APPLICATION OF HYBRID CFD/CAA TECHNIQUE FOR MODELING PRESSURE FLUCTUATIONS IN TRANSONIC FLOWS

SŁAWOMIR DYKAS, WŁODZIMIERZ WRÓBLEWSKI AND SEBASTIAN RULIK

Institute of Power Engineering and Turbomachinery,
Silesian University of Technology,
Konarskiego 18, 44-100 Gliwice, Poland
slawomir.dykas@polsl.pl

Abstract: Solving AeroAcoustics (CAA) problems by means of the Direct Numerical Simulation (DNS) or even the Large Eddy Simulation (LES) for a large computational domain is very time consuming and cannot be applied widely for engineering purposes. In this paper in-house CFD and CAA codes are presented. The in-house CFD code is based on the LES approach whereas the CAA code is an acoustic postprocessor solving non-linearized Euler equations for fluctuating (acoustic) variables. These codes are used to solve the pressure waves generated aerodynamically by a flow over a rectangular cavity and by the vortex street behind a turbine blade. The obtained results are discussed with respect to the application of the presented numerical techniques to pressure waves modeling in steam turbine stages.

Keywords: transonic flow; aeroacoustic noise; hybrid technique

1. Introduction

Compressible Navier-Stokes equations can numerically predict the aerodynamic and acoustic flow fields simultaneously. Solving the Navier-Stokes equations using the DNS or LES methods for capturing both the aerodynamic and acoustic fluctuations is still very time consuming. Since there is a large disparity of the length and time scales between the aerodynamic and acoustic variables, the DES and LES methods are usually used for the source domain, *i.e.* at locations where the aerodynamic disturbances generate noise. For the rest of the computational domain, where the acoustic waves propagate, other methods may be used, *e.g.* the non-linearized Euler equations for fluctuating (acoustic) variables (*e.g.* [1]). In the propagation region it is usually assumed that the flow field does not generate any sound (pressure waves). However, the form of the non-linearized Euler equations applied in the in-house acoustic postprocessor allows such a possibility. The

non-linearized Euler equations can be applied to a wide range of the flow mean Mach numbers and mainly to internal flows. For the computational domain, often referred to as an acoustic source region, where the flow disturbances generate noise, for better modeling of acoustic excitations, the LES method is implemented into the compressible in-house CFD code. The U-RANS in-house method shows an unstable solution for a very fine numerical grid. This CFD code is dedicated to modeling the flow with the flow mean Mach number higher than 0.15.

However, the instabilities which arise in the flow through the turbine stages have a much higher amplitude than ordinary acoustic waves ($\gg 100\,\mathrm{Pa}$). Therefore, the concept of pressure waves together with acoustic waves is used in this paper.

In the flow through the steam turbine stages there are some sources causing excitation at frequencies that are unrelated to rotational frequency. The following may be mentioned for example:

- the unsteady flow separation from stationary walls (stator blades, casing or shaft);
- the unsteady shock and condensation waves in blade passages;
- the vortex shedding from the blades (Figure 1);
- the flow over the cavities (Figure 1).

Figure 1 shows a schematic overview of the last two phenomena occurring during the blade-to-blade channel and the cavity flow with the inlet flow mean Mach number lower than 1. The pressure wave traveling inside the cavity induces the shear layer oscillation, which is the major cause of the propagation of pressure waves.
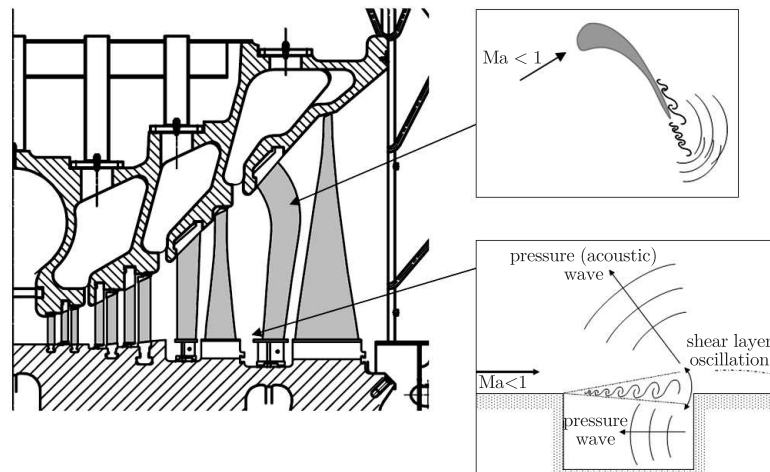

Figure 1. Schematic view of LP steam turbine cross section with marked places of cavity flow and blade-to-blade flow

The pressure waves excited by the cavity flow in the hub region may affect the main flow of steam, especially its wetness fraction.

