NUMERICAL PREDICTION OF THE AEROACOUSTIC SOUND SOURCES IN A LOW PRESSURE AXIAL FAN WITH INFLOW DISTORTION

Hauke REESE¹, Thomas CAROLUS¹, Chisachi KATO²

¹ University Siegen, Institute of Fluid and Thermodynamic, Department of Turbomachinery, Paul-Bonatz-Strasse 9-11, 57068 Siegen, Germany
² University of Tokyo, Institute of Industrial Science, Tokyo, Japan

SUMMARY

In axial flow fan installations gust noise can dominate the sound radiation. In order to predict and eventually reduce sound radiation, advanced computational aeroacoustic methods (CAA) are an appropriate tool. They enable the consideration of the detailed fan geometry. CAA methods generally require an unsteady computational fluid dynamic simulation (CFD) to quantify the sources of sound. Various unsteady CFD methods have been developed in recent years. This study discusses the capability of the most common CFD methods with respect of gust noise prediction.

INTRODUCTION

Fans often operate under highly turbulent inflow conditions, e.g. due to their installation in a duct, downstream of struts or a radiator. This results in highly unsteady aerodynamic blade forces which in turn cause excessive sound radiation. Advanced state-of-the-art computational aeroacoustic methods (CAA) allow increasingly reliable sound predictions. They usually require a detailed knowledge of the unsteady flow field, obtained by a simulation with a computational fluid dynamic method (CFD).

Ideally, a Direct Numerical Simulation (DNS) solves the basic Navier Stokes equation without further simplifications and predicts the unsteady flow and the corresponding acoustic field. However, a DNS is not feasible for a complex geometry such as a realistic fan because of its immense numerical costs. To solve the unsteady flow field with fewer costs, portions of the turbulent fluctuations have to be modeled employing a turbulence model. Mainly two different strategies have been used to reduce the computational costs: (i) the ensemble averaging which is known as the Unsteady Reynolds Averaged Navier Stokes Simulation (URANS) and (ii) the filtering of the basic Navier Stokes equations which leads to the Large Eddy Simulation (LES).

With URANS the reduction of computational costs is immense, the trade-off, however, is the high degree of approximation. All random turbulent fluctuations are modeled, thus only tonal sound sources of an axial flow fan can be predicted. Another drawback is that standard URANS cannot accurately predict turbulence for a detached flow. A LES solves coarse turbulent structures directly and only small, high-frequency fluctuations are modeled. However, by extending the resolved
frequency range towards higher frequencies, the numerical costs increase tremendously, especially in the case flow fields with solid walls. To combine the advantages of a URANS with the higher resolution of a LES, hybrid methods like Detached Eddy Simulation (DES) or advanced turbulence models like the Scale Adaptive Simulation (SAS) have been developed.

The topic of the present paper is to investigate the capability of these different methods in terms of predicting gust noise sources. For the investigation we have carried four unsteady CFD simulations with the URANS, SAS, DES and LES for a low pressure axial flow fan assembly with highly turbulent inflow conditions. Finally the sound radiation is predicted from the simulated sources by the Ffowcs William and Hawkings analogy.

FAN ASSEMBLY

The investigated fan which has a diameter of $D = 300$ mm and a hub/tip ratio $\nu = 0.45$ is installed in a circular duct without guide vanes and rotates at $n = 3000$ rpm, resulting in a tip speed $u_{tip} = 47.1$ m/s and a circumferential Mach number $Ma = 0.14$. The six cambered blades have a NACA 4509 profile. The Reynolds number, based on the chord length $C$ of the blade and the mean relative flow velocity, varies from 118,000 at the hub to 178,000 at the tip. The radial tip clearance is 0.5 mm, which corresponds to 0.18 % of the rotor diameter $D$. The operating point of the maximum efficiency corresponds to a volumetric flow rate of $\dot{V} = 0.59$ m$^3$/s. This operating point was selected for all the investigations in this study. A grid type turbulence generator is installed $0.56D$ upstream of the impeller’s leading edge plane (hereafter, referred to as “reference plane”) (figure 1). The turbulence generator consists of nine struts with a square cross-section of $15 \times 15$ mm$^2$ and a separation distance of 60 mm.

