

ON LINEAR MODELLING OF AXIAL VELOCITY DENSITY RATIO DEVELOPMENT IN COMPRESSOR BLADE CASCADES

T. Kreuzová^{1,2}, D. Šimurda¹, P. Šafařík^{1,2}

¹ Institute of Thermomechanics of the Czech Academy of Sciences, Prague, Czech Republic

² Department of Fluid Dynamics and Thermodynamics, Faculty of Mechanical Engineering, Czech Technical University in Prague, Technická Street 4, 16607, Prague 6, Czech Republic

Abstract

Flow field in transonic compressor blade cascade is examined by means of numerical simulations. Quasi-3D simulations with an influence of axial velocity density ratio (AVDR) are performed. Results of the simulations are compared to data available in literature. It is shown that modelling of AVDR as a linear function does not provide accurate predictions.

Keywords: blade cascade, compressor, transonic

1 Introduction

With the latest concurrent development in the fields of transonic axial compressors design and of numerical analysis of fluid flow, simulations of the behavior of the mentioned devices are not only desirable, but also accessible. In this work, simulations of the flow field in the blade cascades described and experimentally examined in [1] are performed in order to explore an ability of a simplified quasi 3D model to describe flow in a linear blade cascade.

Experiments on blade cascades should ideally meet two requirements: perfect periodicity and perfect planarity of the flow. Periodicity is not discussed in this paper. The planarity is violated by growth of boundary layer on the side walls of a test section. The deviation from the planarity is quantified by a parameter called Axial Velocity Density Ratio (AVDR).

In elementary numerical simulations blade cascade flows are modelled as a two dimensional, the domain consists of one blade or passage with periodic boundary conditions along appropriate streamline ensuring the "infiniteness" of the blade cascade. These simulations can be extended to quasi-3D by including influence of AVDR. To simulate effects of $AVDR \neq 1$ and keep computational effectiveness of two dimensional simulations, modifications of 2D approach have to be applied. All of those consist of modifications of the mass flux in the domain.

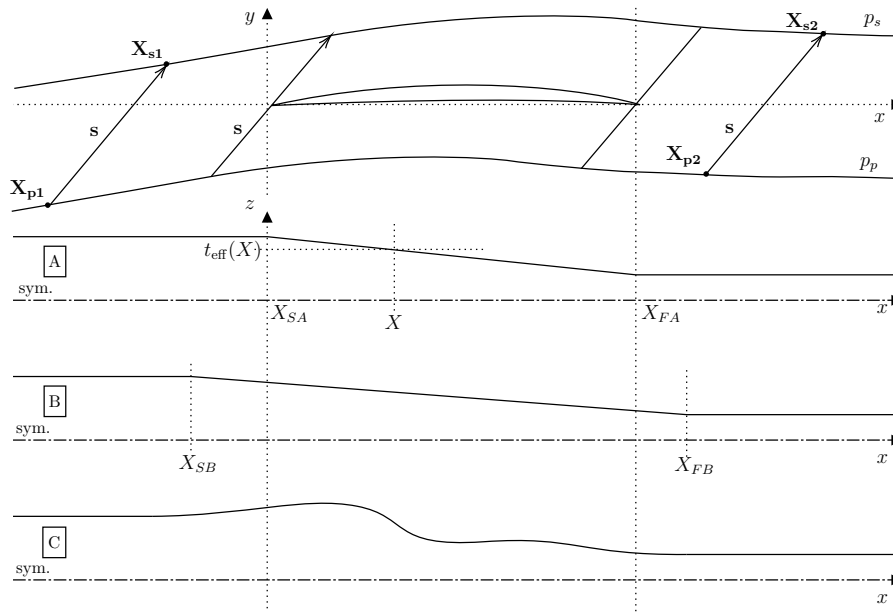


Figure 1: Illustration on mesh shapes with consideration of $\Omega_2 \neq 1$

The most illustrative approach to AVDR modelling seems to be the modification of streamtube thickness which mimics the growth of the displacement thickness of the boundary layer. The artificial side walls are then modelled as a "slip wall" ($u_n = 0, \partial^p / \partial n = 0$). This approach is, for its demonstrativeness, adopted in this work. Sketches of some possible configurations of the change of mesh thickness are shown in Fig. 1. In this image, vertical dimension and AVDR of the whole cascade (Ω_2) are substantially magnified to ensure good readability of the schemes.