The in-house CFD/CAA technique has been used for computations of the cavity flow noise applications, where both broadband and tonal noise is emitted. This transonic flow type is commonly found in aviation, turbomachinery, power engineering and even in environmental flows. In the presented results, the aerodynamic and acoustic fields are considered as 2D. This assumption is made in accordance with the experiment [2] and also in order to reduce the computational domain. This paper focuses mainly on the qualitative assessment of the applied techniques and on the description of the problems that have appeared.

The phenomenon of flow induced by noise radiation in cavities has been studied experimentally [2–4] by many researchers for many years. The presented in-house acoustic postprocessor was validated against the analytical solution and experiments.

Additionally, the numerical modeling of the pressure waves generated by the stator blade vortex street is presented. The experimental and numerical investigations of the unsteady phenomena in the flow through the turbine stages have been the focus of the attention of many researchers. A thorough overview of the literature related to this problem is not possible within this conference paper. Therefore, a short description of the physical models applied to the in-house codes together with the results for the cavity flow, as well as for the blade-to-blade flow only are presented in this paper.

The goals of the research presented in this paper are focused on testing the in-house CFD/CAA technique abilities to model the pressure waves generated by the flow over the cavity and by the vortex street.

2. CFD/CAA in-house codes

For both the in-house codes the systems of the governing equations are discretized on a multi-block structured grid using the finite volume method integrated in time with the explicit Runge-Kutta algorithm. An upwind scheme is used with a one-dimensional Riemann solver. The third-order accuracy in space is achieved by means of the MUSCL approach. The second order finite difference scheme for the diffusive fluxes balance is employed.

A detailed description of the used numerical techniques can be found in the works [5, 6].

2.1. CFD code (LES-CFD)

Although solving the U-RANS equations is possible for the CAA computations the authors have decided to develop the in-house Favre-averaged U-RANS CFD code into the application of the Large Eddy Simulation (LES) method. The LES equations for a compressible viscous flow are obtained by a decomposition of the variables of the Navier-Stokes equations into a Favre-filtered part ($^{-}$, $^{~}$), similarly like in the base in-house code [7], and an unresolved part ($'$) that has to

be modeled with a subgrid-scale (SGS) model. The applied Favre-filtered mass, momentum and energy equations may be written in the following differential form:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}_j}{\partial x_j} = 0$$

$$\frac{\partial \bar{\rho}\tilde{u}_i}{\partial t} + \frac{\partial (\bar{\rho}\tilde{u}_i\tilde{u}_j + \bar{p}\delta_{ij})}{\partial x_j} - \frac{\partial \bar{\tau}_{ij}}{\partial x_j} = \frac{\partial \tau_{ij}^{\mathrm{SGS}}}{\partial x_j} \qquad (1)$$

$$\frac{\partial \bar{\rho}\tilde{E}}{\partial t} + \frac{\partial (\bar{\rho}\tilde{E} + \bar{p})\tilde{u}_j}{\partial x_j} + \frac{\partial \tilde{q}_j}{\partial x_j} - \frac{\partial \bar{\tau}_{ij}\tilde{u}_j}{\partial x_j} = \frac{\partial \tau_{i,j}^{SGS}\tilde{u}_j}{\partial x_j}$$

where E is the total specific internal energy. In the code, the Smagorinsky SGS-model is used, where:

$$\tau_{ij}^{\mathrm{SGS}} = 2\mu_{\mathrm{SGS}}\left(S_{i,j} - \frac{1}{3}S_{k,k}\delta_{i,j}\right)$$

$$\mu_{\mathrm{SGS}} = \bar{\rho}C_s^2 V^{2/3}\sqrt{2S_{i,j}S_{i,j}} \qquad (2)$$

$$S_{i,j} = \frac{1}{2}\left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}\right) \qquad \tilde{q}_i = -\tilde{\lambda}\frac{\partial \tilde{T}}{\partial x_i}$$

where μ_{SGS} is the eddy viscosity, $C = 0.18$ is the Smagorinsky constant and V is a volume of a grid cell.

