NUMERICAL METHOD FOR AN ASSESSMENT OF STEADY AND MOTION-EXCITED FLOWFIELDS IN A TRANSONIC CASCADE WIND TUNNEL

Atsushi Tateishi, Toshinori Watanabe, Takehiro Himeno, Seiji Uzawa
The University of Tokyo
tateishi@aero.t.u-tokyo.ac.jp
Tokyo, Japan

ABSTRACT
This paper presents a numerical method and its application for an assessment of the flow field inside a wind tunnel. A structured CFD solver with overset mesh technique is developed in order to simulate geometrically complex configurations. Applying the developed solver, a whole transonic cascade wind tunnel is modelled and simulated by a two-dimensional manner. The upstream and downstream periodicity of the cascade and the effect of the tunnel wall on the unsteady flow field are focused on. From the steady flow simulations, the existence of an optimum throttle position for the best periodicity for each tailboard angle is shown, which provides appropriate aerodynamic characteristics of ideal cascades in the wind tunnel environment. Unsteady simulations with blade oscillation is also conducted, and the difference in the influence coefficients between ideal and wind tunnel configurations becomes large when the pressure amplitude increases on the lower blades.

INTRODUCTION
A detailed knowledge of the characteristics of motion-excited aerodynamic force is essential for understanding and predicting an aeromechanical behaviour in turbomachinery. In order to measure motion-excited aerodynamic force (often referred to as unsteady aerodynamic force), a number of researches have been conducted so far using linear cascade wind tunnels [1]. A typical way for obtaining unsteady aerodynamic force is to measure the responses of flow field and aerodynamic force acting on the airfoils under prescribed blade motion. Such data are used for validations of numerical model for predicting and optimizing blade vibration characteristics during the design stage of turbomachinery [2].

The operating conditions of the wind tunnels are carefully controlled to realize flow field similar to ideal infinite cascade (i.e. pitchwise periodicity) before detailed aerodynamic measurement [3-4]. Therefore, establishing a guideline for controlling the wind tunnel is beneficial for gathering data over a wide range of flow conditions. In addition, the cascade wind tunnels often have different geometrical details from an ideal infinite cascade, such as finite number of airfoils, tailboards, and suction mechanisms. Thus, the differences that can arise from these real geometrical features should be known in detail when a comparison is made between wind tunnel measurement and numerical simulation results.

Some past studies focuses attention on the effects of wind tunnel geometry on steady flow fields and unsteady aerodynamic force characteristics. Lepicovsky et al. [5] showed the importance of tailboard geometry on the periodicity of steady flow field through experimental and numerical assessments. Buffum and Fleeter [6] discussed the deterioration of uniformity in unsteady pressure coefficient for traveling-wave-mode oscillation, with focusing on propagating wave direction and its interaction with the wind tunnel wall. Later they reported the effect of acoustic mode in the wind tunnel on the measured aerodynamic influence coefficients [7]. Ott et al. [8] conducted a numerical study for extracting the effect of tailboard on the steady and unsteady flow field in a transonic turbine cascade. All the studies reported the tailboard or wind tunnel wall have significant effect on the steady and motion-excited flow fields.

The aim of this study is to develop a numerical method for an assessment of the flow field inside a cascade wind tunnel for establishing a basic procedure for controlling its flow field. The modelling of whole wind tunnel and parametric study of its geometrical setup are enabled by using overset mesh technique. Using the developed method, the steady and unsteady simulations of the whole wind tunnel are conducted with focusing on the periodicity of the cascade section and the effects of tunnel wall on the motion-excited flow field.

TRANSONIC CASCADE WIND TUNNEL
An analysis target for this study is a transonic cascade wind tunnel in the University of Tokyo. This wind tunnel is designed for aeroelastic investigations of fan or compressor cascade, and it had been used for fundamental researches on
the unsteady aerodynamic force characteristics [9-10] and active suppression of cascade flutter [11]. Fig. 1 (a) shows schematics of the wind tunnel. It can be operated from subsonic to supersonic inflow up to Mach 1.6 by changing the nozzle geometry. The test cascade is equipped between upper and lower bypass passage. Two tailboards with throttle isolate the test section from the bypass area. The back pressure downstream the cascade is controlled by changing the opening angle of the throttle. In order to avoid unexpected choking, the suction mechanism, composed by a porous plate on a cavity connected to a vacuum chamber, is installed at the lower wall of the test section.

