Journal of Thermal Engineering, Vol. 4, No. 4, Special Issue 8, pp. 2201-2210, June, 2018
Yildiz Technical University Press, Istanbul, Turkey

A HYBRID NUMERICAL/EXPERIMENTAL STUDY OF THE AERODYNAMIC NOISE PREDICTION

B. Alkan[1,*], S.O. Atayılmaz[1]

ABSTRACT

An accurate noise prediction is important in order to reduce noise emission significantly and to prevent expensive after-design treatments. This study aims to examine the aerodynamics and aeroacoustics performance of an open system consisting of an axial fan and a heat exchanger where hybrid method incorporating CFD (Computational Fluid Dynamics) and CAA (Computational Aeroacoustics) is used to predict the noise behavior. The hybrid model method used consists of three steps. Firstly, the flow is computed by means of flow-computed fluids and the pressure fluctuations are obtained. This is followed by the acquisition of acoustic signals from these fluctuations and the attainment of a sound pressure level approach with the FW-H (Ffowcs Williams & Hawkings) model. Unsteady flow field of the air channel case was obtained by using different turbulence models. The SAS model is capable of resolving largescale turbulent structures without the time and grid-scale resolution restrictions of LES (Large Eddy Simulations), often allowing the use of existing grids created for URANS simulations. For this reason, two different turbulence models, namely URANS (Unsteady Reynolds Averaged Navier Stokes) model, SAS (Scale Adaptive Simulations) model have been applied. Acoustic sources were computed based on the pressure fluctuations and sound pressure level and frequency dependent graphics were plotted with Fast Fourier Transform. On the other hand, acoustic measurements were performed in a semi-anechoic chamber for both of them. When the experimental and numerical results were compared with the previously determined receiver points, the accuracy rate was obtained as SAS, URANS respectively.

Keywords: *Aeroacoustics, Computational Fluid Dynamics, Noise Sources, SAS, Turbulence*

INTRODUCTION

Noise reduction is among the most important design criteria in various technical fields and is a challenging task in mechanical engineering. Increased awareness of the effects of noise on physiological and psychological health and strict government regulations on noise emissions have forced designers to focus more on noise reduction than ever before. Especially the recent governmental regulations enforce noise reduction in aerospace engineering, climatization and fluid machinery. Sufficient noise estimation is required to reduce noise emissions to a significant extent and to avoid costly post-design returns.

The aim of this work is to model the aerodynamic and acoustical performance of a system that includes an axial fan, heat exchanger, and duct, and use an approach to obtain noise production by computational fluid dynamics. As mentioned before, turbulence models play a crucial role in achieving pressure fluctuations during unsteady flow analysis to make good flow-induced noise estimation. For this reason, it has been tried to obtain the most realistic approach by using different turbulence models such as URANS and SAS. In particular, the effects of flow propagation and reversed flow, monopole, dipole and quadruple sources originating from the effects of high-energy vortexes and solid bodies, are calculated by the flow analysis. Thus, the Curle's boundary dipole noise contribution at low frequency and the quadrupole aerodynamic noise contribution generated by the flow field could be reduced.

Lighthill [1]–[3] who made his first work on this subject developed an acoustic analogy approach based on the Navier-Stokes equations that govern a compressible viscous fluid flow, by comparing the left side to the inhomogeneous wave equation and the right side to the acoustic sources. In addition to Lighthill's general aerodynamic sound theory, Curle [4] added the effect of turbulence in sound production and the interaction of stagnant solid surfaces. Most practical solution method to the sound radiation problem are based on an equation derived by Williams and Hawkings [5]. This equation is more general than Curle's equation and describes flow around a solid body, which moves at an arbitrary speed. Unlike Curle's equation, the Ffowcs Williams & Hawkings equation contains a monopole term, which depends on the velocity of the object with respect to a stationary

---

This paper was recommended for publication in revised form by Regional Editor Ahmed Kadhim Hussein

[1] *Department of Mechanical Engineering, Yildiz Technical University, Istanbul, TURKEY*
**E-mail address: beyza.alkan@arcelik.com*

Manuscript Received 1 June 2017, Accepted 25 October 2017

observer. At the same time, the main conlusion of Curle about the dipole characteristics of the radiated sound remains unchanged in the FW&H theory, and for an immoveable object the FWH equation reduces to Curle's equation. Using these approaches, CFD codes used in numerical methods have been developed.

There are various numerical hybrid models available in the literature to evaluate the level of noise generated by the aerodynamic effect of the systems. The hybrid model method used consists of three steps. Firstly, the unsteady flow is computed and the pressure fluctuations are obtained. This is followed by the acquisition of acoustic signals from these fluctuations and the attainment of a sound pressure level approach with the FW-H model. Therefore, it is very important to analyze the pressure fluctuations correctly. The SAS turbulence model is hybrid model capable of resolving largescale turbulent structures without the time and grid-scale resolution restrictions of LES, often allowing the use of existing grids created for RANS simulations.

In numerical analysis, the Direct Numerical Simulation turbulence model expands the solving time because it solves Navier-Stokes equations directly without any simplification. Particularly complicated geometries such as fans are not suitable for mesh quality and high Reynolds numbers. The Reynolds Average Navier-Stokes turbulence model is highly desirable because it results in a short time due to the averaging theory that does not calculate pressure fluctuations, but it cannot provide a sufficiently precise solution.