2.2. *Euler Acoustic Postprocessor (EAP-CAA)*

The full non-linear Euler equations are selected for the description of the aerodynamic noise generation and propagation in the flow. These equations are formulated using the decomposition of the actual variables into the CFD flow parts ($_0$) and the fluctuating parts ($'$). The conservative acoustic variables are in this case defined as:

$$\begin{aligned}
\rho' &= \rho - \rho_0 \\
(\rho u)' &= (\rho u) - \rho_0 u_0 \\
(\rho v)' &= (\rho v) - \rho_0 v_0 \\
(\rho w)' &= (\rho w) - \rho_0 w_0 \\
(\rho E)' &= (\rho E) - \rho_0 E_0
\end{aligned} \qquad (3)$$

In the Cartesian coordinates, the full Euler equations have the following form:

$$\frac{\partial \vec{Q}}{\partial t} + \frac{\partial \vec{E}}{\partial x} + \frac{\partial \vec{F}}{\partial y} + \frac{\partial \vec{G}}{\partial z} = 0 \qquad (4)$$

where the vectors of conservative variables and fluxes can be written as follows:

$$\vec{Q} = \begin{bmatrix} \rho' \\ (\rho u)' \\ (\rho v)' \\ (\rho w)' \\ (\rho E)' \end{bmatrix}$$

$$\vec{E} = \begin{bmatrix} (\rho u)' \\ (\rho u)'u' + p' + \rho_0 u_0 u' + (\rho u)'u_0 \\ (\rho v)'u' + \rho_0 v_0 u' + (\rho v)'u_0 \\ (\rho w)'u' + \rho_0 w_0 u' + (\rho w)'u_0 \\ ((\rho E)' + p')u' + \rho_0 E_0 u' + (\rho E)'u_0 \end{bmatrix}$$

$$\vec{F} = \begin{bmatrix} (\rho v)' \\ (\rho u)'v' + \rho_0 u_0 v' + (\rho u)'v_0 \\ (\rho v)'v' + p' + \rho_0 v_0 v' + (\rho v)'v_0 \\ (\rho w)'v' + \rho_0 w_0 v' + (\rho w)'v_0 \\ ((\rho E)' + p')v' + \rho_0 E_0 v' + (\rho E)'v_0 \end{bmatrix} \tag{5}$$

$$\vec{G} = \begin{bmatrix} (\rho w)' \\ (\rho u)'w' + \rho_0 u_0 w' + (\rho u)'w_0 \\ (\rho v)'w' + \rho_0 v_0 w' + (\rho v)'w_0 \\ (\rho w)'w' + p' + \rho_0 w_0 w' + (\rho w)'w_0 \\ ((\rho E)' + p')w' + \rho_0 E_0 w' + (\rho E)'w_0 \end{bmatrix}$$

The relations for the primitive fluctuating variables in the function of the conservative fluctuating variables and the CFD values can be written in the form:

$$u' = \frac{(\rho u)' + \rho_0 u_0}{\rho' + \rho_0} - u_0$$

$$v' = \frac{(\rho v)' + \rho_0 v_0}{\rho' + \rho_0} - v_0$$

$$w' = \frac{(\rho w)' + \rho_0 w_0}{\rho' + \rho_0} - w_0 \tag{6}$$

$$p' = (\gamma - 1)\left[(\rho E)' - \frac{1}{2}\big(\rho_0 u_0 u' + (\rho u)'u_0 + (\rho u)'u_0\big) \right.$$
$$- \frac{1}{2}\big(\rho_0 v_0 v' + (\rho v)'v_0 + (\rho v)'v_0\big)$$
$$\left. - \frac{1}{2}\big(\rho_0 w_0 w' + (\rho w)'w_0 + (\rho w)'w_0\big) \right]$$

The presented Euler acoustic postprocessor is dedicated to the acoustic field assessment in the flows with a wide range of Mach numbers, especially for transonic internal flows. The Euler acoustic postprocessor should be used in the part of the computational domain where the CFD grid is not fine enough to capture the fluctuations in a correct way. Therefore, the computational domain should usually be split into the CFD and CAA parts. In the case of internal flows, such as turbine channels, the same numerical grid can be used for both analyses; on the basis of the CFD results, the EAP-CAA can be used in the regions where the grid is coarser using the CFD (state "0" in Equations (4)–(6)) solution as boundary conditions.

3. Numerical results

The systems of flow governing equations are solved on multi-block structured grids. The computational domains of the cavity and blade-to-blade steam turbine stator channel are discretized and used for numerical analysis.