The geometrical parameters for controlling the wind tunnel are summarized in Fig. 1 (b). The geometry of the supersonic nozzle is designed to realize the inflow speed of Mach 1.2 and fixed throughout this study. The \( \theta_{t_0} \), \( \theta_{t_3} \), and \( \theta_{t_{18}} \) are the test section angle, tailboard angle relative to the chordwise direction, and throttle angle relative to the tailboard, respectively. The cascade consists from seven double circular airfoils (Blade -3 to 3), whose parameters and reference flow condition are summarized in Table 1.

Figure 1 (c) shows positions of flow measurement for the evaluation of operating condition of the cascade. Two measurement lines denoted ML1 (\( \xi_1 \) axis, subscript 1) and ML2 (\( \xi_2 \) axis, subscript 2) are located by 50% chord upstream and downstream from the leading and trailing edges, respectively. The periodicity of the flow field is also assessed by the pressure difference between two neighbouring channels.

\[
\Delta p/p_t = |p_2(s/2) - p_2(-s/2)|/p_t
\]

\[
\text{TV} = \frac{1}{p_t} \sum_{m=-2}^{2} \left| p_2 \left( \frac{2m+1}{2} \right) - p_2 \left( \frac{2m-1}{2} \right) \right|
\]

The \( \Delta p/p_t \) quantifies the pressure difference between two centre channels. The TV is an indicator named “total variation”, which quantifies global periodicity by the sum of pressure difference between two neighbouring channels.

**Evaluation of steady and unsteady flow fields**

The steady flow field upstream and downstream the cascade is evaluated by Mach number \( M \), flow angle \( \beta \), and static pressure \( p \). In addition to these distributions, the periodicity downstream the cascade is assessed by the following two simple indicators:

The steady aerodynamic force acting on the blade is evaluated by pressure and lift coefficients defined as follows.

\[
C_p = \frac{p - p_1(0)}{\frac{1}{2} \rho U^2}
\]

\[
C_l = -\frac{1}{cl} \int C_p n \cdot dS
\]

The reference static pressure is obtained at 50% chord upstream from the leading edge (i.e., \( \xi_1 = 0 \)). On the other hand, unsteady static pressure and aerodynamic force under the
blade oscillation \( h(t) = \overline{h} e^{i\omega t} \) are evaluated by the Fourier transformation of the \( C_p \) and \( C_l \) as follows.

\[
C_{ph} = \frac{c}{T h} \int_{t=t_0}^{t_0+T} C_p e^{-i\omega t} dt \quad (5)
\]

\[
C_{lh} = \frac{c}{T h} \int_{t=t_0}^{t_0+T} C_l e^{-i\omega t} dt \quad (6)
\]

**NUMERICAL METHOD**

**Flow solver**

The baseline code employs finite-volume discretization of compressible Reynolds-averaged Navier-Stokes equations on the structured mesh which have been developed in our past study [13]. Inviscid and viscous terms are evaluated by the AUSM-type SHUS scheme by Shima [14] with the third order MUSCL interpolation and the second order central difference, respectively. Time integration is conducted by the first order backward Euler scheme for the steady flow analysis, while the three-point backward difference with three Newton iterations is used for the unsteady simulations. As a turbulence model, one-equation Spalart-Allmaras model [15] is employed with fully-turbulent treatment.

**CFD mesh and overset mesh technique**

Our key progress for enabling simulations of whole the wind tunnel is the extensive use of the overset mesh technique. The every component of the wind tunnel is meshed by simple O- or H-mesh with algebraically extruding the surfaces, and they are assembled like a patchwork quilt. The data communication is conducted by the trilinear interpolation.

The connectivity data is generated by the combination of distance-based blanking and iterative adjustment of the fringe cells [16]. Figure 2 shows the process of creating connectivity data. Before searching data source (donor) for interpolation, the cells are blanked based on the distance of the components. Then the fringes of activated cells are marked as receptor, and donors are searched. The cells that do not have donors are considered as “orphan” cells. After finishing the donor search, the blanked cells neighbouring the orphans are activated and donor search is run again. The dataset is completed after all orphan cells are eliminated. In the present computation, the receptor cells cannot be donors, and any orphan cells are not allowed since they bring significant numerical error [17].