Other approaches are usually based on continuous change of mass flow rate by source terms or boundary conditions [2]. Those are more suitable for interactive modelling of AVDR development with dependency on current flow field or of systems with e.g. boundary layer suction.

The overall velocity density ratio in the domain is defined as a ratio of mass flux through traversing plane and the inlet mass flux and is denoted as Ω_2

$$\Omega_2 = \frac{\rho_2 \|\mathbf{U}_2\| \cos \alpha_2}{\rho_1 \|\mathbf{U}_1\| \cos \alpha_1}. \quad (1)$$

AVDR is an integral parameter and can be defined at any plane parallel to the plane of leading edges of the cascade σ at the distance x . When integrating along the plane $\varphi(x)$ between its intersections with p_p and p_s (see Figs. 2 and 1):

$$\Omega(x) = \frac{1}{\|\mathbf{s}\|} \int_{\varphi(x) \cap p_p}^{\varphi(x) \cap p_s} \frac{\rho(y) \|\mathbf{U}(y)\| \cos \alpha(y)}{\rho_1 \|\mathbf{U}_1\| \cos \alpha_1} dy \quad (2)$$

To describe the variation of effective thickness of the domain, local contraction factor $\psi(\mathbf{x})$ is introduced. This parameter is given as a ratio of the domain thickness at the inlet (t_1 , it is assumed constant in this work) reduced by displacement thickness δ^* of the side walls boundary layer and the effective domain thickness t_{eff} at $\mathbf{x}$

$$\psi(\mathbf{x}) = \frac{t_1 - 2\delta_1^*}{t_1 - 2\delta^*(\mathbf{x})} = \frac{t_{\text{eff}1}}{t_{\text{eff}}(\mathbf{x})}. \quad (3)$$

In this work only $\Omega_2 > 1$ is studied, but the effective thickness of the domain can be locally increased ($\nabla \psi(\mathbf{x}) < 0$) and even $\psi(\mathbf{x}) < 1 \Leftrightarrow \delta^*(\mathbf{x}) < \delta_1^*$ is allowed. Example of such situation is