The simplest approach to the calculation of the flow field is based on the numerical solution of the Reynolds Averaged Navier-Stokes equations with the appropriate turbulence model. Page et al. [6] combined standard RANS (k-ε) with Lighthill acoustic analogy to study coaxial jet noise. Bailly et al. [7] obtained aerodynamic field from a numerical RANS solution associated with the k-ε model. The sound pressure levels obtained with the RANS model do not add up to pressure fluctuations, so the results are quite wrong.

Hu et al. [8] developed a numerical approach to noise caused by the interaction of rotating and stationary wings of an axial fan. In this study; numerical analyses are divided into three phases. First, the sound source consisting of pressure fluctuations on the channel surface is obtained by the CFD method with Large Eddy Simulation and Unsteady Reynolds Average Navier-Stokes turbulence models. As a second step, the received pressure swings are written on the frequency base by Fast Fourier Transform and the free field sound pressure is calculated by Curle analogy. Finally, the propagation and scattering of the sound source are solved by the normal derivative integral equation method.

Younsi et al. [9] used the Scale Adaptive Simulation model for non-time-dependent flow of a centrifugal fan. On the basis of the SAS model, the solution is based on the Von Karman length scale in the turbulence scale equations. This allows the SAS model to be a suitable solution for unstable flow regions with LES contents.

Another study, Kim et al. [10] have optimized a model and a developable algorithm to reduce the flow-induced sound power of an air purifier fan. The sound power broadband and wing transition frequency analysis obtained from rotating fan blades was examined separately for narrow band. Optimized fan blades analytically and experimentally overlap the results and noise is minimized as expected.

Zhao et al. [11] modified both the CFD simulations and the experimental measurements of fan geometry to reduce the turbulence-induced noise level and improve flow of the axial fan in the exterior of an air conditioner. As described in the hybrid model, the outdoor unit first analyzed the complex flow in the fan duct system by using the LES turbulence model and then applied the acoustic analogy for dominant noise sources.

Jeon et. al. [12] investigated a method to figure out the unsteady flow fields and aeroacoustics sound pressure in the centrifugal fan of a vacuum cleaner. Unsteady flow-field data are calculated by the vortex method. The sound pressure is then calculated by an acoustic analogy. The predicted tonal sound pressure levels spectra of an acoustic pressure agree very well with the measured data.

Reese et al. [13] examined influence of using different turbulent models to predict gust noise by CFD and CAA simulations. As compared to URANS, the SAS, DES and LES correctly predicted turbulence intensity. The characteristics of the sound field on the suction side, where the impeller more or less radiates into a free field, are predicted very well applying the Ffowcs Williams and Hawkings analogy fed from source data from the SAS, DES and LES. URANS can only predict tonal components of the pressure fluctuations caused by the wakes of the turbulence generator.

Borges et al. [14] developed a reliable analytical model for the evaluation of the aerodynamic noise in fans used for cooling electrical motors. In the correctness of the proposed model, 135 experimental evaluations of noise level on different fans were made. Approximately 70% of the results obtained were ± 1.5 dB different from the new model and 93% of the results showed ± 2.5 dB difference.

EXPERIMENTAL FACILITIES

The physical model in which the aerodynamics generated noise approach is studied consists of a heat exchanger, an axial fan and a duct as shown figure 1. The ambient air is absorbed by the axial fan and passed through the heat exchanger and sent to atmosphere medium again. The flow noise of the system is measured in the acoustic room and a hybrid noise approach is applied and then the results are compared.

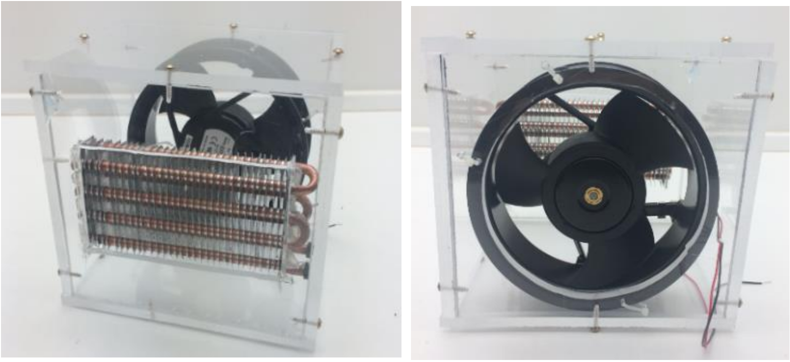


Figure 1. The physical model

The aerodynamic test method was used to measure the volumetric flow of the reference channel and the designed channel. The aerodynamic test rig is constructed according to ISO 5807-1997. The reference system rotating fan at 2000 rpm gives a volume flow rate of 39 l/s in volumetric flow measurement in the tunnel.