![Figure 1: Fan assembly with main flow form right to left](image)

AERODYNAMIC SIMULATION METHODS

Large Eddy Simulation

The numerical flow code employed throughout the LES is named FrontFlow/Blue. It has been developed by C. Kato and successfully used for several applications [1, 2]. The code is based on a finite element discretization of the filtered incompressible continuity and Navier Stokes equations. In the present study the sub grid scales (SGS) are modeled employing the dynamic Smagorinsky model proposed by Germano [3]. A streamline-upwind finite element formulation previously
reported by Kato and Ikegawa [4] is used to discretize the governing equations. This scheme combines the Streamline Upwind - Petrov Galerkin (SU-PG) method [5] with the Taylor Galerkin method [6] and has second order accuracy in terms of both space and time. For the pressure algorithm, the Fractional Step method is used together with the BiCGStab method [7] as the matrix solver. Details of the numerical scheme have been described by Kato and Ikegawa [4] and Kato [1].

The interaction between the rotating impeller and the stationary parts is taken into account by a dynamic oversetting of the grids from multiple frames of reference [2]. Each grid domain includes appropriate margins of overlap with its neighboring grid domains downstream and upstream. For each time step, the values of the static pressure and the velocity components in the margin are interpolated in the corresponding neighbor element-wise by a tri-linear interpolation. Due to their different frames of reference, the transfer of the velocity components requires an appropriate coordinate transformation between the rotational and stationary domains. The interpolation method has been discussed in detail by Kaiho [8].

**Numerical Grid and Boundary Conditions.** The numerical hexahedral grid depicted in figure 2 is divided into four sections. The grid farthest upstream covers the inlet section, which is a cylindrical duct. A uniform axial velocity profile is set at the inlet. The impeller grid downstream of the turbulence generator is subdivided into five blocks for each blade passage. For the near blade region, an O-topology grid is used. In order to reduce the computational cost, the leakage flow through the tip clearance was not simulated.

The hub of the impeller is extended down to the outlet where the static pressure is set to zero. In order to prevent reverse flow from the outlet boundary during the iteration process, a dummy section upstream of the outlet with a sudden expansion and a subsequent gradual contraction of the cross sectional area is installed. No-slip wall conditions are applied to the remaining boundaries of the flow domain.

![Figure 2: Numerical grid; left: complete flow domain (only every third gridline is plotted), right: detail of the grid near a fan blade](image)

In order to ensure an acceptable simulation time, the overall number of the hexahedral elements for the entire flow domain is limited to approximately 5 million. However, it should be mentioned, that due to this limitation, the turbulent boundary layer on the suction surface of the blades as well as on the casing wall will not resolved by the present LES.

The time increment $\Delta t_{\text{solv}}$, which is primarily determined by the stability limit of the simulation, is set such that 10,000 time steps corresponds to a single revolution of the impeller.
Unsteady Reynolds Averaged Navier Stokes Simulation

In a URANS the effect of the lost fluctuations - due to the averagering - must be taken into account by a turbulence model. This model must capture the whole turbulent energy spectrum from the anisotropic coarse to the isotropic fine scales. This makes a universal turbulence model nearly impossible. In the URANS the shear stress transport model (SST) [9] is used. This model is a two-equation model which combines a $k-\omega$ model for the near wall region with a $k-\varepsilon$ model for the outer part. In addition to the model, an automatic wall function is used inside the boundary layer [10].

The URANS simulation is carried out with the commercial flow code ANSYS CFX10, which is based on a finite volume discretization. An implicit upwind differential scheme is used with a numerical advection correction which is formally of second order. The time derivatives are solved with a second order backward Euler scheme. A detailed description of the program is given in [11].

CFX10 uses a two dimensional sliding interface to connect the numerical grid of the turbulence generator in a stationary reference system with the one of the impeller in the rotational frame. This requires some modifications of the former LES grid.

Numerical Grid and Boundary Conditions

As compared to the LES, we used a new grid for the URANS which meets the requirement of the different interfaces and the different computational facilities. It has basically the same structure and grid density $h$ (table 1).

$$h = \left( \frac{1}{N} \sum_{i=1}^{N} \Delta V_i \right)^{1/3}$$  \hspace{1cm} (1)

In the equation $N$ denotes the number of the elements and $\Delta V_i$ the volume of a hexahedral cell. In order to reduce the numerical costs while maintaining the same grid resolution, only half of the impeller and the turbulence generator is meshed (figure 3). This is possible because of the rotational symmetry of both components. The trade-off is the necessity of circumferentially periodic boundary conditions which might damp the turbulent structures close to these boundaries.