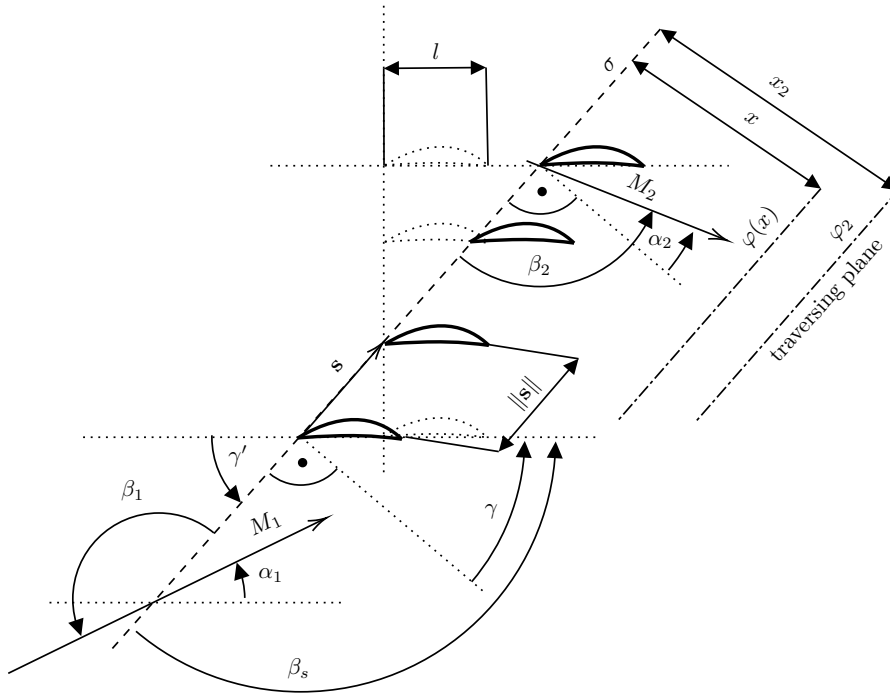


Figure 2: Geometry of a blade cascade

illustrated in scheme C in Fig. 1. $\psi(\mathbf{x})$ only needs to meet the condition of periodicity: $\forall \mathbf{X}_P \in p_P, \mathbf{X}_S \in p_S, \overrightarrow{\mathbf{X}_P \mathbf{X}_S} = \mathbf{s} : \psi(\mathbf{X}_P) = \psi(\mathbf{X}_S)$. Vector $\mathbf{s}$ is shown in Fig. 2 and is defined by stagger angle β_s and the pitch $\|\mathbf{s}\|$.

Proper definition of $\psi(\mathbf{x})$ is crucial for correct capturing of the behavior of the flow. Even the most straightforward approach - linear change of the mesh thickness (cases A and B in Fig. 1) - requires definition of the beginning (X_S ¹) and the end (X_F) of the range. Contrary to quite popular approach (e.g. [5]), aligning X_S and X_F with the leading (resp. trailing) edge of the blade (scheme A in Fig. 1) does not guarantee "reality corresponding" results.

The geometry of the studied cascade is characterized by the shape of the suction and pressure side of the profile, chord ($l = 0.09\text{ m}$), stagger angle ($\beta_s = 138.51^\circ$) and pitch-chord ratio ($\|\mathbf{s}\|/l = 0.621$). These and accompanying parameters are illustrated in the Fig. 2. The design point of the cascade is characterized by inlet Mach number $\text{Ma}_1 = 1.1$ and inlet flow angle $\beta_1 = 148.50^\circ$. In [1] the tests were performed for $\text{Ma}_1 \in [0.74; 1.10]$ and $\beta_1 \in [145.0; 151.5]^\circ$ with various values of AVDR ($\Omega_2 \in [0.9; 1.34]$).

2 Numerical Simulations

The flow in the cascade is modeled as a turbulent flow of a compressible gas, which is assumed to be ideal (Eqs. 4 are closed by equation of state $p = \rho r T$).

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \quad (4a)$$

$$\frac{\partial (\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U}) = \nabla \cdot \tau_{eff} \quad (4b)$$

$$\frac{\partial (\rho E)}{\partial t} + \nabla \cdot [(\rho E + p) \mathbf{U}] = \nabla \cdot [\tau_{eff} \cdot \mathbf{U}] + \nabla \cdot (\lambda_{eff} \nabla T) \quad (4c)$$

where τ_{eff} is the effective stress tensor, λ_{eff} is the effective thermal conductivity and E is the specific total energy.

Navier-Stokes equations are discretized by finite volumes method. The steady state solution is obtained with the implicit Euler discretization. For the solution of the resulting system of algebraic equations lower-upper symmetric Gauss-Seidel scheme is used (solver is described in [3]). For modelling of the turbulent flow SST k- ω turbulence model [4] is utilized.

Perfectly periodic and quasi-three dimensional flow is considered. A blade-centered (the periodic boundaries are positioned between two blades) multi-block hexahedral mesh with refinement in the vicinity of the wall (target $y^+ = 1$) is created in a manner allowing variable AVDR of the flow. The mesh consists of one cell in the third (z in Fig. 1) direction, but it has variable thickness ($t_{eff}(\mathbf{x})$) in this dimension. This thickness represents local $\psi(\mathbf{x})$; $t_{eff}(\mathbf{x}) = t_{eff1} / \psi(\mathbf{x})$.