Acoustic measurements were made to compare the effect of turbulence models on the sound pressure on the hybrid model used for numerical analysis and to investigate the effect of channel design and flow homogenization on the sound pressure level. Aeroacoustics experiments of fan coil are carried out in the semi-anechoic chamber and full anechoic chamber. There is no significant difference between them. The acoustic measurement rig is shown in Figure 2. Sound pressure level was taken at different microphone positions for 2000 rpm providing the flow rate value. Then their graphs were drawn for each microphone positions. The dimension of the semi-anechoic chamber is 4 x 4 x 5 m, with ambient noise and cut-off frequency of 17.3 dB and 165 Hz, respectively.

Figure 2. Experimental setup for acoustic measurement

Acoustic measurements were obtained for the same 5 points specified in the numerical model. The experimental measurement results are obtained for each receiver depending on 1/3 octave band of the sound pressure level. However, in the experimental measurements, the motor noise of the fan could not be neglected.

NUMERICAL METHODS

A hybrid method is used to predict the flow-induced noise of fan coil, which incorporates the CFD (Computational Fluid Dynamics) and CAA (Computational Aerodynamic Acoustics) simulations. The hybrid method for aeroacoustics noise approach consists of three steps. First, the pressure fluctuation at the receiver points are obtained by performing flow analysis. The pressure fluctuations are then converted to acoustic signals by the

FWH acoustic model. In the last stage, the acoustic signal is plotted with the FFT to the sound pressure level depending on the frequency.

Physical Model

The flow field was obtained from the atmosphere through the fan, the sucked air was passed through a heat exchanger and then sent back to the atmosphere. Fan geometry is not simplified and heat exchanger fins are formed as sheet metal. The red dots in 3 show the microphone points where acoustic signals are received. The nearest microphone to the heat exchanger is 1, the farthest microphone is 5 in Figure 3.

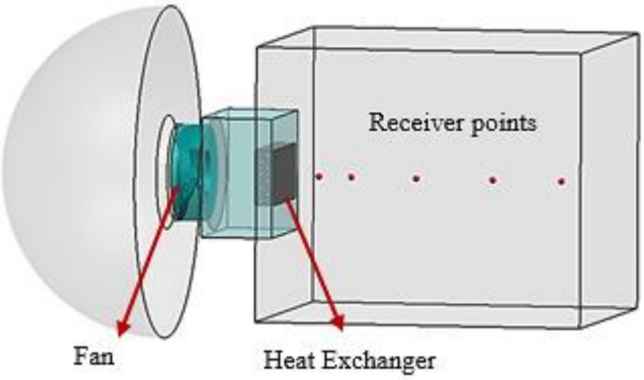


Figure 3. Numerical flow field model

For the simulation, the mesh was generated using ANSYS-Mesh-17.2 3.5 million unstructured tetra elements for CFD domain. To determine a moderate grid size suited for the present flow simulation, grid dependency study is firstly conducted. The heat exchanger and the fan region were more frequently meshed as seen in Figure 4.

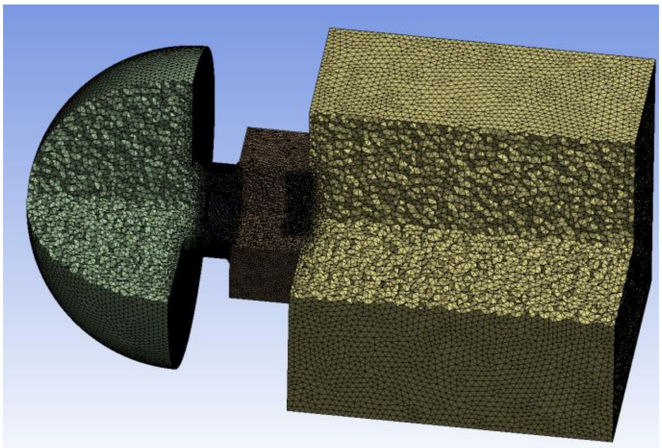


Figure 4. Mesh structure

In the discretization, the time-dependent term is discretized by the second-order implicit scheme and the convection and diffusion terms by the second-order bounded central-differencing scheme. The pressure–velocity coupling is handled by the SIMPLE algorithm. Inlet and outlet boundary condition were defined as pressure inlet and pressure outlet. A fan blade was turned 6° in a time step and 4 cycles in total at 2000 rpm by sliding mesh method. No-slip conditions are used at the solid surfaces and the moving mesh approaches are applied at the fan rotor and stationary components to consider the influence of the fan rotor and stationary component interaction.

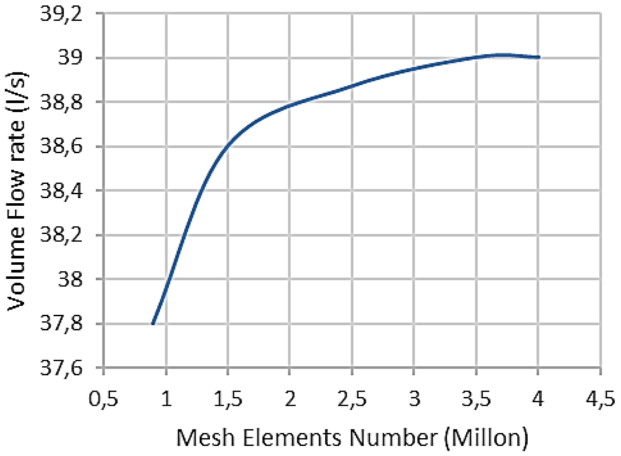


Figure 5. Mesh independency

In the discretization, the time-dependent term is discretized by the second-order implicit scheme and the convection and diffusion terms by the second-order bounded central-differencing scheme. The pressure–velocity coupling is handled by the SIMPLE algorithm. Inlet and outlet boundary condition were defined as pressure inlet and pressure outlet. A fan blade was turned 6° in a time step and 4 cycles in total at 2000 rpm by sliding mesh method. No-slip conditions are used at the solid surfaces and the moving mesh approaches are applied at the fan rotor and stationary components to consider the influence of the fan rotor and stationary component interaction.