3.1. Cavity flow

The cavity dimensions analyzed here are assumed according to Ahuja and Medoza experiments [2] (Figure 2). The cavity is $31.75\,\mathrm{mm}$ long and $12.7\,\mathrm{mm}$ deep. The Length/Depth ratio is equal to 2.5, whereby the acoustic field can be considered as 2D. Therefore, 10 control volumes only are assumed in the third direction of the cavity width. It allows a reduction in the computational time.

In experiments, the jet flow over the cavity has a velocity of $\mathrm{Ma} = 0.53$. For the calculations, the total parameters at the inlet are assumed as $p_0 = 121\,082.8926\,\mathrm{Pa}$ and $T_0 = 316.854\,\mathrm{K}$, whereas the static pressure for the outlet and the extrapolation boundary conditions are $100\,000\,\mathrm{Pa}$.

The in-house CFD code is used for the LES modeling. The computational domain and the boundary conditions used for the modeling are presented in Figure 2. The structured numerical mesh comprising 7 blocks has $\sim 0.6\,\mathrm{M}$ grid points for each XY plane. For the far-field boundaries, the non-reflecting boundary conditions based on the extrapolation of the flow variables are applied.

The inlet is placed at $22.86\,\mathrm{mm}$ from the cavity leading edge and it is $12.7\,\mathrm{mm}$ high.

The transient modeling is performed with a time step of $10^{-8}\,\mathrm{s}$. Figure 3 shows the acoustic pressure field obtained from the LES calculations. Acoustic waves are mainly generated by the shear layer oscillation above the cavity, close to its trailing edge, and they propagate rather upstream.

The acoustic waves generated directly from the CFD and by means of the CAA postprocessor are analyzed in respect of the sound pressure level and their frequency. The amplitude of the generated pressure waves is in the range of $\sim 1000\,\mathrm{Pa}$, which is a relatively high value. The SPL spectrum in a far field was presented in the experiment of Ahuja and Mendoza [2]. Figure 4 shows a comparison of the SPL spectrum determined from the LES and EAP results with the experimental spectrum at a point located at the azimuthal angle of $90°$.

However, the distance of this point from the cavity in the experiment was about $3.6\,\mathrm{m}$, whereas it is only $0.36\,\mathrm{m}$ in the case of the presented calculations. This certainly affected the differences in the value of the Sound Pressure Level (SPL), which is visible in Figure 4. Moreover, the comparison of the SPL spectrum shows that it is possible, by means of the in-house codes, to predict every other main frequency measured in the experiment. This fact may result from the assumption of the 2D character of the calculated phenomenon – limiting the size of the computational domain widthwise.

3.2. Blade-to-blade channel flow

The experimental determination of the instantaneous pressure field in the turbine blade-to-blade channel flow is extremely difficult, especially as it deals with high frequency pressure fluctuation.

The geometry of a low-pressure steam turbine stator is chosen for the blade-to-blade flow. The boundary conditions at the inlet and outlet correspond to the
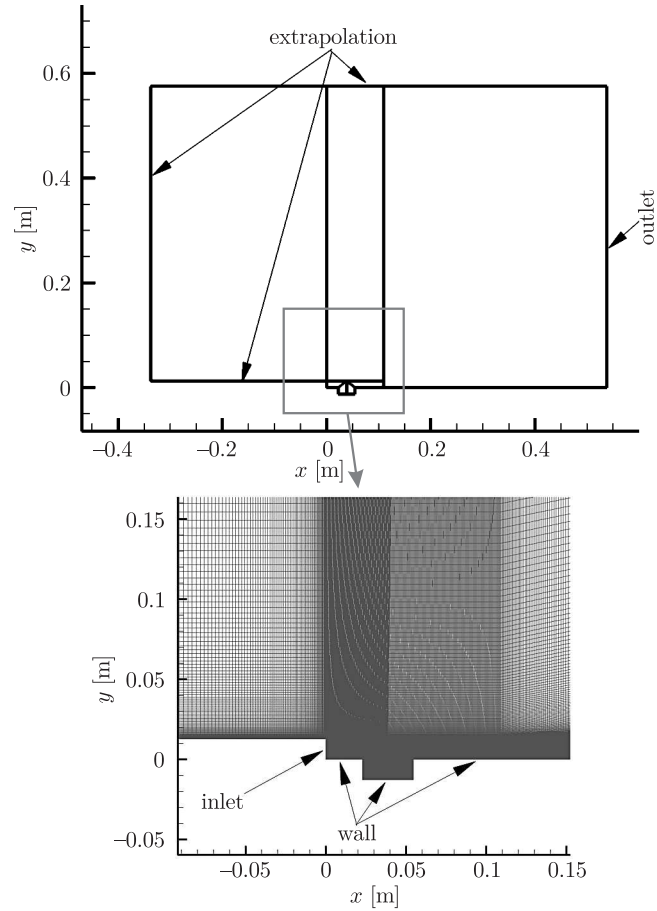
Figure 2. Computational domain of cavity and boundary conditions for LES-CFD modeling
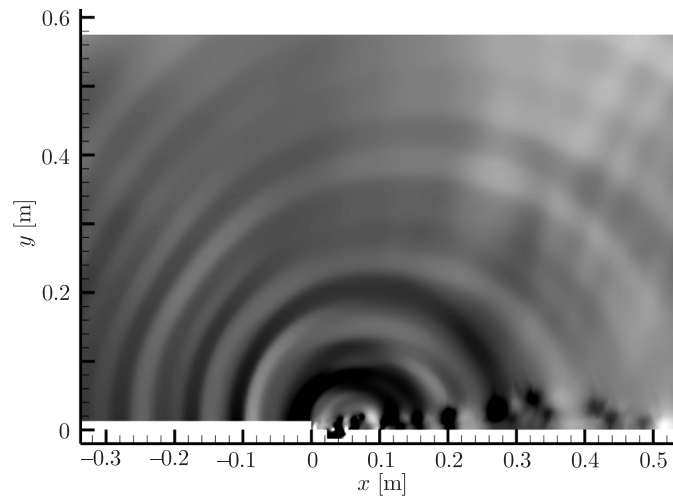

