高精度流体シミュレーションによる小型ファンまわりの流体現象理解と空力騒音発生メカニズムの解析：「騒音」という環境課題の改善にむけて

課題責任者
青野　光　　東京理科大学

著者
青野　光 [*1]，高橋　勇次 [*1]，関本　諭志 [*1]，浅田　健吾 [*1]，阿部　圭晃 [*2]，
立川　智章 [*1]，藤井　孝藏 [*1]，村上　直哉 [*3]，板倉　憲一 [*4]，廣川　雄一 [*4]
[*1] 東京理科大学，[*2] 東京工業大学，[*3] ミネベアミツミ株式会社，[*4] 海洋研究開発機構 地球情報基盤センター

空冷システムを利用する家庭用電化製品などでは、ファンの高速回転化により冷却ファンの性能向上を図っており、現在ファンモータの製品設計において、ファンの高速化に伴い増加した空力騒音の低減が課題の一つとなっている。これまでのファンモータの空力騒音低減の実現を目指した解析によりファンモータ周りの流体現象が徐々に明らかになっており、具体的な設計指針を得るために必要な流れ場と音源との相関関係など一歩踏み込んだ解析が始まっている。そのような背景の下、本研究では汎用的な軸流ファンが生み出す流体現象の理解とその空力騒音発生メカニズムの解明を目指した、高解像度流体シミュレーションを実施した。対象としたファンのインペラ径と周速度に基づくレイノルズ数は 1.6×10^5 である。解析により、インペラ・ボスの回転によりファン後方に複雑な流れが生じ、強い軸流れと静止している外部流れの境界において大きな静圧変動が生じていることを明らかにした。さらに、計算した流れ場が遠方の音響場の距離減衰や圧力変動などの予測で重要な特性を十分に捉えられていることも確認した。

キーワード：Large-Eddy Simulation, 高精度計算手法, 剥離, 渦, ファン

1. 研究背景

電気機器の主要な騒音源は機器冷却のための軸流ファンやシロッコファンが作る流れの圧力変動による流体騒音である。その流体騒音の低減は長年議論されているが、ファン自身の振動起因の騒音と比較し、流体現象の特性によりその低減化が難しく設計工夫が遅れているのが現状である。現在、外径が数センチから数十センチメーターの軸流ファンやシロッコファンが家電・OA 機器・自動車・サーバ設備・工作機械などといった広範な製品分野に利用されているが、今後、社会の IoT 化に推進によりサーバ等の産業用機器や電気自動車のバッテリー用の冷却ファンだけでなく人工知能機能を搭載した先端家電や医療機器向け冷却ファンの需要が伸びて市場規模も拡大することが予想されている。従って、ファンモータの低騒音化に関する設計指針に社会がこれまで以上に期待が高まっている。

ファンモータの空力音響予測のための流体シミュレーションには、直接計算の対象を乱流の比較的大きな構造とし、それより細かい乱れだけをモデル化するラージエディシミュレーション（LES）か、乱流の直接数値計算（DNS）が必要である。しかし、ファンモータの DNS は、最新のスーパーコンピュータのパワーと資源を活用したとしても現実的ではない。他方最近では申請者らを含めスーパーコンピュータを利用したファンモータの LES 解析が進められている [1-6]。また、音響場の予測にはこれまで流れ場と音場を独立して求める分離解法が主に利用されているが、近傍場から遠方につながる一定領域において流体変動が生み出す騒音レベルを正確に評価し、かつ音源やその特性を予測するには、既存研究のように物体近傍の流れ場変動データから音響方程式により音響場全体を予測するのではなく、両者を統一的に記述する LES による音響直接計算の実施も必要となる。しかし、本研究が対象とするファンモータ周りにおいて、騒音レベルも含めて実験と比較しうるレベルの定量的なデータを得るためには、一般的な空間 2 次精度の手法では最低でも数十億、さらに数百億点の計算格子が必要となる。また、ファン 1 回転あたりの要求時間ステップも数十万を超え、騒音の低周波成分予測のためには最低ファン 10 回転分以上の計算が必要となる。著者らのこれまでの研究により高い空間解像度を有するスペクトル的高次精度の手法によれば、大凡これまでの数分の一程度の計算格子点数、1/4 程度の総時間積分ステップ数に抑えることが可能となることや本研究対象の三次元複雑形状に対しても格子のつなぎ位置の工夫などによって重合構造格子系手法を利用可能であること等が分かっている。そこで、本研究では著者らの持つ高解像度数値解析技術を活用し、小型ファンまわりの流体現象理解と空力騒音発生メカニズムを解析し、長期的な目標として音響予測レベルの向上による「騒音」という環境課題の改善にも繋げることを目指す。