Computional Method and Solution Control

In order to capture the complicated and complex physical features of a fan coil system, a commercial computational fluid dynamics CFD code, Fluent, is utilized to perform the flow field analysis, which solves the Navier–Stokes equation using an unstructured finite volume method.

For the pressure fluctuations, a transient simulation was carried out using the advanced SAS and k-ω-SST turbulence modelling. To avoid such undesirable grid sensitivity, Menter and Egorov [15], [16] developed an improved URANS method which can provide a LES-like behavior in unsteady and detached regions of the flow field. This concept, called Scale Adaptive Simulation (SAS), is based on the introduction of the von Kárman length scale into the turbulence scale equation. The Von Karman scale dynamically solves unstable structures that cause the SAS model to calculate like a LES in the unstable regions of the flow field. At the same time, the model provides standard URANS capabilities in stable flow regions.

For the system, firstly the flow rate was determined by steady state analysis method. After that, the unsteady flow field in the axial fan could be solved using the turbulence models such as Scale Adaptive Simulation (SAS) and Unsteady Reynolds Average Navier–Stokes (URANS) equations. In the following section the noise predicted according to numerical results will be compared with the experimental results shown in the other section.

Aeroacoustics Noise Prediction Method

To calculate aerodynamic noise, FLUENT offers three approaches, direct approach, a method using broadband noise source models, and an integral method based on FW-H acoustic analogy. The overall noise is predicted by solving the FWH equations.

The FW-H formulation adopts the most general form of Lighthill’s acoustic analogy, and is capable of predicting sound generated by equivalent acoustic sources. A further extension of this Lighthill-Curle theory was developed by Ffowcs-Williams and Hawkings to include the arbitrary motion of the solid boundaries, which e.g. occurs for fan and helicopter noise applications. ANSYS Fluent adopts a time domain integral formulation wherein time histories of sound pressure, or acoustic signals, at prescribed receiver locations are directly computed by evaluating corresponding surface integrals.

$$\frac{\partial^2 p'}{c_0^2 \partial t^2} - \frac{\partial^2 p'}{\partial x_i^2} = \frac{\partial[\rho_0 v_n \delta(f)]}{\partial t} - \frac{\partial[\boldsymbol{p_{ij}} n_j \delta(f)]}{\partial x_i} + \frac{\partial^2[\boldsymbol{T_{ij}} H(f)]}{\partial x_i \partial x_j} \qquad (1)$$

where T_{ij} and P_{ij} are;

$$\boldsymbol{T_{ij}} = \rho v_i v_j + p_{ij} - c_0^2(\rho - \rho_0)\delta_{ij} \tag{2}$$

$$\boldsymbol{p_{ij}} = p\delta_{ij} - \mu\left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i}\right) + \frac{2\partial v_k}{3\partial x_k} \tag{3}$$

There are three types of sources on the right-hand side of (1) and shows three types of sources:

- monopolar, which results from introducing a mass (per unit volume) into the considered area.
- dipolar, that takes into account the aerodynamic forces.
- quadrupolar, due to the turbulence and represented by the Lighthill's tensor.

The aeroacoustics noise includes different sources, such as the dipole noise, quadrupole noise, and monopole noise, and each of them plays different role in different applications. After the FWH acoustic model is applied, the SPL spectrum is obtained by an FFT algorithm. Fourier transform is a mathematical way to convert a time signal into its amplitude and phase.

In the acoustic model we have imposed 5 receiver points (virtual microphones) for the computation of the sound level characteristics. Acoustic analyses require an adequate time discretization, depending on the time step size the sound is computed on 160 to 2000 frequencies ranges. In this case we have set the time step at 5e-4 second, in order to reveal the sound up to 2 kHz. Running time of the numerical simulation relate to 4 complete rotaions of the fan. The pressure amplitude is given by the Sound Pressure Level as:

$$SPL = 20\log_{10}\frac{p'_{rms}}{p_{ref}} \tag{4}$$

where the SPL is given in decibels (dB). The human ear hearing in air as medium is the threshold for determining the reference pressure, $p_{ref} = 2 * 10^\wedge - 5$ Pa. This reference pressure corresponds to a SPL value of 0 dB.