Figure 3: Numerical grid; left: complete grid (only every third gridline is plotted), right: detail of the grid near a fan blade

The tip clearance is taken into account in contrast to the LES grid. This is possible because CFX10 uses an implicit scheme. In this case the time increment $\Delta t_{\text{SolV}}$ is not linked directly to the grid spacing, which allows a fine spatial grid resolution in the tip region without forcing very small time increments. $\Delta t_{\text{SolV}}$ is set such that 1,000 time steps corresponds to one revolution of the impeller.

The mass flow, the direction of the velocity and a medium degree of turbulence are defined as the inflow conditions. Similar to the LES, the relative static pressure is set to zero at the outlet. Because of a higher stability of the outlet conditions in CFX10, the dummy section (as in the case of the LES) is no longer necessary. For all remaining boundaries the no-slip wall condition is used.
Table 1: Dimension and density $h$ of the two different numerical grids

<table>
<thead>
<tr>
<th>Block</th>
<th>INFLOW</th>
<th>RPG2</th>
<th>FAN</th>
<th>OUTFLOW</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$h \times 10^{-3}$</td>
<td>Elements</td>
<td>$h \times 10^{-3}$</td>
<td>Elements</td>
<td>$h \times 10^{-3}$</td>
</tr>
<tr>
<td>URANS, DES, SAS</td>
<td>5.45</td>
<td>39.950</td>
<td>2.05</td>
<td>407.896</td>
<td>1.79</td>
</tr>
<tr>
<td>LES</td>
<td>3.37</td>
<td>420.500</td>
<td>1.62</td>
<td>1.779.296</td>
<td>1.60</td>
</tr>
</tbody>
</table>

Detached Eddy Simulation

The hybrid DES, which was proposed first by Spalart [12], combines a classical Reynolds averaged Navier Stokes simulation (RANS) with elements of a LES. The RANS method is applied in the near wall regions whereas the LES is active in detached flow regions. By this the moderate costs of a RANS in the near wall region is combined with the advantages of a LES in the outer regions.

DES relies on the comparison of the turbulent length-scale computed from the turbulence model and the local grid spacing. If the grid spacing is sufficiently smaller than the turbulent length-scale, the model switches to the LES mode.

The commercial flow code ANSYS CFX10 is also used for the DES simulation. This program uses a SST-DES formulation based on the idea from Strelets which is extended with a zonal limiter to avoid grid induced separation inside the boundary layer. Strelets also noted that a switch between different numerical treatments should be employed to avoid excessive numerical diffusion in the LES mode. In CFX10 a second order upwind scheme with numerical advection correction is used in the RANS and a central difference scheme in the LES region is used. The time integration is done by a second-order backward Euler scheme. A detailed description of the DES-SST formulation is given in [13].

The DES is performed on the same grid and with the same boundary conditions as for the URANS. The time increment was reduced by a factor of 2 so that 2000 time steps correspond to one revolution of the impeller.

Scale Adaptive Simulation

A DES shows strong grid sensitivity because of the switching process. Grid refinements close to the boundary can cause grid-induced flow separations when the grid spacing is below a critical size. On the other hand upon switching to LES the grid spacing immediately must meet the LES requirements, otherwise the turbulence model will produce a undefined mix of RANS and LES components.

In order to avoid such an undesirable grid sensitivity Menter et al [14, 15] developed an improved URANS method which can provide a LES-like behavior in detached flows. This concept, which is called Scale Adaptive Simulation (SAS), is based on the introduction of the von Kármán length-scale into the turbulence scale equation. The von Kármán-scale allows a dynamic adjustment of the SAS model to resolve unsteady structures, which in turn results in a LES-like behavior in unsteady regions of the flow field. The introduction of the von Karman-scale is based on the reformulation of Rotta’s equation for the integral length-scale.

In principle the SAS provides functionality similar to the DES but without explicit switching that depends on the grid spacing.
The SAS method is transformed to the SST turbulence model in the commercial flow solver ANSYS CFX10, which is used for the test case. Similar to the DES-SST, unsteady parts of the flow field are solved with a second-order central difference scheme, otherwise a second-order upwind scheme with numerical advection correction is used. The time integration is done by a second order backward Euler scheme. A detailed description of the SAS is given in [15].

The SAS is performed with the same numerical grid and time increment as for the DES.

