by finite element analysis solver with external solver and the result at the highest efficiency flow rate is shown in Fig. 13. The red area is the internal range of blower, the black block is the position where the overall blower sound pressure field value is large, it is in the curved tube of runner, and the passage of flow field increases the frictional capability, the pressure value increases accordingly, and the calculated noise decibel value is relatively large.

In the region where the sound pressure is high in the blower, the position of high pressure amplitude (dB(A)) can be observed, the pressure distribution and noise value are displayed, and all the decibel conditions of various noise frequencies in the blower are presented, as shown in Fig. 14. According to the simulation results of noise frequency of all regions, the maximum average noise value is on noise frequencies 700 Hz and 1400 Hz, and the internal maximum noise intensity is as high as 110 dB(A) (RMS). In addition, the setting follows the specific noise frequency of fan and the internal sound pressure field distribution is analyzed, the noise source in the blower is as high as 109.03 dB(A) on the noise frequency of 700 Hz, as shown in Fig. 15. When the noise frequency is 1400 Hz, the noise source in the blower is as high as 108.08 dB(A), as shown in Fig. 16.

However, the acoustic simulation can analyze the noise source inside the fan, and can detect and calculate the ambient noise of flow field around the blower, and the peripheral sound pressure field distribution is displayed by simulation. Therefore, the simulation calculation is implemented about 1.5 m distant from the blower, as shown in Fig. 17.

The maximum noise at 1.5 m above the blower is 81.79 dB(A) on noise frequency of 1,400 Hz; the maximum noise at 1.5 m left to the blower is 81.68 dB(A) on noise frequency of 700 Hz; the maximum noise at 1.5 m below the blower is 80.27 dB(A) on noise frequency of 1,400 Hz. These sound levels are quite near measure-

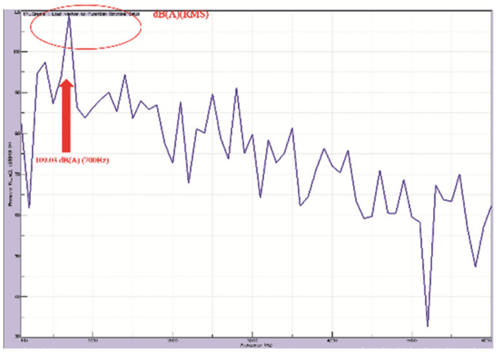

Fig. 15 Noise value analysis at frequency 700Hz dB(A).

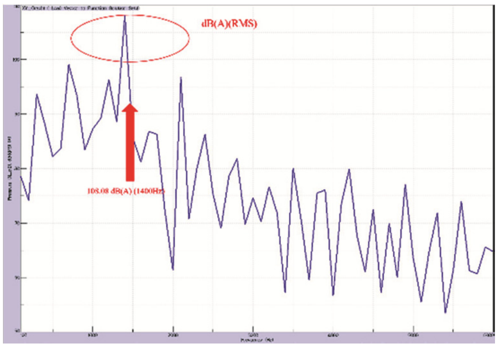

Fig. 16 Noise value analysis at frequency 1400Hz dB(A).

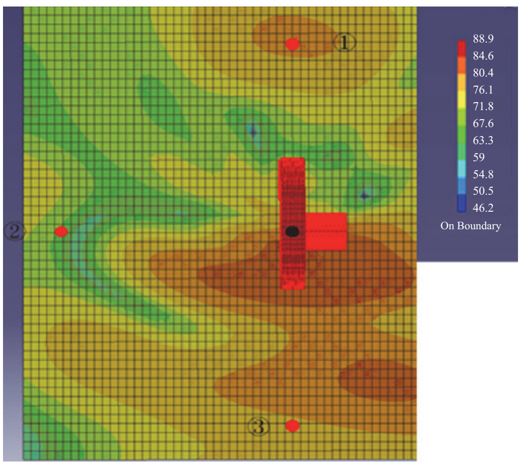

Fig. 17 Noise distribution around the blower.

ment (82 dB(A)) although the measurement was not performed inside the anechoic chamber or semi-anechoic chamber. From results of Fig. 12-15, we find that the noise at 350, 700, 1400 Hz is come from the impeller pass through the tongue of casing. A sharp noise was heard during experimental testing. Designers of the blower highly increase the space between impeller and tongue of casing and modify the shape of tongue. A new test results show that although the efficiency is 1% decrease, the noise is 1 dB(A) decrease. Moreover, the sharp noise disappears.