Boundary conditions are prescribed with respect to the physical nature of the system. The domain (again, slightly deformed, similarly to sketches in Fig. 1) is illustrated in Fig. 3. At the inlet constant total pressure (p_{01}), constant total temperature (T_{01}) and constant direction of the flow ($\alpha_1 = \beta_1 - \beta_s$) are prescribed, while at the outlet only constant pressure (p_2) is given. In terms of turbulence modelling turbulent intensity ($Tu = 3.3\%$) and turbulence length scale ($Tu_L = 4 \cdot 10^{-3}\text{ m}$) are assumed at the inlet. At the wall of the blade zero velocity and zero gradient in the direction of normal are prescribed for T and p . On the side walls, "slip" condition is prescribed ($U_n = 0$, zero gradient in the direction of normal for pressure and temperature). On the patches represented by p_s and p_p in Figs. 1 and 3 periodicity with separation vector $\mathbf{s}$ is required.

Mesh Independence Study was performed for the regime $\text{Ma}_1 = 1.0, \beta_1 = 148.50^\circ, \Omega_2 = 1.1$ with linear change of thickness between leading and trailing edge. Three (fine, medium and coarse) meshes with refinement factor $\sqrt{3}$ in one direction (leading to a triplicating of the number of cells in the domain) were considered. With total pressure loss (ω [-], Eq. 5) as the target quantity the Grid Convergence Index (GCI, a measure of how far the value obtained with given grid is from an asymptotic value, see [6]) with safety factor 3.0 of the medium grid with $320 \cdot 10^3$ cells is 0.002 and

¹While $\mathbf{x}$ denotes one specific point in the domain, x stands for distance from σ .

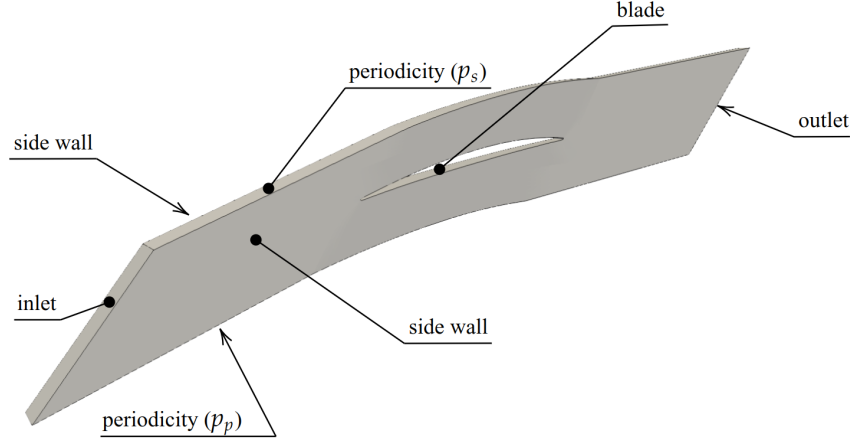


Figure 3: Domain with labeled patches

of the coarse grid is 0.007. The resulting total pressure loss obtained on coarse and fine grid differ by less than 1.5 % and the difference in position of shock wave is also less than 1.5 % of chord l . Therefore, for most computations, coarse mesh was chosen in spite of computational efficiency.

3 Results and Discussion

Representative values of variables of the outlet flow (U_{x2} , U_{y2} , ρ_2 , p_2 , T_2) are obtained by data reduction [7]. Values of integral characteristics, such as total pressure loss ω

$$\omega = \frac{p_{01} - p_{02}}{p_{01} - p_1} \quad (5)$$

and blade pressure coefficient C

$$C = \frac{p_{01} - p}{p_{01} - p_1} \quad (6)$$

are derived from these representative data.

Data $(p_1/p_2, \beta_2, \omega)$ for a wide variety of cases ($\alpha_1 \in \{147.0, 150.0, 151.5\}^\circ$, $\text{Ma}_1 \in \{0.82, 0.92, 1.02, 1.10\}$) are given in [1]. These data were interpolated or, where the interpolation is given, the original fit was used. Results of simulation of selected cases are compared with interpolated data in Table 1.