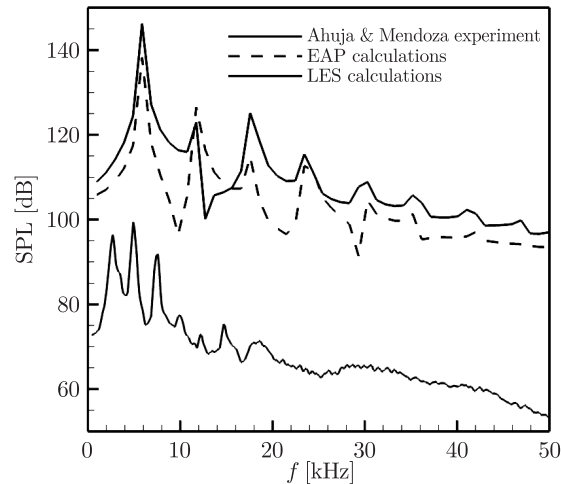**Figure 3.** Snapshot of acoustic pressure field from the LES computations

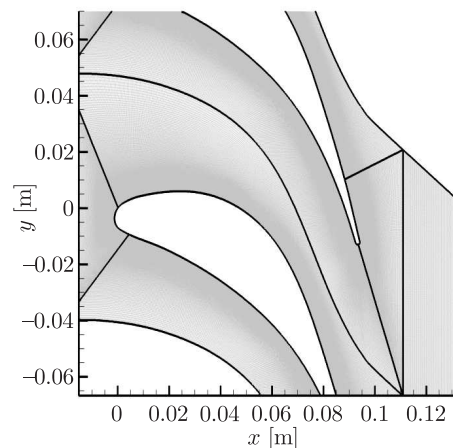**Figure 4.** Comparison of sound pressure level (SPL) far from cavity

Figure 5. Computational domain and numerical grid of blade-to-blade channel

outlet Mach number $Ma = 1.2$. The computational domain is presented in Figure 5. The applied computational grid comprises $0.5\,\text{M}$ grid points for each XY plane.

A transient analysis was performed for a constant time step of $10^{-9}\,\text{s}$. The photograph of the pressure waves generated by the blade-to-blade flow is presented in Figure 6. It is visible that these pressure waves are reflected off the suction side of the blade and there is no reflection off the outlet boundary.