2. 解析条件

本課題では、昨年度の課題 [6] に引き続き家庭用電気機器などに組み込まれている小型軸流ファンを解析対象とした。ファンの動翼は 7 枚、作動流体は空気（比熱比 1.4）とした。また、代表長さをインペラ直径、代表速度をインペラチップ周速とし、レイノルズ数を $Re= 1.6\times10^5$ とした。ファンの作動条件は静圧が 0、最大流量となる回転数とした。

3. 計算手法

流体解析ソルバ LANS3D[7] をベースにファンモータ解析用に複雑形状への対応を強化したソフトウエアを用いて解析を行った。支配方程式は 3 次元圧縮性 NS 方程式を用い、対流・粘性項とヤコビアンは高解像度計算手法である 6 次精度コンパクト差分法 [8-11] を用いて離散化した。陽的なサブグリットスケールモデルは用いずに高次精度空間フィルタがその役割を担う仮定をする陰的 LES[12-16] を用いた。時間積分として、内部反復 [17] と 2 次精度 3 点後退差分の ADI-SGS[18,19] を用いて計算を行った。計算格子は総格子点数約 6000 万点で 22 ゾーンからなる重合格子 [20] を用いた。境界条件として、外部境界は自由流出条件とし、外部境界近傍に境界の影響を減らすためにスポンジ領域 [21] を与えた。さらに、壁面は断熱すべり無し条件とした。ファンの回転は格子全体を剛体回転させ、1 回転に必要とする時間ステップは 10 万ステップである。初期条件は静止流れとし、最初にファンを 20 回転させ流れを発達させた後、更に 12 回転させた結果を用いて位相平均処理などを行い解析した結果を示す。

4. 結果と議論

図 1 はインペラ・ボスが作り出した瞬間の流れ場を流線で可視化したものである。流線の色は速度の大きさを示す。流れは複雑な三次元構造を有し、ファンの回転運動と同じ方向にうねりつつ回転軸の方向に流れていくことが分かる。続いて、図 2 に軸流れと静圧の位相平均分布を示す。位相平均分布は、12 回転分の計算結果を用い、回転 1 周期を 20 分割し、平均を行った。瞬間場で見られたように、インペラの後方にインペラ・ボスの回転により誘起された軸流れが発生し、この誘起された軸流れとボスの後方に生じている低速流れが干渉し、ボス後方には渦のような構造が確認できる。この干渉領域に生じた渦とせん断層領域で大きな静圧の変動も確認できる。さらに、インペラ・ボスの前方方向にも静圧の変動が確認でき、その大きさも後流領域より小さいことが分かる。最後に、図 3 に位相平均静圧分布に加え音圧レベルと距