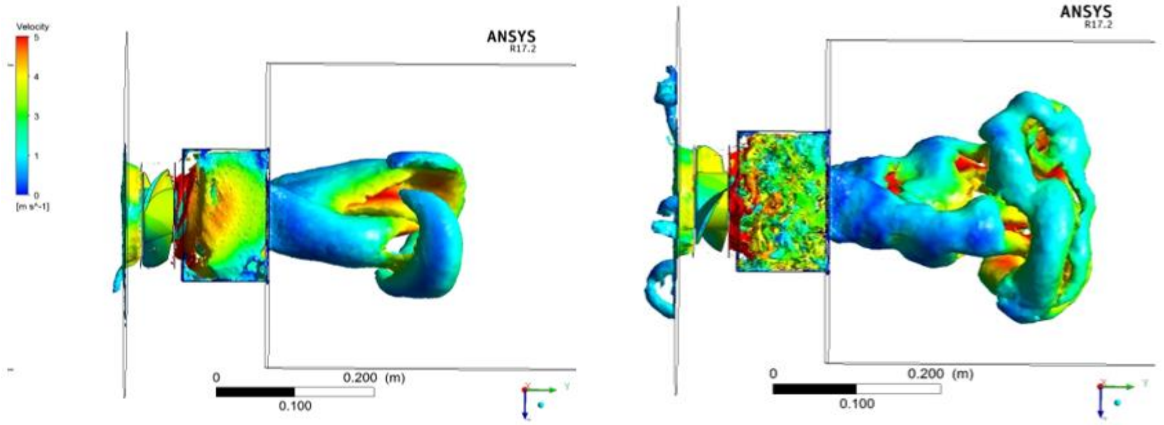
RESULTS AND DISCUSSION

This paper examined the aerodynamic and acoustical performance of a system that includes an axial fan, heat exchanger and duct and use an approach to obtain noise production by computational fluid dynamics. The fan coil flow performance and noise level are investigated both numerically and experimentally. Some of the obtained results are shared and discussed below.

This method is composed of three steps. The first step was obtained the unsteady flow analysis in order to capture pressure fluctuations for the system. SAS and URANS turbulence models were applied.

Compared to the two models in the figure 6, the SAS model seems to capture vortices and flow fluctuations more detail for the unsteady flow. In the two models, although the theories are based on the two-equation k-ω model, fluctuations can be calculated by the additional term developed by the empirical results in the SAS turbulence model. The SAS turbulence model detects the vortices that are being solved and adjusts accordingly. Thus, the high-energy vortices formed in the free-flowing region can be captured like the LES model. The URANS model in general has achieved similar results in the flow velocity distribution, but a significant difference has been observed between the physical behavior of the flow obtained with the SAS model. As the SAS model has been calculated better pressure fluctuations solution up to RANS model, sound pressure level result of SAS model has been obtained closer to experimental results.

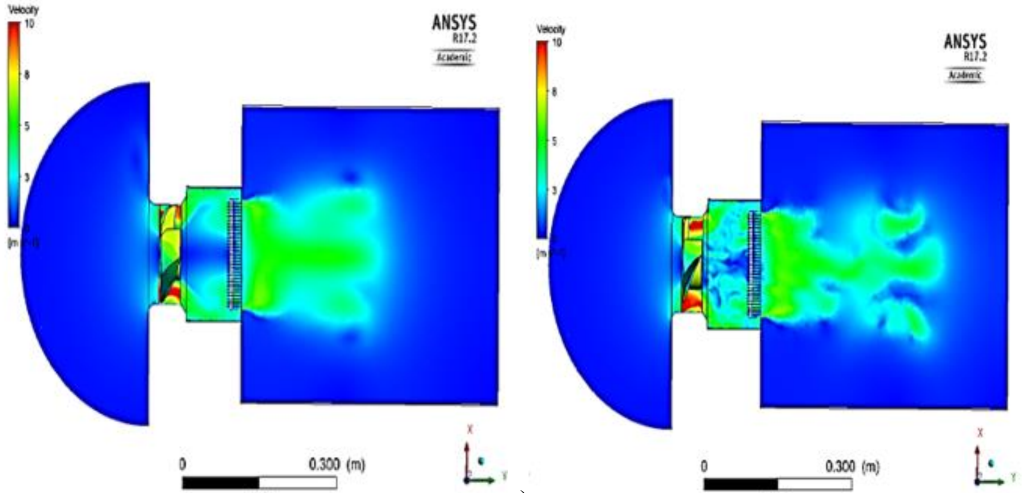
As shown Figure 6 and Figure 7, the velocity distribution results of two turbulence model were compared. The Figure 6 show the flow structure results of the URANS model on the upper side and the SAS model on the lower side. It is seen that the maximum velocity values are similar on same places. But the physical behavior, the flow structures, the model of the high-energy vortex structures are different. Obtaining reversed flow and high-energy vortices (physical behavior), one of the sources of noise, is very important for aeroacoustics noise analysis to be done correctly.



a) b)

