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

近年、家庭用電化製品に加えて産業機器や通信機器などの小型化が進んでいる。それらの小型化は機器内部の熱密度増加の原因となり、空冷システムを利用する機器では主にファンの高速回転化により冷却性能の向上を図っている。このファンの回転数の増加は、冷却性能を改善するが空力騒音の増加にも繋がってしまい、現在それらの騒音の低減が課題の一つとなっている。そのような背景の下、本研究では汎用的な軸流ファン周りの流れ場構造の理解およびその空力騒音源の予測・理解を目指した、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。解析結果から、インペラの端と先端部から流れが剥離し、その剥離せん断層から複数の渦が放出していることを確認した。それらの渦の特性が剥離する部位により変化していることも明らかにした。これらの特徴的な流れ場構造とインペラ面の圧力変動との関係も調べ、それらが騒音源である可能性が高いことも示した。

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

近年、家庭用電化製品に加えて産業機器や通信機器の小型化・高速化が加速され、部品や内部素子の実装密度が非常に高くなっている。この機器内部の高密度化は発生する熱量の増加を招く。結果的に、そのような狭小な場所に設置する換気・冷却ファンも小型でかつ高速化されることにより増大する空力騒音の低減が課題となっている。国内外の産業界でもファン等の性能予測のための数値解析は注目を集め、ナビエ・ストークス（NS）方程式をレイノルズ平均した計算モデル（RANS）がファン等の空力性能の数値的予測手法として広く普及している。この RANS は渦を時間平均したモデルを用いるため、原理的にははく離流れなどの非定常現象を捉えることができず、空力音響予測には不適な面が多い。そこで、空力音響予測には時間平均モデルを用いない LES（Large-Eddy Simulation）や DNS（Direct Numerical Simulation）を適用することが必要となるが、計算格子の取り扱いや計算時間などに関して課題が多々存在していた。しかし、ここ最近の計算手法と計算機性能の向上と課題の重要性より、LES を適用した数値解析例 [1,2] は増えており、その予測性能の向上が示されつつある。以上の背景より、本研究では、小型ファンをはじめ回転機器周りの非定常流れの理解とその空力音響予測について、著者らの持つ高解像度数値解析技術を利用することで、流れ場の理解と音響予測レベルの確認、長期的な目的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

本研究では、家庭用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 $Re = 1.6 \times 10^5$ 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[3] 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 [4-7] 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[8-12] を用いた。時間積分として、内部反復 [13] と 2 次精度 3 点後退差分の ADI-SGS[14,15] 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 [16] 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 [17] 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た。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 5 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 はファンの翼間一ピッチ分について、渦度分布の断面図を複数示している。(a) は 25 回転目の瞬時場、(b) は 20 回転から 25 回転の 5 回転分の位相平均場を示している。また、図 1 は、同断面位置における瞬間場 (a-2) と位相平均場 (b-2) の渦度分布について、別の角度から可視化したものである。インペラとボス周りの流れ場は、インペラの前縁、後縁、先端部より剥離、剥離渦が生じている。それらの渦の特性を考察してみると、前縁と翼端から発生する渦は位相平均場と瞬間場ともに確認できるが、後縁渦は位相平均場では確認できない。これは、それぞれの渦が異なった時間スケールを有することを示唆している。図 2 は、インペラ根元と先端部付近の断面近傍流れ場に着目した可視化であり、インペラ面圧力分布からインペラ面上に複数の剥離渦の存在し、剥離せん断層が生じる面がインペラ根元から翼端に向けて変化していることが分かる。これはファンによって誘起される軸流れとインペラ周速度で定義される有効気流角がスパン位置に応じて変化している可能性を示唆している。図 3 は、位相平均場より算出した圧力変動の実効値のインペラ表面分布である。図 1 と 2 で確認した特徴的な流れ場構造がインペラ表面分布に強い圧力変動の原因になっていることが確認できる。このことから、インペラ端部から派生する渦、その剥離渦とインペラとの干渉が空力騒音源である可能性が高い。