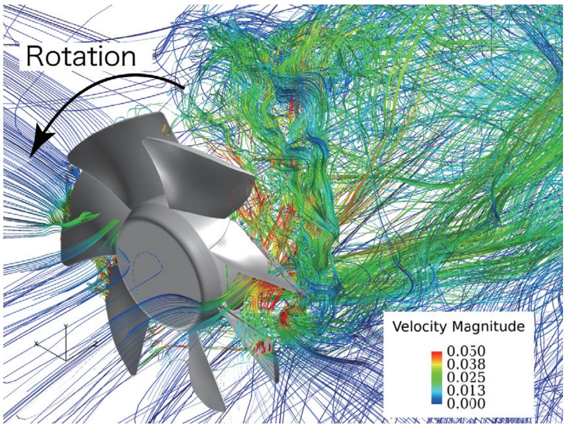
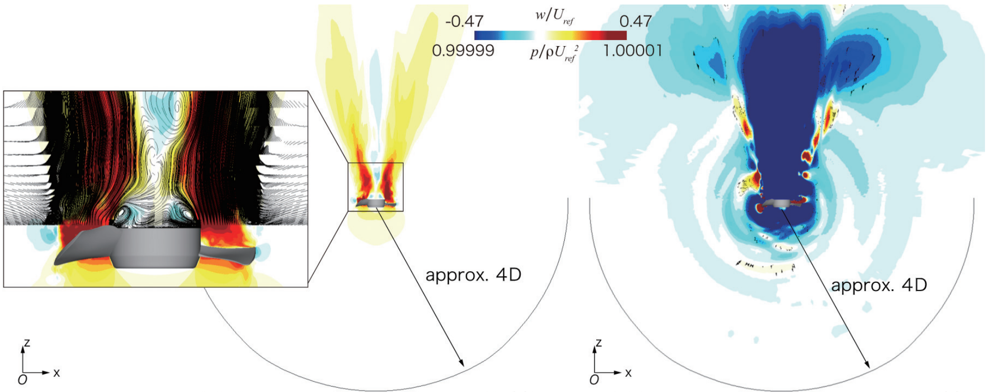
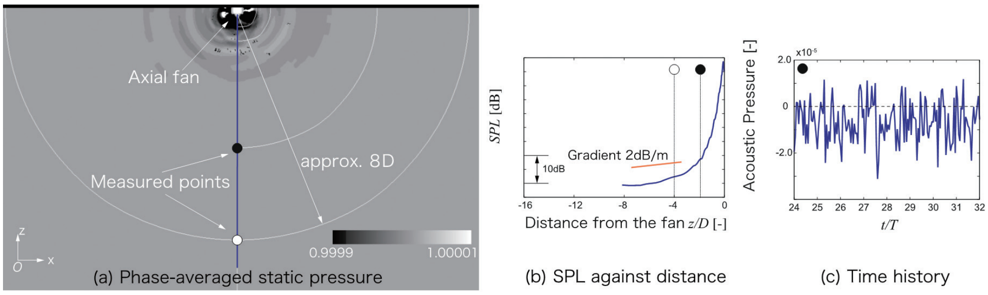


図 1　インペラ・ボス周りの瞬間流れ場構造



(a) Phase-averaged axial velocity (1/20)　(b) Phase-averaged static pressure (1/20)

図 2　位相平均場の可視化 : (a) 軸方向速度分布 , (b) 静圧分布



(a) Phase-averaged static pressure　(b) SPL against distance　(c) Time history

図 3　位相平均場 : (a) 静圧分布 , (b) 音圧レベルと距離の関係 , (c) 音圧の時間履歴

離の関係と音響圧力の時間履歴を示す。申請者らの特徴である高解像度コンパクトスキームを用いることで流れ場と遠方の音響場（距離減衰）と遠方場の圧力変動も十分に捉えられていることも分かった。

5. まとめ

小型軸流ファンの流れの高解像度流体シミュレーションを行い、ファンのインペラとボスまわりの流れ場構造の詳細な解析を行った。軸流ファン周りの近傍場の詳細な解析から、インペラが作り出した特徴的な軸流れを捉え、特にボス後方位置に生じた低速領域内の循環流およびこの誘起された回転軸方向の流れと静止している外部流れの境界において大きな静圧変動を明らかにした。さらにLES解析結果より計算した音圧レベルの距離減衰特性は、点音源を仮定した場合と概ね良好な一致を示した。今後は本課題で得た詳細な流れ場と音響場の情報を活用した解析およびケーシングの干渉を考慮した解析も行い、ファンモータの低騒音化に関する設計指針を得る。

謝辞

地球情報基盤センター情報システム部HPC支援・運用技術グループに支援を受けました。特に、斎藤氏、上原氏に継続的なサポートを頂きました。ここに謝意を記す。

文献

[1] 笹島知佳，川口清司．小型軸流ファンにおける翼周り流れの数値解析，日本機械学会論文集(B編)，2011;77:255-263.