**AERO ACOUSTIC SIMULATION METHOD**

Ffowcs Williams and Hawkings [16] derived a solution of the acoustical wave equation for the density fluctuation $\rho'$ at an observer point $x$ at a given time $t$ in the presence of hard walls and for free field conditions:

$$
\rho'(x,t) = \frac{1}{4\pi c_0^2} \left\{ \frac{\partial^2}{\partial x_i \partial x_j} \int_{s(\tau)} \left[ \frac{T_{ij}}{r[D]} \right] ds(\sigma) - \frac{\partial}{\partial x_i} \int_{s(\tau)} \left[ \frac{f_i}{r[D]} \right] ds(\sigma) \right\}
- \frac{\partial}{\partial x_i} \int_{w(\tau)} \left[ \frac{\rho_0 \alpha_i}{r[D]} \right] d(\sigma)
+ \frac{\partial^2}{\partial x_i \partial x_j} \int_{w(\tau)} \left[ \frac{\rho_0 \nu_i \nu_j}{r[D]} \right] d(\sigma)
\right\}
$$

In Eq. (2) $\sigma$ denotes the coordinate system of the moving source, $\tau$ the emission time and $D$ the Doppler Factor which is defined as

$$
D = 1 - \frac{r}{c_0 \partial \tau}
$$

Further variables in Eq. (2) are the Lighthill Tensor $T_{ij}$ which contains the velocity fluctuation of the source region $v(\tau)$, the force $f_i$ due to interaction of the flow with the moving surface $s(\tau)$, the acceleration $a_i$ and the velocity $v_i$ of the volume $v(\tau)$ which is enclosed by the surface $s(\tau)$. The distance between the acoustic source at $y$ and the observer point $x$ is $r$ (Figure 4).

According to Curle [17] the sound radiation in a subsonic flow - as in the present case - is dominated by the dipole sound sources. The dipole sources are contained in the surface integral in eq. (2). To calculate the sound pressure this surface integral is simplified employing a far field approximation. Also, in the far field $c_0^2 \rho'$ is replaced by the sound pressure $p$

$$
p(x,t) = \frac{1}{4\pi c_0^2} \int_{s(\tau)} \left[ \frac{r}{r^2 D \partial \tau \partial [D]} \frac{f_i}{r[D]} \right] ds(\sigma)
$$

---

*Figure 4: Moving Source (on a blade) $y$ in the relation to the observer point $x$*
AERODYNAMIC RESULTS

Flow field downstream of the turbulence generator

In order to verify the accuracy of the steady and unsteady flow predictions around the turbulence generator, the fan blades are removed from the hub but the hub is still present and non-rotating. For this exact configuration, Schneider [18] measured the velocity by hot wire anemometry in the reference plane (figure 1).

Figure 5 depicts fringe plots of the predicted distribution of the time averaged axial velocity, normalized by the meridional velocity \( c_m = \sqrt{V / (r_2^2 - r_1^2)} \). In all fringe plots the wake/vortex structure of the turbulence generator is clearly discernible. Compared with the measurements, the LES simulation shows a slightly smoother velocity distribution. The SAS results show a satisfying agreement with the measurements. DES and URANS show the jet/wake structure in a more pronounced manner. Because of the turbulence model the URANS cannot capture the wakes of the struts accurately. In the case of the DES the overprediction of the wakes is caused by the insufficient blending of LES to RANS.

The left-hand side Figure 6 depicts the averaged local velocity, the local turbulence intensity \( T u_{loc} \) and the integral length scale \( A_{loc} \)

\[
Tu_{loc} = \frac{\sqrt{\overline{c_w'^2}}}{c_w}; A_{loc} = \overline{c_w'} \int_0^\infty \frac{c_{w}\tau c_{w}(t-\tau)}{\tau_2} d\tau
\]

(5)

at the monitor point (MP) downstream of a middle strut. In eq. (5) the over bar denotes the direct solved axial velocity fluctuations \( c_w \) and the tilde denotes a time averaging.

\[
v/v_{c_m}
\]

\[
0.0 0.1 0.2 0.3 0.4 0.5 0.6 0.7
\]

\[
r/r_2
\]

\[
0.0 0.1 0.2 0.3 0.4 0.5 0.6 0.7
\]

\[
v/v_{c_m}
\]

\[
0.0 0.1 0.2 0.3 0.4 0.5 0.6 0.7
\]

\[
r/r_2
\]

\[
0.0 0.1 0.2 0.3 0.4 0.5 0.6 0.7
\]

\[
\frac{\epsilon}{c_{\epsilon, m}}
\]

\[
0.50 0.75 1.00
\]