Table 1: Comparison with experiment

	case			ω		β_2	
	Ω_2	β_1	Ma_1	exp.	CFD	exp.	CFD
1	1.00	147.00	1.02	0.054	0.055	137.08	137.32
2	1.05	147.00	1.02	0.046	0.045	136.27	136.72
3	1.10	150.00	1.06	0.077	0.078	136.00	136.00
4	1.20	151.50	1.10	0.133	0.122	136.12	135.32

An instability for $\Omega_2 \approx 1$, $\beta_1 = 151.5^\circ$ is commented on in the article [1] and is also predicted by numerics. The comparison suggests that the numerical simulations can predict the ω and β_2 quite satisfactory. Also the position of the shock waves in the channel is captured well. Schlieren images are compared (their overlay is shown) in Fig. 4. As the periodicity in the numerical solution is inherent, the figure also illustrates that the periodicity in the experimental setup is not perfect.

Blade pressure coefficient distribution for design point regime with $\Omega_2 = 1.0$ obtained numerically is compared with the experimental data in Fig. 5a. The slight shift of the numerical data to the lower values and the absence of slight bulging of the distribution on the suction side can have - apart from errors in CFD calculation - two reasons. The 0.02 difference in Ω_2 can cause, at least

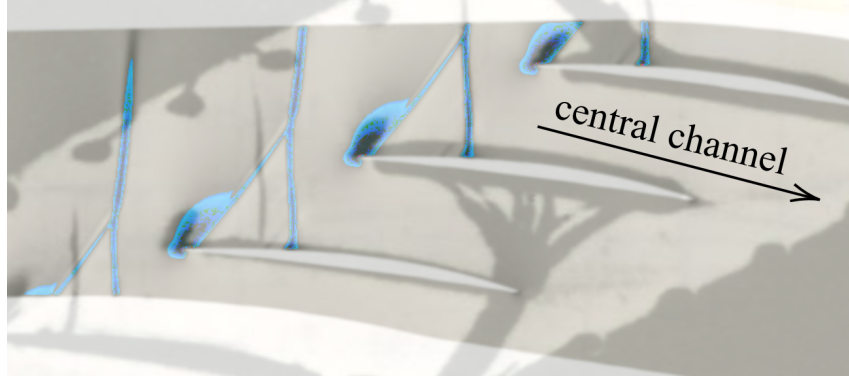


Figure 4: Comparison of CFD (blue) and experiment (black and white)
(schlieren image - $Ma_1 \approx 0.92$, $\beta_1 = 148.5^\circ$, $\Omega_2 \approx 1.10$)

part of this difference. Following experimental data, the distribution of C (Eq. 6) is shifted and bulged to higher values with increasing Ω_2 . As sensitivity of transonic compressors to changes of boundary conditions is a well known problem, this reason should not be condemned. Another possible reason is that for this particular experimental data the equivalency $\Omega_2 = 1 \Leftrightarrow \psi(\mathbf{x}) = 1$ does not hold. This would indicate that the linear approximation of $\psi(\mathbf{x})$ does not fit the experiment well and better model according to properties of boundary layer control system used in the wind tunnel, where measurement was carried out, should be used.

Pressure field at the design regime is shown in the Fig. 6a. The asymptote of the detached shock wave and the streamline are at the angle of 65° which corresponds to a value that can be assessed theoretically.

In Fig. 5b blade pressure coefficient distributions are compared for a working point with lower inlet Mach number and $\Omega_2 \approx 1.10$ (point for which mesh independence study was performed). For CFD calculation X_S and X_F were aligned with the leading (resp. trailing) edge. In the whole subsonic region, the numerically obtained profile corresponds with the data with $\Omega_2 = 1.04$ well and differs in the supersonic region with expansion. This can, again, indicate that linear approximation of $\psi(\mathbf{x})$ is not fully correct especially for flow fields with shock waves as even a decrease of $\psi(\mathbf{x})$ can appear in the supersonic expansion region.