[2] Iwase T, Obara H, Yoneyama H, Yamade Y, Kato C. Calculation of Aerodynamic Noise for Centrifugal Fan of Air-conditioner, Proc of ASME FEDSM2013, 2013, pp. 1-10.

[3] Pogorelov A, Meinke M, Schroder, W. Effects of Tip-gap Width on the Flow Field in an Axial Fan, Int. J. Heat Fluid Fl. 2016;61:466-481.

[4] Aono H, Takahashi Y, Mamori H, Yamamoto M, Tatsukawa T, Fujii K, Murakami N, A Numerical Study on Flow Structures around a Rotating Small Axial Fan with High-resolution Computational Scheme, *Inter-Noise and Noise-Con Congress and Conference Proceedings*, InterNoise17, Hong Kong CHINA, 2017, pp.4292-4298.

[5] Buchwald P, Vogt DM, Grilliat J, Laufer W, Schmitz MB, Lucius A, Schneider M. Aeroacoustic Analysis of Low-speed Axial Fans With Different Rotational Speeds in the Design Point, J. Eng. Gas. Turb. Power. 2018;140:052601-1-052601-12.

[6] 青野光，高橋勇次，関本諭志，浅田健吾，阿部圭晃，立川智章，藤井孝藏，村上直哉，板倉憲一，廣川雄一．高精度流体シミュレーションによる小型ファンから発生する音響予測レベルの確認と向上による「騒音」という環境問題の改善にむけて，Annual Report of the Earth Simulator April 2016-March 2017, pp.1-4.

[7] Fujii K, Obayashi S. High-resolution Upwind Scheme of Vortical-flow Simulations, J. Aircraft. 1989;26:1123-1129.

[8] Vinokur M, Yee HC. Extension of Efficient Low Dissipation High-order Schemes for 3D Curvilinear Moving Grids, Proc Frontiers of Computational Fluid Dynamics, 2002, pp.129-164.

[9] Deng X, Min Y, Mao M, Liu H, Tu G, Zhang H. Further Studies on Geometric Conservation Low and Applications to High-order Finite Difference Schemes with Stationary Grids. J. Comp. Phys. 2013;239:90-111.

[10] Abe Y, Nonomura T, Iizuka N. Fujii K. Geometric Interpretations and Spational Symmetry Deforming Grids. J. Comp. Phys. 2014;26:163-203.

[11] Lele SK. Compact Finite Difference Schemes with Spectral-lie Resolution. J. Comp. Phys. 1992;103:16-22.

[12] Visbal MR, Gaitonde DV. Computation of Aeroacoustic Fields on General Geometries Using Compact Differencing and Filtering Schemes. AIAA paper 1999-3706, 1999.

[13] Gaitonde DV, Visbal MR. Pade Type Higher-order Boundary Filters for Navier-Stokes Equations. AIAA J. 2000;38:2103-2122.

[14] Visbal M, Gaitonde DV. Very Higher-order Spatially Implicit Schemes for Computational Acoustics on Curvilinear Meshes, J. Comp. Acoustics. 2001;9:1259.

[15] Visbal MR, Morgan PE, Rizzeta DP. An Implicit LES Approach Based on High-Order Compact Differencing and Filtering Schemes. AIAA paper 2003-4098 2003.

[16] Grinstein FF, Margolin LG, Rider WJ. Implicit Large Eddy Simulation. Cambridge University Press 2007.

[17] Chakravarthy SR. Relaxation Methods for Unfactored Implicit Upwind Schemes. AIAA paper 1984-0165 1984.

[18] Fujii K. Simple Ideas for the Accuracy and Efficiency Improvement of the Compressible Flow Simulation Methods. Proc the International CFD Workshop of Supersonic Transport Design 1998.

[19] Nishida H, Nonomura T. ADS-SGS Scheme on Ideal Magnetohydordynamics. J. Comp. Phys. 228 2009 pp. 3182-3188.