Figure 5: Turbulence generator: Fringe plots of the time averaged axial velocity (normalized by the meridional velocity) in the reference plane 0.56D downstream of the turbulence generator
Considering the uncertainties in the measurements, the agreement between predictions and measurement is satisfactory. Only URANS shows larger deviations especially for the turbulence intensity. The flow field predicted with URANS contains fewer velocity fluctuations because of its high degree of modeling. The simulated and the experimentally determined $A_{\text{loc}}$ are of the same order of magnitude. Again URANS shows the highest deviations, because only some coarse structures are developed. Furthermore, the spectral density of the velocity fluctuations $\text{PSDL}_{cw} = 10\log\left(\frac{\text{d}c_w^2}{\text{d}f}\right)\left(\frac{c_w^2}{f_0}\right)$ dB is given in figure 6 (right-hand side) for the same monitor point.

Compared to the hot wire results all methods excluding URANS show a good agreement in the lower frequency ranges. The cut-off frequency of the LES, i.e. the frequency at which the predicted spectrum starts to deviate from the theoretical slope $f^{-5/3}$ of the inertia sub-range, is approximately 300 Hz. In comparison to the LES the spectra of the DES and SAS resolve the unsteady fluctuation up to 200 Hz. The level of the URANS spectrum is more than one magnitude lower over the entire frequency range. A vortex shedding was not predicted by URANS at the struts. Generally a dominant shedding frequency cannot be observed in all predicted spectra. An influence of the domain’s inflow boundary condition is not observed because the turbulence in the reference plane is completely dominated by the turbulence generator.

Aerodynamic results of the fan assembly

The discretisation error due to the various numerical grids was checked without the turbulence generator. The predicted steady state design point agrees within less than 10% deviation with the measurements [19].

Figure 7 depicts a snapshot of the absolute velocity distribution on a coaxial surface at 50% blade height. The velocity distribution of the URANS is dominated by the wakes of the struts, which interact periodically with the impeller blades. On the contrary DES and SAS show a development of turbulent structures from the wakes which move downstream and interact finally with the blades. Compared with LES these structures are relatively coarse. Unfortunately in the LES some of the turbulent structures are destroyed numerically by the dynamical oversetting between the impeller and the turbulence generator grid.

Figure 8 depicts the predicted and measured [20] power spectral density levels of the wall pressure fluctuations $\text{PSDL}_{cw} = 10\log\left(d(\text{P}^2)/\text{d}f\right)\left(p_0^2/f_0\right)$ dB on the blade suction-side at approximately mid span for two monitor points, one close to the leading edge (left side, P1) and one...
near mid-chord (right side, P2). The predicted LES spectra are based on 10 impeller revolutions, all others on more than two revolutions after the flow field has become stationary in average. The power spectra from all simulated blades are averaged in order to reduce statistical uncertainties.

The predicted spectra show a satisfactory agreement with the measurements close to the leading edge (P1). Both the predicted and the measured spectra show peaks at 200 and 400 Hz, which are caused by the wakes of the upstream turbulence generator struts. These peaks are also well predicted by the URANS. The figure shows that SAS and DES can only predict pressure fluctuations lower than 1 kHz. The RANS in the boundary layer acts like a low pass filter. The
predict pressure fluctuations are caused directly by the inflow turbulence. The LES predicts the pressure fluctuations up to 5 kHz because no statistical approximation of the boundary is applied, the inflow turbulence can directly interact with the boundary layer.

Comparing P2 with P1, the influence of the turbulent inflow ceases to exist. The level of the pressure fluctuations decreases in the downstream direction. All applied methods predict this behavior well. However, the LES shows less satisfying results at P2 when compared to P1. The deviation might be due to the fact that the grid is too coarse to resolve the turbulent boundary layer accurately. Artificial coarse structures are developing and cause higher levels of pressure fluctuations. Here DES and the SAS have a clear advantage. Because of their RANS inside the boundary layer, no overprediction of pressure fluctuation takes place.