Measurements of boundary layer thickness on side walls of a high speed wind tunnel during compressor blade cascade testing are documented in [8]. These illustrate the same trends as has been commented in previous paragraphs. In supersonic expansion the local AVDR slightly decreases and after the strong shock wave grows rapidly. Also contours of possible profile of K_f ,

$$K_f(\mathbf{x}) = \frac{\Omega(\mathbf{x}) - 1}{\Omega_2 - 1} \quad (7)$$

$$\Omega(\mathbf{x}) = \frac{\rho(\mathbf{x}) \|\mathbf{U}(\mathbf{x})\| \cos \alpha(\mathbf{x})}{\rho_1 \|\mathbf{U}_1\| \cos \alpha_1} \quad (8)$$

presented in [8] indicate that $\Omega(\mathbf{x})$ increases by 50 - 70 % along the blade and the rest is gained between the blade and traversing plane which is in an opposition to the approach A in Fig. 1. Notice that as the flow field is not uniform, $\Omega(\mathbf{x})$ does not directly represent the domain contraction.

In the region of supersonic expansion across the operating modes a slight glitch appears in the pressure distribution. It is connected to a Mach wave which is formed here. It is not result of a discontinuity in the derivation of the shape of the profile, as there is no such discontinuity present in this region. The fact that this is not documented by experimental data can be justified by their sparseness and does not necessarily mean such phenomenon is not present in the experimentally tested cascade.