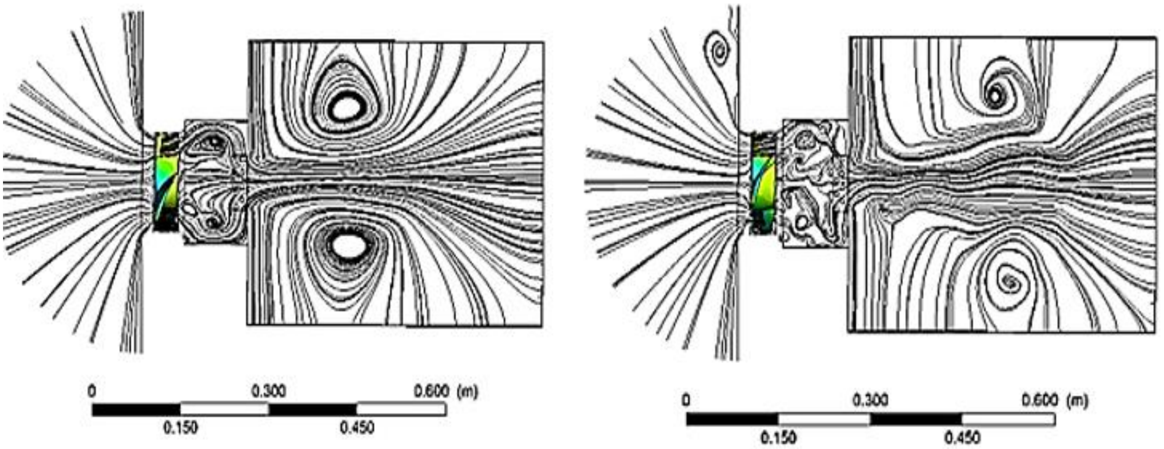
Figure 6. a) URANS b) SAS model flow analysis results

Looking at the velocity distributions in Figure 7, the left side is the URANS model and the right side is the SAS model. Although the time step for the URANS model is wider than SAS model, which also reduces the cost of the solution, URANS equations could not solved the time-dependent variable flows well since they are based on the theory of the averaged solution without adding the fluctuations to the calculations.



a) b)

Figure 7. a) URANS b) SAS model velocity distribution results



a) b)

Figure 8. a) URANS b) SAS model flow streamline structures

It was stated that the flow noise caused by the double and quadruple noise sources was caused by the reverse flow and vortices. The streamline cfd post results in Figure 8 belong to the upper figure URANS model, the lower SAS model. As noted, the SAS model provides a more detailed analysis of the reverse flow and vortices, while the URANS model provides a rougher solution.

Acoustic signals are obtained with the acoustic model applied on the developed flow. The results of the numerical studies performed for the specified receiver points are shown in Figure 9a and Figure 9b and the results of the experimental studies are shown in Figure 9c.

Figure 9b shows the FFT graph plotted against 1/3 octave band for each microphone distance after the flow analysis obtained by the SAS turbulence model. SAS turbulence model is more successful than URANS model because the SAS hybrid turbulence model implements the LES model, which is more capable of capturing pressure fluctuations in the free flow region. For this reason, it has achieved more realistic results at medium and high frequencies, which are caused by monopole and quadruple noise sources.

In Figure 10a, 10b, 10c, there are FFT graphs and experimental results obtained after applying the acoustic model after numerical SAS and URANS turbulence models at microphone 1, 3, 5 positions. Looking at the figures, the microphone position does not change the noise behavior but only the noise level is reduced. It was noticed that SAS model based on URANS and LES model. Both of two models, due to the use of the URANS equations in the