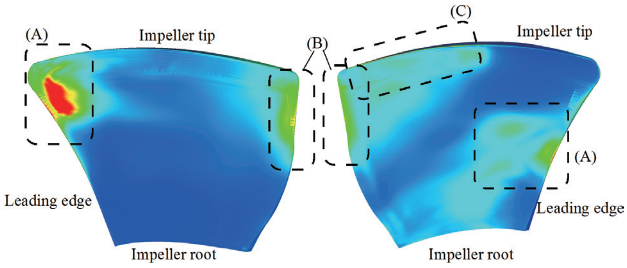

図 3　位相平均場の圧力変動の実効値分布 : サクション面側（左）と圧力面側（右）。

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。特に、インペラ周りに発生する三種類の渦（前縁剥離渦、翼端渦、後縁剥離渦）の存在を確認し、前縁剥離渦と翼端渦は回転周期と同じ周期性、他方後縁剥離渦は非定常性が強い渦であることを明らかにした。最後に、それらの流れ場特性とインペラ面圧力変動特性との関係からそれらが騒音源の可能性を示した。

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。

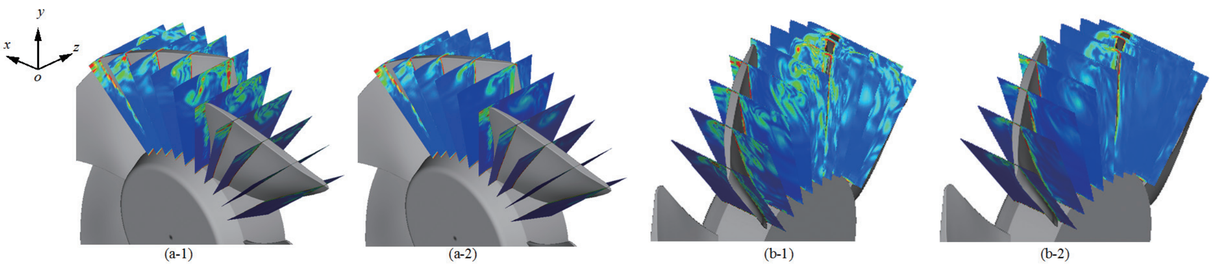

図 1　インペラ周りの渦構造の可視化 : 瞬間場 (a-1) and (b-1); 位相平均場 (a-2) and (b-2)。

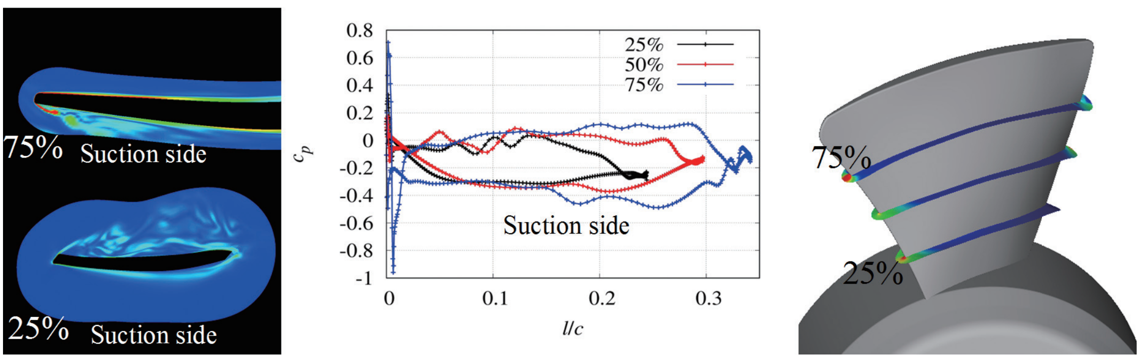

図 2　瞬間場の可視化：断面の渦度分布とインペラ面圧力分布。

文献

[1] 立原昌義，藪修一，勝俣慎一，村田敏行，加藤千幸，高山糧，上原均，廣川雄一 ,“電子機器ファンダクト系の空力騒音の数値解析 ,”平成 19 年度先端研究施設イノベーション創出事業【産業戦略利用】「地球シミュレータ産業戦略利用プログラム」利用成果報告書 2008, pp. 93-98.

[2] 岩瀬拓 ,“空調用ファンの空力騒音の予測と低減に関する研究 ,”伝熱，Vol.54 (227), 2015, pp. 25-30.

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

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

[5] 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.

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

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

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

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

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

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

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

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

[14] 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.