[20] Fujii K. Unified Zonal Method Based on the Fortified Solution Algorithm. J. Comp. Phys. 118 1995 pp.92-108.

[21] Bodony DJ. Analysis of Sponge Zones for Computational Fluid Mechanics. J. Comp. Phys. 2006;212:681-702.

High-accuracy Fluid Dynamic Numerical Simulations of a Propeller Fan for Obtaining Further Understanding of Fluid Physics and Mechanisms of Aerodynamic Noise: Towards the Improvement in Noise-related Environmental Problems

Project Representative
Hikaru Aono Tokyo University of Science

Authors
Hikaru Aono*1, Yuji Takahashi*1, Satoshi Sekimoto*1, Kengo Asada*1, Yoshiaki Abe*2,
Tomoaki Tatsukawa*1, Kozo Fujii*1, Naoya Murakami*3, Kenichi Itakura*3 and Yuichi Hirokawa*3
*1 Tokyo University of Science, *2 Tokyo Institute of Technology, *3 MinebeaMitsumi Inc., and *4 Center of Earth Information Science and Technology, Japan Agency of Marine-Earth Science and Technology

Current study analyzes flow around a rotating small axial fan that is one of general-purpose products and aerodynamic noise production. A reference length and velocity are a diameter of the fan and the velocity of impeller at the tip, respectively. Based on the reference length and velocity, a Reynolds number is 1.6×10^5. The flow around the fan without the casing is simulated by means of large-eddy simulations (LES) with highly accurate and resolution computational scheme. Results present that rotating the fan produces strong axial flows that have complicated structures. The axial flows interact with the low velocity region in the behind of the boss resulting in generation of the vortex-like flow structures. The static pressures remarkably fluctuate near the vortices in the interference region and the shear layers. It is seen that the distance decay of sound pressure level and fluctuation of acoustic pressure are reasonably captured. This implicates that current simulations with the high-resolution computational scheme are promising and useful for predicting aerodynamic sounds radiated by the axial fan.

Keywords: Large-eddy simulations, Vortex, Fan, Noise, High Resolution Scheme

1. Background

The heating densities inside the industrial and commutated machinery and industrial house electric appliances have kept increasing due to downsizing the products year by year. It is most likely to make the rotational speed of the cooling fan faster in order to improve the cooling performance. However, the cooling fan with such high rotational velocity causes a considerable increase of a pressure fluctuation of flow around the fans, hence the increase of the aerodynamic noise could be accompanied. Currently, the reduction of such aerodynamic noise is one of main issues in the process of the design of the fan motor's companies.

Either large-eddy simulations (referred as LES, hereafter) or direct numerical simulations (referred as DNS, hereafter) are necessary for the reliable prediction of aerodynamic sound radiated by the cooling fans. However, the DNS of the cooling fans is unrealistic though recent high-performance computing infrastructure is used. On other hand, recently several studies have conducted the LES of the fans and reported detailed flow structures around the fans [1-6]. Moreover, although an aerodynamic/acoustic splitting method is widely adopted to calculate acoustic pressures, acoustic pressure calculations based on LES in the entire region of interest could be instructive in order to accurately evaluate the sound pressure level (SPL) and predict a source and characteristics of the sound. However, to obtain quantitative data around the fan considered in this study, several billion computational grid points is required when the standard second-order computational method for spatial discretization is considered. Furthermore, several hundred thousand computational timestep is required in order to compute low frequency component of the aerodynamic noise. If it is possible to adopt a high-order and resolution computational scheme for spatial discretization, it could reduce approximately a fraction of number of original computational grid point. Furthermore, overset methods enable us to compute flow around realistic three-dimensional fan configurations. As motivated by previous efforts, current study investigates the flow around a rotating axial fan with a high-resolution computational scheme. The main objectives of this study are to investigate flow structures produced by the fan without the casing and to discuss a possible source and characteristics of the aerodynamic fan blade noise.