The FFT analysis for points P_1 and P_2 was carried out for $1\,\text{ms}$ and its results are presented in Figure 7. In this case, the pressure fluctuation in the function of frequency is presented as no experimental data of the SPL are available. The frequency of the first harmonic is high and amounts to about $30\,\text{kHz}$, whereas its amplitude is about $100\,\text{Pa}$. However, the amplitude of the pressure fluctuation is in the range of $1000\,\text{Pa}$ (Figure 6).
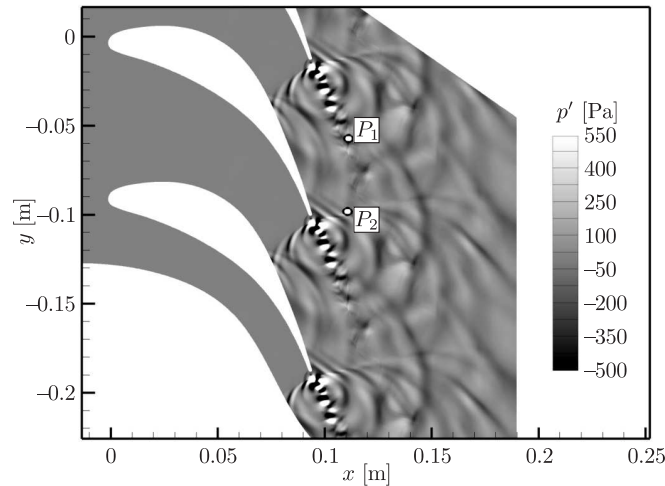
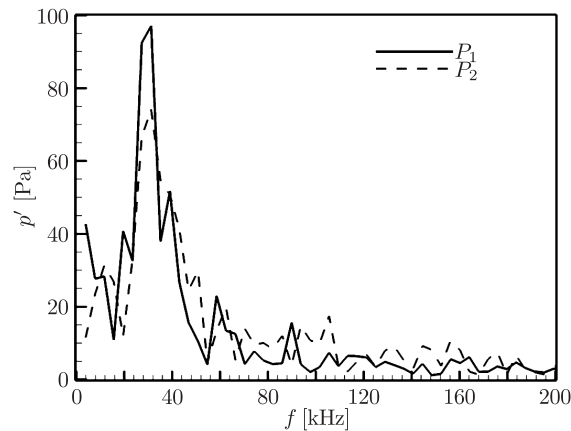**Figure 6.** Snapshot of pressure waves in blade-to-blade channel

Figure 7. The FFT analysis for two points behind the blade-to-blade channel

4. Conclusions

This paper deals with the numerical prediction of the cavity and blade-to-blade flow noise using in-house CFD and CAA codes. The main target of the presented calculations is a qualitative estimation of the applied techniques in respect of the acoustic waves generation, their frequency and amplitude. Attention is focused mainly on the computational time assessment and the determination of the boundary conditions for the coupling of CFD and CAA computations. Both computations are carried out using the same size of the computational domain.

The following conclusions may be drawn from the conducted analysis:

- The in-house CFD-LES calculations require a much finer numerical grid in order to capture more spatial and time scales.
- The presented acoustic postprocessor features a good ability to calculate the pressure wave propagation.

- Due to the common use of the in-house CFD/EAP numerical techniques it is possible to capture both tonal and broadband pressure fluctuations saving the computational time.
- The amplitude of the pressure waves generated in the flow through the turbine stages is in the range of 100–1000 Pa, whereas their frequency is very high, reaching tens of kHz.
- The frequency of the pressure waves should not coincide with the natural frequency of the blade system.
- The amplitude of the modeled pressure waves is significant, compared to the mean pressure value prevailing in the LP steam turbine.

These results show a good qualitative agreement between the acoustic wave modeling performed using the in-house CFD and CAA techniques. Instantaneous flow variables from the CFD calculations in the whole computation domain should be used for a CAA postprocessor for the aerodynamic noise assessment in a near field. The application of the extrapolation boundary conditions ensures no reflection of acoustic waves.

Acknowledgements

References

[1] Dykas S, Wróblewski W, Rulik S and Chmielniak T 2010 *Archives of Acoustics* **35** (1) 35
[2] Ahuja K K and Mendoza J 1964 *Effects of Cavity Dimensions, Boundary Layer, and Temperature on Cavity Noise with Emphasis on Benchmark Data to Validate Computational Aeroacoustic Codes*, NASA CR 4653
[3] Weyna S 2005 *Acoustic Energy Dissipation of Real Sources*, WNT, Warsaw (in Polish)
[4] Takeda K and Shieh C M 2013 *Int. J. Comput. Fluid Dynamics* **18** (6) 439
[5] Wróblewski W, Dykas S, Gardzilewicz A and Kolovratnik M 2009 *ASME J. Fluids Engineering* **131** 041301-1
[6] Dykas S, Rulik S and Wróblewski W 2012 *Open J. Fluid Dynamics* **2** 65
[7] Wróblewski W 2000 *Numerical Simulation of Flow Phenomena in Turbines*, Silesian University of Technology Publishing, Energetyka, **132** (in Polish)