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

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

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

High-accuracy Fluid Dynamic Numerical Simulations of Aerodynamic Noise Radiated from a Propeller Fan 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[*4] and Yuichi Hirokawa[*4]
*1 Tokyo University of Science, *2 Tokyo Institute of Technology, *3 MinebeaMitsumi Inc., *4 Center of Earth Information Science and Technology, Japan Agency of Marine-Earth Science and Technology

This paper investigates flow structures around a rotating small axial fan that is one of general-purpose products. A reference length and velocity are a diameter of the fan and the velocity of impeller at the tip. Based on such reference length and velocity, a Reynolds number is 1.6×10^5. The flow around the fan is simulated by means of large-eddy simulations (LES) with highly accurate and resolution computational scheme. Results present that flow separates from the edges of the impeller and the tip of the impeller during the rotation. It is observed that multiple vortices are shed from the separated shear layers from the edges. Also, it is found that the leading edge and tip vortices possess periodic behavior while the trailing edge vortex has unsteady one. From above-mentioned observations, it is suggested that an interaction among the vortices observed at the suction side of the impeller could be one of aerodynamic noise sources.

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

1. Background

Recently, miniaturization of industrial and commutated machinery and industrial house electric appliances has been required and promoted. Such downsizing the products often cause the increase of the heating density inside them. For most of products employing the air-cooling, it is likely to necessary to increase a mount of the airflow for cooling the products by increasing the rotational speed of the fan. Increasing the rotational speed of the fan results in an increase of aerodynamic noise due to the pressure fluctuation of flow around the fans. In order to reduce such aerodynamic noise it is necessary to measure the flow structures around the fan and the noise generation. Large-eddy simulations (referred as LES, hereafter) have been performed using high performing computing infrastructure in order to obtain detailed flow structures around the rotating fans. Numerical results have offered our understanding of flow structures around the fan [1,2]. Computational results in the case of five hundred million computational grid points show certain improvement in the prediction of turbulence-related aerodynamic noise of a centrifugal fan because of accurately capturing near the boundary layer turbulence [2]. Unlike increase of computational grid point, a LES of the fan with high-resolution computational scheme is attractive and has a benefit in terms of computational costs and resolution of the results in comparison with the results using standard computational scheme.

As motivated by previous efforts, this project investigates the flow around a rotating axial fan with a high-resolution computational scheme. The objectives of this study are to obtain better understanding of detailed flow structures around the fan and to discuss a possible source of the aerodynamic fan blade noise.

2. Case Description

Current study considers the axial fan. 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 [3-5] and Jacobian [5] are evaluated by a sixth-order compact difference scheme [6,7]. 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 [7-11]. 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*. [7-10] 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 [12] of the alternating directional implicit symmetric Gauss-Seidel implicit method [13,14] in each time step. The computational time step is $5\times10^{-4}\ L_{ref}/U_{ref}$ that corresponds to maximum Courant number of approximately 25. Total number of time step required for one revolution of the fan is ten thousand. A zonal method [15,16] 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 [17] 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.

4. Results

Figure 1 shows a comparison of instantaneous and phase-averaged flow structures around the fan at the specific time. Although the leading-edge and tip vortices are observed in both instantaneous and phase-averaged flow structures, trailing-edge vortices can be found in only the phase-averaged flow structures. This indicates that the dominant time scale of trailing-edge vortices is different to the period of rotation of the fan. Furthermore, Fig. 2 presents an instantaneous surface pressure distribution over the selected locations of the impeller in the spanwise direction. It is found that shedding vortices from the leading-and trailing-edge and the tip significantly affect the sectional pressure distribution of the impeller. A separated leading-edge shear layer is presented on the pressure side of the impeller near the root of the impeller while is shown on the suction side of the impeller near the tip of the impeller. An effective angle of the flow based on the impeller peripheral speed and the axial flow induced by the rotating fan seems to vary in the spanwise direction. Figure 3 shows distribution of the surface pressure fluctuation with respect to phase-averaged data

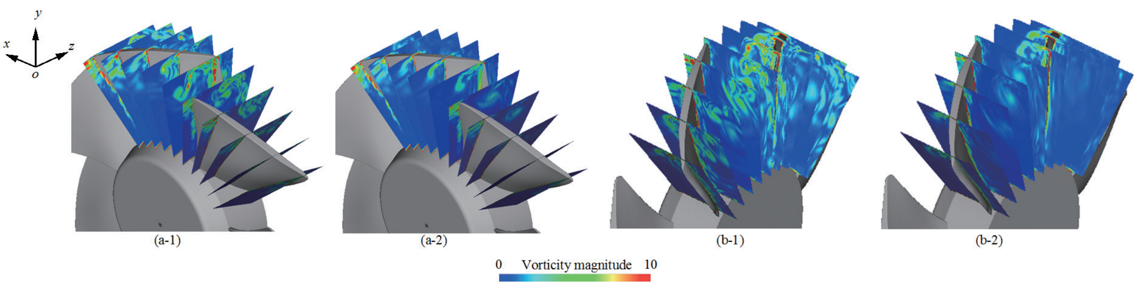

Fig. 1 Visualization of flow structures around the impeller: instantaneous field, (a-1) and (b-1); phase-averaged field, (a-2) and (b-2).