2. Case Description

Current study considers the axial fan [6]. The number of the impellers of the fan is seven. The fan is a general-purpose and often used in domestic electrical appliances. For the simple purpose, this study considers only the boss and the impellers. A Reynolds number based on the diameter of the impeller (L_{ref}) and the impeller peripheral speed at the tip (U_{ref}) is 1.6×10^5. This condition corresponds to a maximum flow rate of the fan. Flow around the axial fan is initially quiescent.

3. Numerical Method

There-dimensional compressible spatially filtered Navier-Stokes equations are employed as the governing equations are solved in the generalized curvilinear coordinates. The spatial derivatives of the convective and viscous terms and symmetric conservative metrics [7-9] and Jacobian [9] are evaluated by a sixth-order compact difference scheme [9,10]. The sixth-order compact scheme can reduce the total number of computational grid points required due to its high-resolution. At the first and second points off the wall boundary, a second-order explicit difference scheme is used. In this study, the LES approach is selected so as to avoid the uncertainty of the results that arise due to modeling boundary layer turbulence. Resolving the boundary layer turbulence is important to accurately compute unsteady flow phenomena such as flow separation and reattachment. Whereas additional stress and heat flux terms are appended in an ordinary LES approach, they are not implemented in an implicit LES approach [11-15]. In this study, based on the supposition that a high-order low-pass filter selectively damps only unresolved high-frequency waves, the implicit LES is employed. A sixth-order filtering is used with a filtering coefficient of 0.45. This implicit LES approach has been well validated by Visbal *et al*. [11-15] for many problems and results of the implicit LES model have shown a good agreement with the experimental data and numerical results with standard subgrid-scale models. For time integration, the second-order backward difference is adopted and it is converged by the five sub-iterations [16] of the alternating directional implicit symmetric Gauss-Seidel implicit method [17,18] in each time step. The computational time step is 5×10^{-4} L_{ref}/U_{ref} that corresponds to maximum Courant number of approximately 25. Total number of time step required for one revolution of the fan is one hundred thousand. A zonal method [19] is used to treat the complicated geometry of the fan. At the outer boundary, all variables are extrapolated from one point in front of the outflow boundary. Here, the static pressure is fixed as the atmospheric pressure. Moreover, a sponge region [20] is assigned in order to avoid reflection of pressure wave due to existence of boundary in the space (over fifty-five L_{ref} from the center of the boss). No-slip and adiabatic-wall conditions are adopted for the surfaces of the impellers and the boss. For treatment of moving impeller and boss, all computational grids rigidly rotate about the axial axis and rotational speed is prescribed on the moving surface of the fan.

3. Results

Figure 1 shows the streamlines around the axial fan. The color of streamlines indicates magnitude of the flow velocity. Figure 2 presents the phase-averaged axial flow and static pressure distribution around the axial fan. Note that computed results for twelve rotations are employed in the phase-average process and one rotation is divided into twenty phases. As seen in Figs.1 and 2, strong axial flows are produced by the rotation of the impellers and the boss and show complicated structures. The axial flows