5. CONCLUSIONS

The performance curves of experiment and simulation are compared and analyzed, the numerical difference is only 0.4% according to average calculation. In addition, according to the comparison result of efficiency performance curves, the blower efficiencies of experiment and simulation are relatively similar, meaning the flow field simulation result is reliable. The acoustic simulation noise analysis result can present the pressure field and noise level simultaneously. In the simulation results of maximum noise level in the fan, the noise frequency 700 Hz has a noise source as high as 109.03 dB(A), and the noise frequency 1400 Hz has a noise source as high as 108.08 dB(A). According to the oscillation log data of this simulation, the physical operation convergence can be observed at any time, and various physical conditions can be observed by post-process. From the calculated periodic pressure output value, the CGNS file of stable time interval can be extracted as important parameter for calculating acoustic simulation, so as to estimate the aerodynamic noise of blower. The noise distribution in the flow field environment around the blower can be analyzed. The maximum noise above the blower is 81.79 dB(A) (1400Hz); the maximum noise on the left of blower is 81.68 dB(A) (700Hz); the maximum noise below the blower is 80.27 dB(A) (1400Hz). The experimental measurement (82 dB(A)) agrees with the simulation. Moreover, the designers find the source of sharp noise from simulation results. After modifying the tongue geometry, the bothersome sharp noise is removed.

ACKNOWLEDGEMENTS

The authors acknowledge the financial support from the Ministry of Science and Technology of Taiwan (MOST 105-3113-E-167-001-CC2).

REFERENCES

1. Dau, V. T. and Dinh, T. X., "Numerical Study and Experimental Validation of a Valueless Piezoelectric Air Blower for Fluidic Applications," *Sensors and Actuators B: Chemical*, **221**, pp. 1077-1083 (2015).
2. June, M. S., Kribs, J. and Lyons, K. M., "Measuring Efficiency of Positive and Negative Ionic Wind Devices for Comparison to Fans and Blowers," *Journal of Electrostatics*, **69**, pp. 345-350 (2011).
3. Wagner, M. R. and Popel, H. J., "Oxygen Transfer and Aeration Efficiency — Influence of Diffuser Submergence, Diffuser Density, and Blower Type," *Water Science and Technology*, **38**, pp. 1-6 (1998).
4. Li, Y. S., "Numerical Analysis of The Performance of Double-Suction Backward-Curved Centrifugal Fan," M. S. Thesis, Department of Mechanical and Electro-Mechanical Engineering, National Sun

Yat-sen University, Taiwan (2007).

5. Ramponi, R. and Blocken, B., "CFD Simulation of Cross-Ventilation for a Generic Isolated Building: Impact of Computational Parameters," *Building and Environment*, **53**, pp. 34-48 (2012).
6. Aksenov, A. A., Kharchenko, S. A., Konshin, V. N. and Pokhilko, V. I., "Flow Vision software: Numerical Simulation of Industrial CFD Applications on Parallel Computer Systems," *Parallel Computational Fluid Dynamics 2003*, pp. 401-408 (2004).
7. Glodová, I., Lipták, T. and Bocko, J., "Usage of Finite Element Method for Motion and Thermal Analysis of a Spe-Cific Object in Solid Works Environment," *Procedia Engineering*, **96**, pp. 131-135 (2014).
8. Aksenov, A. A., Gudzovsky, A. V., "The Software FlowVision for Study of Air Flows, Heat and Mass Transfer by Numerical Modelling Methods," *Proceedings of the Third Forum of Association of Engineers for Heating, Ventilation, Air Conditioning, Heat Supply and Building Thermal Physics,* pp. 31-35, U.S. (1993).
9. Aksenov, A. A., Gudzovsky, A. V. and Serebrov, A. A., "Electrohydrodynamic Instability of Fluid Jet in Microgravity," *Proceedings of 5th International Symposium on Computational Fluid Dynamics*, Japan (1993).
10. Zaghi, S., Mascio, A. D, Broglia, R. and Muscari, R., "Application of Dynamic Overlapping Grids to the Simulation of the Flow around a Fully-Appended Submarine," *Mathematics and Computers in Simulation*, **116**, pp. 75-88 (2011).

(Manuscript received December 29, 2016,
accepted for publication April 9, 2017.)