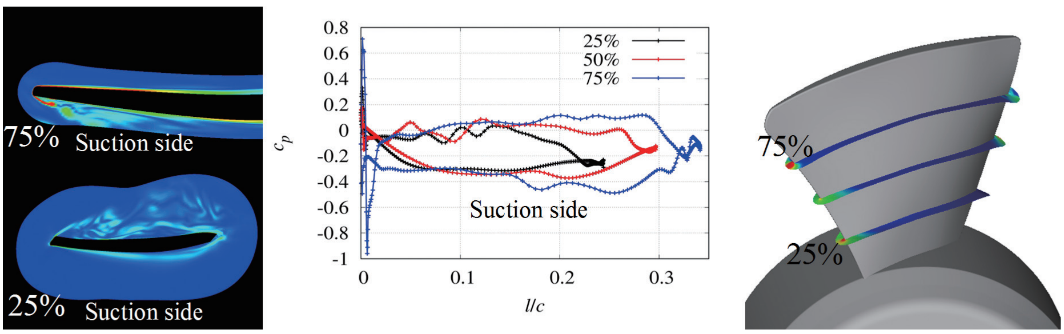

Fig. 2 Sectional vorticity and surface pressure distributions around the selected spanwise location of the impeller.

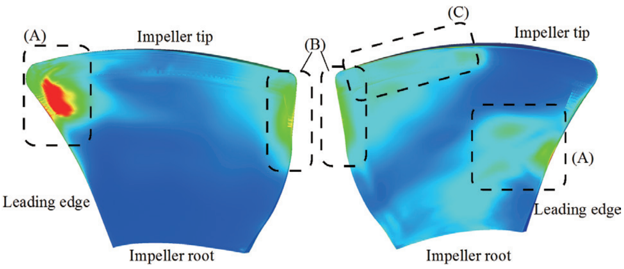

Fig. 3 Root-mean square of pressure fluctuation distribution of the impeller surface: Suction side (left) and pressure side (right).

on the impeller. The left and right side of images correspond to the suction and pressure side of the impeller, respectively. Together with Figs. 1 and 2, there are pressure fluctuations due to the leading- and trailing-edge vortices. Moreover, the pressure fluctuation appears near the impeller tip of the pressure side because of an interaction among the impellers nearby and trailing-edge vortices. In total, the surface pressure fluctuations are most likely to be source of aerodynamic fan noise. The main mechanisms of them are the vortex shedding from the edges and the interaction between shedding vortices and the impellers as expected.

5. Conclusion

This study studied the flow structures around the rotating axial fan at the Reynolds number of 1.6×10^5 by using the LES with the high-resolution computational scheme. Numerical simulations presented separated flow and vortical structures around the impellers and the boss. Vortices generated from the trailing-edge possessed the different time-scale in comparison with those of other vortices observed. However, all vortices and the interaction between the impellers and shedding vortices produce pressure fluctuation on the surface of the impeller that seems to be a main source of aerodynamic fan noise in the operational condition considered.

Acknowledgement

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

References

[1] Tachihara M, Yabu S, Katsumata S, Murata T, Kato C, Takayama R, Uehara H, Hirokawa Y, "Numerical Simulation of Aeroacoustic Noise Generated from a Fan Used for Electronics," Annual Report of the Earth Simulator April 2007- March 2008, pp. 93-96 (in Japanese).

[2] Iwase T, "Study of Predicition and Reduction of Aerodynamic Noise for Fan System of Air-Conditioner," J. Heat Trans Soc. Japan, 2015;54:25-30 (in Japanese).

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

[4] 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.

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

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

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

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

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

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

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

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

[13] 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.

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

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

[16] Kato C, Kaiho M, Manabe A. An Overset Finite-element Large-eddy Simulation Method with Applications to Turbomachinery and Aeroacoustics. J. Appl. Mech. 2003;70:32-43.

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