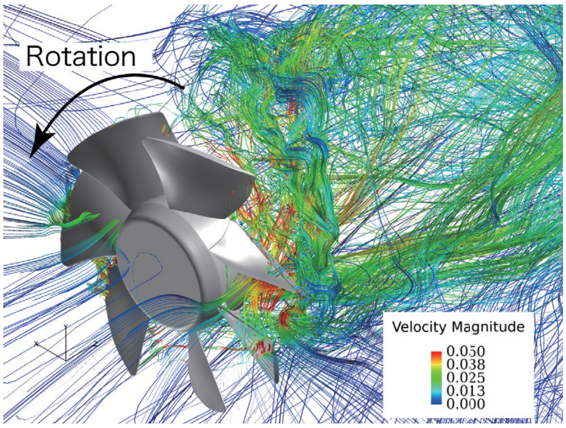
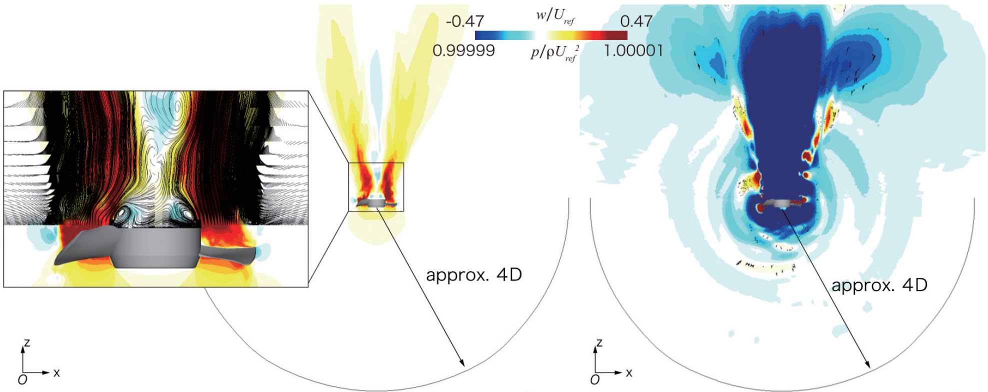
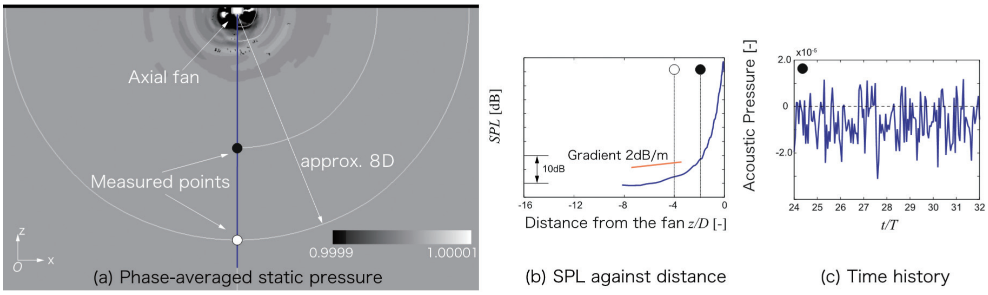


Fig. 1 Visualization of instantaneous three-dimensional flow structures around the impellers.



(a) Phase-averaged axial velocity (1/20) (b) Phase-averaged static pressure (1/20)

Fig. 2 Phase-averagaed flow structures: (a) axial velocity contours, (b) static pressure contours.



(a) Phase-averaged static pressure (b) SPL against distance (c) Time history

Fig. 3 (a) Phase-averaged static pressure distribution; (b) Sound pressure level (SPL) against distance from the axial fan; and (c) Time history of acoustic pressure at the measurement point.

interact with low velocity region in the behind of the boss resulting in the vortex-like flow structures. Large fluctuations of the static pressure can be seen near vortices in the interference region and shear layers. Moreover, static pressure fluctuates in front of the axial fan and its magnitude becomes smaller than that observed in the axial flow region. Lastly, Fig. 3 visualizes the phase-averaged static pressure distribution around the axial fan, a distance decay of the SPL, and a time history of the acoustic pressure at the measured point. Note that the SPL and acoustic pressure are calculated based on the LES results. It is seen that the distance decay of SPL and fluctuation of acoustic pressure are reasonably predicted. This suggests that current simulations with high-resolution computational scheme are promising and useful for predicting aerodynamic sound radiated by the axial fan.

4. Conclusion

This study computed and analyzed the flow structures around the rotating axial fan at the Reynolds number of 1.6×10^5 by means of the LES with the high-resolution computational scheme. Numerical flow fields near the axial fan showed complicated three-dimensional flow including strong axial flows, circular flows in the behind of the boss, and the interaction between the axial flows and quiescent air around the fan. The distance decay of SPL calculated using the LES results is in reasonable agreement with that of the point sound source. In the future data mining techniques will apply for the current LES results in order to find sources of aerodynamic noise of the cooling fans and develop low noise fan motor design.

Acknowledgement

Authors sincerely thank Mr. Saito and Dr. Uehara of JAMSTEC for continuous support.