**AERO-ACOUSTICAL RESULTS**

The sound pressure is calculated at various observer points on a circle with a radius of one meter around the center of the impeller. The inlet duct section is assumed acoustically transparent, which is comparable to neglecting the short inlet section in the experiment [19]. The small difference of the microphone positions in the experiments and the position of the observer points with respect to the impeller is neglected. Figure 9 shows the predicted and the measured power spectral density of the sound pressure \( P_{SPL_p} = 10 \log [d(p_{SPL}^2 / df) / (p_0^2 / f_0)] \) dB for two different observer points: on the rotational axis and at an angle of 45° to the rotational axis. With exception of the URANS based predictions, the results are in satisfactory agreement with the experiment for both observer points. The spectra show that a standard URANS can only predict tonal sound. SAS and DES, however, are able to predict broad band noise up to a frequency of 1 kHz. LES yields acoustic predictions which are satisfactory up to 5 kHz in our test case.

None of the simulations predict the dominant tone of the blade passing frequency observed in the experimental data. One reason might be that the experimental setup includes effects which the simulations do not take into account; for example, some coarse structures could develop in front of the inlet nozzle, or the impeller could be not fully balanced.

![Figure 9: Comparison of predicted and measured sound pressure power spectral density in a distance of one meter to the impeller center point or the inlet nozzle; left: upstream of the rotational axis, right: upstream and at an angle of 45° to the rotational axis; (reference pressure \( p_0 = 2 \times 10^3 \) Pa, reference frequency \( f_0 = 1 \) Hz)](image-url)
CONCLUSIONS

In this study LES, DES, SAS and URANS are tested to predict gust noise. As a test case a low pressure fan assembly is simulated with highly turbulent inflow conditions. As compared to URANS, the SAS, DES and LES correctly predicted turbulence intensity. Its length scale downstream of the turbulence generator agrees very well with experimental data from the hot wire measurement. The response of the blade leading edge region to the inflow turbulence in terms of surface pressure fluctuations is predicted accurately. In the test case only the LES is capable of capturing pressure fluctuations up to a range of 5 kHz.

With regards to the limits of the frequency resolution, the applied CFD methods are able to resolve velocity fluctuations downstream of the turbulence generator up to approximately 200 Hz (SAS, DES URANS) or 300 Hz (LES). However, the radiated sound is predicted reasonably correctly far above 1000 Hz for the LES case. This is due to the fact that the convection velocity of the turbulent eddies over the involved solid surfaces varies roughly by a factor of 5. In the turbulence generating mesh array, the convection velocity is on the order of the axial velocity in the stationary frame of reference, whereas over the blade surfaces these eddies are converted with approximately the relative velocity in the rotating frame of reference. The kinematics in low-pressure fans is typically such that the axial velocity is much smaller than the circumferential and thus the relative velocity. Hence, assuming a size of turbulent eddies which does not vary as the eddies are converted, the increase of convection velocity by a factor 5 leads to a captured frequency range up to 1500 Hz in the force fluctuations on the blades and subsequently in the acoustic spectra.

In the LES case the poor wall resolution tends to predict too strong wall pressure fluctuations downstream from the leading edge. The RANS inside the boundary layer within the SAS and DES method suppresses the development of such artificial structures. The RANS method of the boundary layer acts like a low pass filter so that only pure gust noise source can be predicted. URANS can only predict tonal components of the pressure fluctuations caused by the wakes of the turbulence generator struts.

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 employing the Ffowcs Williams and Hawkings analogy fed from source data from the SAS, DES and LES. Again, only the LES was able to predict sound up to realistically interesting frequencies up to 5 kHz. Although the pressure fluctuations near mid span of the blades are over predicted, the overall acoustic predictions fit the measurements well. This is due to the fact that the level of the well predicted surface pressure (at the leading edge) dominates the overall acoustics by far.

ACKNOWLEDGMENT

This work was supported by the German Academic Exchange Service (DAAD) and the German Research Foundation (DFG). We gratefully acknowledge this support.

REFERENCES


[10] F. Menter, T. Esch - *Elements of Industrial Heat Transfer Predictions*. 16th Brazilian Congress of Mechanical Engineering (COBEM), Uberlandia, Brazil, **2001**


[18] M. Schneider - *Der Einfluss der Zuströmbedingungen auf das breitbandige Geräusch eines Axialventilators*. Fortschr.-Ber. VDI Reihe 7, Nr. 478, VDI Verlag, Düsseldorf, **2006**