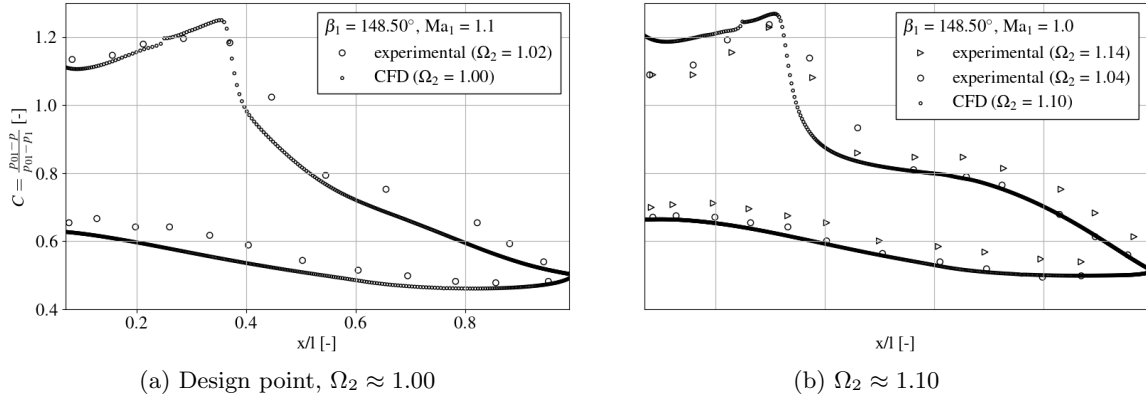


Figure 5: Blade pressure coefficient distribution

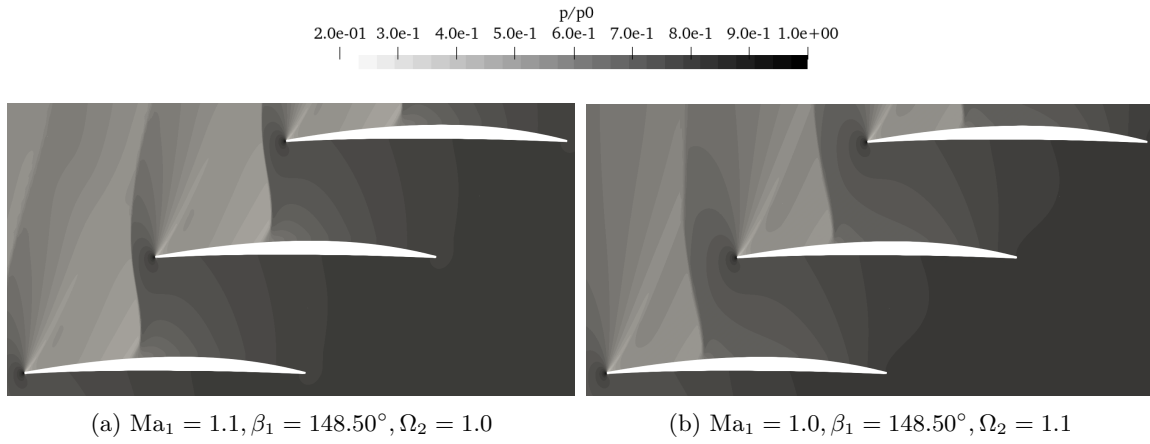


Figure 6: Image of flow field

4 Conclusions

Numerical analysis of flow field in transonic compressor blade cascade was performed. To take AVDR into consideration quasi three dimensional model was employed. The effect of the AVDR was modelled by changing of thickness of the streamtube.

Following results of performed simulations and their comparison with experimental data it can be concluded that the equivalency $\Omega_2 = 1 \Leftrightarrow \psi(\mathbf{x}) = 1$ does not need to hold in an experimental setup. This also implies that modelling $\psi(\mathbf{x})$ as a linear function might not be sufficient. Also, small differences in Ω_2 can have significance.

With focus on strong sensitivity of transonic compressors to even slight changes of boundary conditions these results suggest that modelling of blade cascade flows with $\Omega_2 \neq 1$ or with suction by passive - non-interactive - quasi 3D models might not be sufficient if not only overall quantitative results are desired, but also information about data such as pressure distribution are required.

Two ways are possible to improve modelling of flows in compressor blade cascades. The first one is to model the flow as three dimensional. While this seems quite straightforward it might not be the optimal approach, especially for larger studies with focus on more working points, for its great computational expense, requirements on the mesh quality, etc. Therefore a second option, an interactive way to model contraction of the streamtube, comes to use. The computational domain is then modelled as purely two dimensional and the contraction can be modelled by source terms dependent on an actual flow field. If the specific simulation of blade cascade flow is supposed to mimic a measurement, focus should be placed on resolution of suction system of a specific experimental setup and therefore the distribution of $\psi(\mathbf{x})$ should be modelled as dependent not only on the flow field, but also on the boundary conditions given by the setup.

Acknowledgment

The research was supported by the Czech Ministry of Education Youth and Sports under the project Inter-Excellence LUAUS23231 Origins and mechanisms of flutter and non-synchronous vibration in modern turbomachines operating at wide range of regimes.

It was also supported by the Grant Agency of the Czech Technical University in Prague, grant No. SGS23/105/OHK2/2T/12.

Computational resources were provided by the e-INFRA CZ project (ID:90254), supported by the Ministry of Education, Youth and Sports of the Czech Republic.

References

- [1] Schreiber, H. A. & Starken, H.: Experimental Cascade Analysis of a Transonic Compressor Rotor Blade Section. *Journal of Engineering for Gas Turbines and Power*, (1984) pp. 288-294.
- [2] Chen, Z., Zhou, M., Xu, Q. & Huang, X.: A Novel Quasi-3D Method for Cascade Flow Considering Axial Velocity Density Ratio. *International Journal of Turbo & Jet-Engines* (2018) pp. 81-94.
- [3] Fürst, J. Development of a coupled matrix-free LU-SGS solver for turbulent compressible flows. *Computers & Fluids*, (2018).
- [4] Menter, F. & Kuntz, M. & Langtry, RB. Ten years of industrial experience with the SST turbulence model. *Heat and Mass Transfer*, (2003)
- [5] Schreiber, H. A. & Starken, H. On the definition of the axial velocity density ratio in theoretical and experimental cascade investigations. in MTT0681 Lyon (1981).
- [6] Roache, P. J. Perspective: A Method for Uniform Reporting of Grid Refinement Studies. *Journal of Fluids Engineering*, (1994) pp. 405-413.
- [7] Amecke, J. & Safarik, P.: Data Reduction of Wake Flow Measurements with Injection of an other Gas. *DLR-Forschungsbericht*, (1995) pp. 95-32.
- [8] Safarik, P. & Prochazka, P.: A contribution to a method of blade cascade measurement with consideration of contraction factor (Tech. report Z-1091/88). Institute of Thermomechanics of the CAS, (1988).