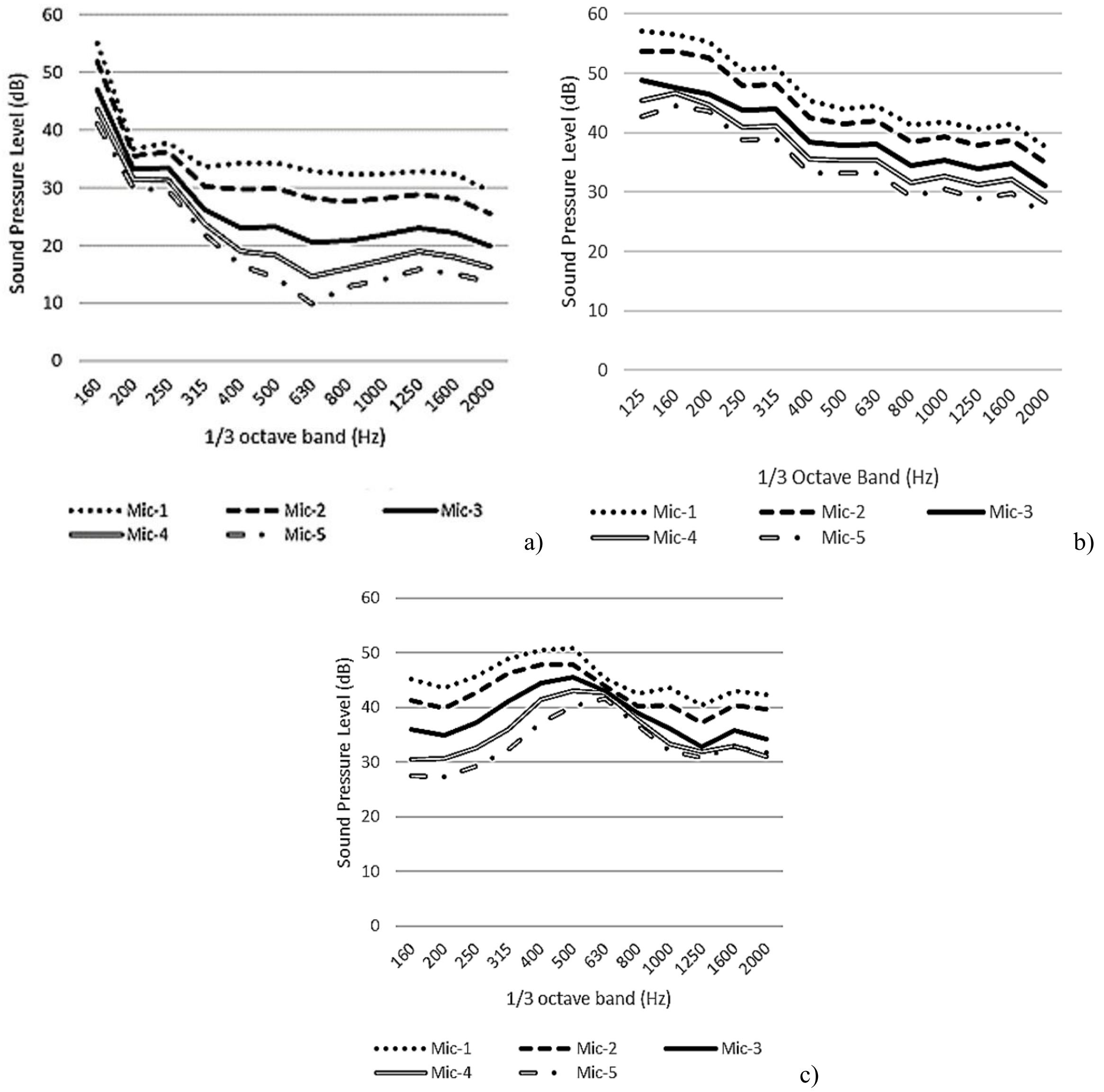
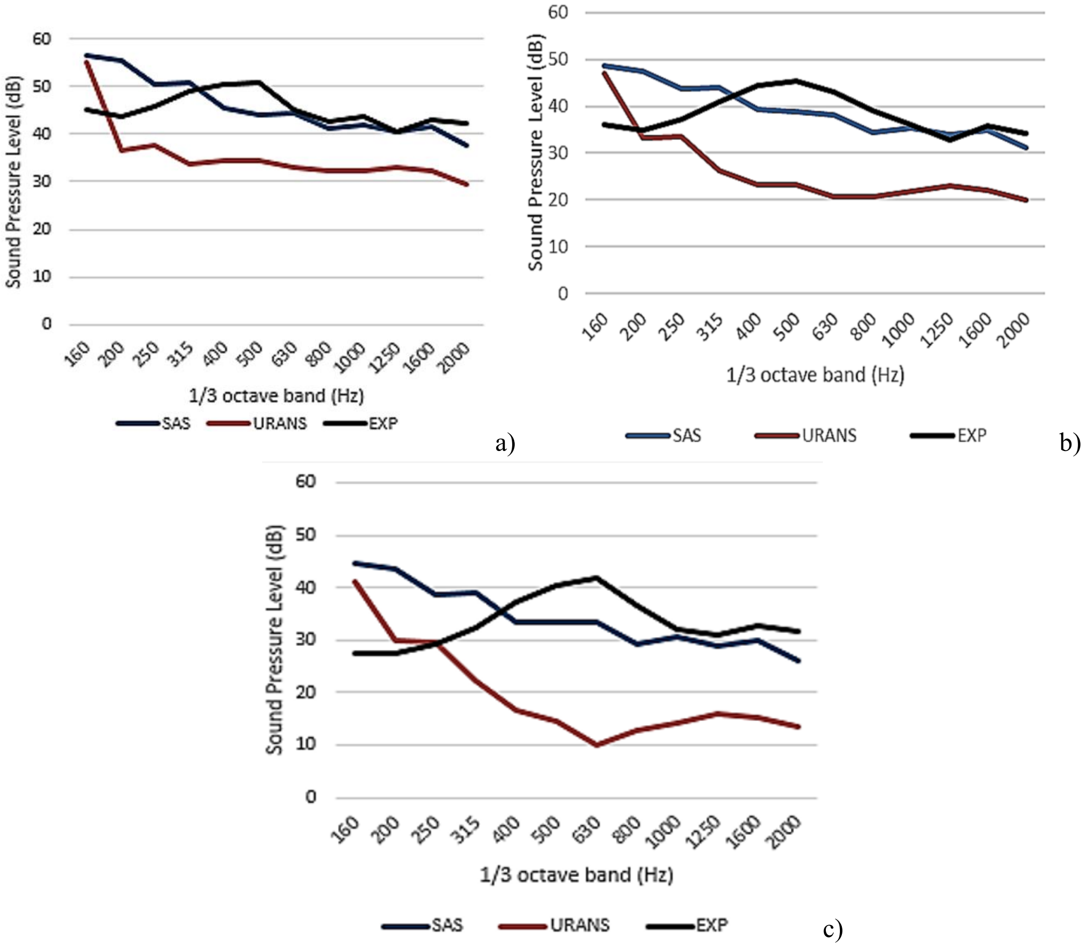


Figure 9. a) URANS model b) SAS model c) Experimental Sound pressure level result

solution beside the wall, it was not possible to obtain the structurally induced noises that occurred at low frequencies. The SAS model is a hybrid model and since it applies the LES model in the free flow region, it better solves the quadrupole noise source due to the reversed flow and vortex, except low frequencies.



a) b) c)

Figure 10. a) SPL Results comparison at Mic-1 position, b) Mic-3 position c) Mic-5 position

Figure 10a, 10b and 10c show that SAS model results are close to the experimental ones for the medium and high frequencies. However, it differs from the numerical model because it does not contain motor-induced and vibration-induced noises in experimental measurements at low frequencies.