References

[1] Sasajima T, Kawaguchi K. Numerical Analysis of Flow around Blades in Axial Flow Small Fan, Transactions of the JSME (B), 2011;77:255-263. (in Japanese)

[2] Iwase T, Obara H, Yoneyama H, Yamade Y, Kato C. Calculation of Aerodynamic Noise for Centrifugal Fan of Air-conditioner, Proc of ASME FEDSM2013, 2013, pp. 1-10.

[3] Pogorelov A, Meinke M, Schroder, W. Effects of Tip-gap Width on the Flow Field in an Axial Fan, Int. J. Heat Fluid Fl. 2016;61:466-481.

[4] Aono H, Takahashi Y, Mamori H, Yamamoto M, Tatsukawa T, Fujii K, Murakami N. A Numerical Study on Flow Structures around a Rotating Small Axial Fan with High-resolution Computational Scheme, Inter-Noise and Noise-Con Congress and Conference Proceedings, InterNoise17, Hong Kong CHINA, 2017, pp.4292-4298.

[5] Buchwald P, Vogt DM, Grilliat J, Laufer W, Schmitz MB, Lucius A, Schneider M. Aeroacoustic Analysis of Low-speed Axial Fans With Different Rotational Speeds in the Design Point, J. Eng. Gas. Turb. Power. 2018;140:052601-1-052601-12.

[6] Aono H, Takahashi Y, Sekimoto S, Asada K, Abe Y, Tatsukawa T, Fujii K, Murakami N, Itakura K, Hirokawa Y. High-accuracy Fluid Dynamic Numerical Simulations of Aerodynamic Noise Radiated from a Propeller Fan Towards the Improvement in Noise-related Environment Problems, Annual Report of the Earth Simulator April 2016-March 2017, pp.1-4.

[7] Vinokur M, Yee HC. Extension of Efficient Low Dissipation High-order Schemes for 3D Curvilinear Moving Grids, Proc Frontiers of Computational Fluid Dynamics, 2002, pp.129-164.

[8] Deng X, Min Y, Mao M, Liu H, Tu G, Zhang H. Further Studies on Geometric Conservation Low and Applications to High-order Finite Difference Schemes with Stationary Grids. J. Comp. Phys. 2013;239:90-111.

[9] Abe Y, Nonomura T, Iizuka N. Fujii K. Geometric Interpretations and Spational Symmetry Deforming Grids. J. Comp. Phys. 2014;26:163-203.

[10] Lele SK. Compact Finite Difference Schemes with Spectral-lie Resolution. J. Comp. Phys. 1992;103:16-22.

[11] Visbal MR, Gaitonde DV. Computation of Aeroacoustic Fields on General Geometries Using Compact Differencing and Filtering Schemes. AIAA paper 1999-3706, 1999.

[12] Gaitonde DV, Visbal MR. Pade Type Higher-order Boundary Filters for Navier-Stokes Equations. AIAA J. 2000;38:2103-2122.

[13] Visbal M, Gaitonde DV. Very Higher-order Spatially Implicit Schemes for Computational Acoustics on Curvilinear Meshes, J. Comp. Acoustics. 2001;9:1259.

[14] Visbal MR, Morgan PE, Rizzeta DP. An Implicit LES Approach Based on High-Order Compact Differencing and Filtering Schemes. AIAA paper 2003-4098 2003.

[15] Grinstein FF, Margolin LG, Rider WJ. Implicit Large Eddy Simulation. Cambridge University Press 2007.

[16] Chakravarthy SR. Relaxation Methods for Unfactored Implicit Upwind Schemes. AIAA paper 1984-0165 1984.

[17] Fujii K. Simple Ideas for the Accuracy and Efficiency Improvement of the Compressible Flow Simulation Methods. Proc the International CFD Workshop of Supersonic Transport Design 1998.

[18] Nishida H, Nonomura T. ADS-SGS Scheme on Ideal Magnetohydordynamics. J. Comp. Phys. 228 2009 pp. 3182-3188.

[19] Fujii K. Unified Zonal Method Based on the Fortified Solution Algorithm. J. Comp. Phys. 118 1995 pp.92-108.

[20] Bodony DJ. Analysis of Sponge Zones for Computational Fluid Mechanics. J. Comp. Phys. 2006;212:681-702.