CONCLUDING REMARKS

According to the presented development, it was proposed a hybrid analytical/experimental method for noise prediction in fan coil system. This study was examined the effect of turbulence models on pressure fluctuations, an important step of aeroacoustics simulations.

1) Since the SAS model implements the LES model in the unsteady free-flow region, it better solves the quadrupole noise source due to reverse flow and vortices.

2) The application of the FW-H equation with the SAS model gave more similar results than the experimental results compared to URANS model. But due to the influence of the fan motor, experimental results and numerical results differed at low frequency.

3) SAS model is suitable turbulence model to obtain aeroacoustics simulation at medium and high frequency.

4) The most commonly used URANS model is insufficient for flow-induced noise.

5) In general, a satisfactory agreement between predicted results and experimental data is obtained from 400 Hz up to 2000 Hz.

6) LES turbulence model will be better to make aeroacoustics simulation but cost immense is high.

7) Future investigations will concern the validation of the unsteady velocity field by comparison with Particle Image Velocimetry measurements.

NOMECLATURE

CAA	Computational Aero-Acoustics
CFD	Computational Fluid Dynamics
dB	Desibel
FFT	Fast Fourier Transform
Hz	Hertz
LES	Large Eddy Simulation
p	Pressure [Pa]
RPM	Revolution per minute
SAS	Scale Adaptive Simulation
SPL	Sound Pressure Level
t	Time [s]
URANS	Unsteady Reynolds-Averaged Navier Stokes
v	velocity [m/s]
μ	viscosity [Pa s]
ρ	density

REFERENCES

[1] Lighthill, M. J. (1952). On sound generated aerodynamically I. General theory. Proc. R. Soc. Lond. A, 211(1107), 564-587.

[2] Lighthill, M. J. (1954). On sound generated aerodynamically. II. Turbulence as a source of sound. Proc. R. Soc. Lond. A, 222(1148), 1-32.

[3] Lighthill, M. J., 1978 Waves in Fluids, Cambridge University Press, ISBN 0- 521-29233-6.

[4] Curle, N. (1955). The influence of solid boundaries upon aerodynamic sound. Proc. R. Soc. Lond. A, 231(1187), 505-514.

[5] Williams, J. F., Hawkings, D. L. (1969). Sound generation by turbulence and surfaces in arbitrary motion. Phil. Trans. R. Soc. Lond. A, 264(1151), 321-342.

[6] Page, G., McGuirk, J., Hossain, M., Self, R., Bassetti, A. (2003). A CFD coupled acoustics approach for coaxial jet noise. In 9th AIAA/CEAS Aeroacoustics Conference and Exhibit (p. 3286).

[7] Bailly, C., Candel, S., Lafon, P. (1996). Prediction of supersonic jet noise from a statistical acoustic model and a compressible turbulence closure. Journal of sound and vibration, 194(2), 219-242.

[8] Hu, B. B., OuYang, H., Wu, Y. D., Jin, G. Y., Qiang, X. Q., Du, Z. H. (2013). Numerical prediction of the interaction noise radiated from an axial fan. Applied acoustics, 74(4), 544-552.

[9] Younsi, M., Bakir, F., Kouidri, S., Rey, R. (2007). Numerical and experimental study of unsteady flow in a centrifugal fan. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 221(7), 1025-1036.

[10] Kim, J. S., Jeong, U. C., Kim, D. W., Han, S. Y., Oh, J. E. (2015). Optimization of sirocco fan blade to reduce noise of air purifier using a metamodel and evolutionary algorithm. Applied Acoustics, 89, 254-266.

[11] Zhao, X., Sun, J., & Zhang, Z. (2013). Prediction and measurement of axial flow fan aerodynamic and aeroacoustic performance in a split-type air-conditioner outdoor unit. International journal of refrigeration, 36(3), 1098-1108.

[12] Jeon, W. H., Baek, S. J., Kim, C. J. (2003). Analysis of the aeroacoustic characteristics of the centrifugal fan in a vacuum cleaner. Journal of sound and vibration, 268(5), 1025-1035.

[13] Reese, H., Carolus, T., Kato, C. (2007). Numerical prediction of the aeroacoustic sound sources in a low pressure axial fan with inflow distortion. Fan noise.

[14] Borges, S. S., Zdanski, P. S. B., Barbieri, R. (2015). A hybrid analytical/experimental model for evaluation of the aerodynamic noise in fans. Applied Acoustics, 90, 81-87.

[15] Menter, F., Egorov, Y. (2005, January). A scale adaptive simulation model using two-equation models. In 43rd AIAA Aerospace Sciences Meeting and Exhibit (p. 1095).

[16] Menter, F. R., Egorov, Y. (2005). Turbulence models based on the length-scale equation. In TSFP DIGITAL LIBRARY ONLINE. Begel House Inc.