Figure 3 shows the mesh assembly for the present study. The mesh system has two cells in the spanwise direction and 0.75 million cells in total. All the inter-blade passages, bypass channels, and small gaps are successfully connected around the midpoint between the parts. The percentage of activated cells against the total cells is 72.4%.

In addition to the wind tunnel mesh, a conventional line-matched multi-block mesh is generated for preparing baseline flow field with perfect pitchwise periodicity. This configuration is referred to as “ideal” case. Figure 4 and Table 2 provide the mesh parameters around the blade.

| Parameters     | Wind tunnel | Multi-block |
|---------------|-------------|-------------|
| Chordwise     | 147 pts     | 178 pts     |
| Pitchwise     | About 100 pts | 120 pts     |
| Wall mesh spacing | 0.9µm   | 0.9µm       |

Fig. 4 Mesh around the blade 0

Then the fringes of activated cells are marked as receptor, and donors are searched. The cells that do not have donors are considered as “orphan” cells. After finishing the donor search, the blanked cells neighbouring the orphans are activated and donor search is run again. The dataset is completed after all orphan cells are eliminated. In the present computation, the receptor cells cannot be donors, and any orphan cells are not allowed since they bring significant numerical error [17].

Figure 3 shows the mesh assembly for the present study. The mesh system has two cells in the spanwise direction and 0.75 million cells in total. All the inter-blade passages, bypass channels, and small gaps are successfully connected around the midpoint between the parts. The percentage of activated cells against the total cells is 72.4%.

In addition to the wind tunnel mesh, a conventional line-matched multi-block mesh is generated for preparing baseline flow field with perfect pitchwise periodicity. This configuration is referred to as “ideal” case. Figure 4 and Table 2 provide the mesh parameters around the blade.
show the appearance and mesh parameters for the O-mesh around the blade. The multi-block configuration have slightly higher mesh resolution.

STEADY FLOW RESULTS AND DISCUSSION

Baseline flow field

At the beginning of the discussion part, the baseline flow field for the discussion hereafter is presented for both wind tunnel and ideal configurations. The back pressure for the both cases are $0.67 \rho_l D_45D/\rho_l D_{st}$. The wind tunnel settings are: $\theta_{ts} = 6$, $\theta_{th} = 2.4$, $\theta_{bh} = 1.2$ degrees, and the suction velocity of $v_s = 6$ m/s.

Figure 4 shows Mach number distributions for the both cases. Figure 4 (a-1 and 2) show the wind tunnel case. The supersonic started inflow is realized for all passages of the cascade and the bypass channel. In the blade passage, the oblique shock from the leading edge and normal passage shock can be observed. The shock pattern including lambda-shaped shock-boundary layer interaction within the passage for the wind tunnel case is quite similar with that in the ideal case shown in Figure 4 (b).

Figure 5 shows $C_p$ distributions on the blade No. -2 to 2 and the ideal case. As expected from the Mach distributions, almost periodic flow field is realized for the wind tunnel case. The positions of the shock are deviated by approximately $5\% c$ within the blade -1 to 2 both on the suction and pressure sides.

Effect of suction on the upstream periodicity

The effect of transonic wall suction is evaluated for making the inflow as uniform as possible. Figure 6 (a) and (b) show comparison of shock pattern upstream the cascade, which is visualized by the horizontal density gradient. The lower bypass is unstarted and significant non-uniformity can be seen in the non-suction case, while appropriate amount of suction gives uniform shock structure as shown in Fig. 4 (b).

Fig. 4 (c) shows detailed comparison of Mach number and flow angle along ML1 against different suction velocities. Both Mach number and flow angle are significantly affected.
by the amount of suction, and the effect of suction mainly appears upstream the blade 2 and 3. In addition, an appropriate amount of suction gives similar distribution as the ideal case.

**Effect of throttling on the downstream periodicity**

The effect of throttling angle on the downstream periodicity is evaluated by fixing all other parameters. Figure 7 shows a comparison of static pressure distributions along ML2 for five different throttling opening angles: $\psi_{th} = 0.4, 0.8, 1.0, 1.2,$ and $1.6$ [deg]. Under the fixed tailboard, the pressure distribution downstream the cascade is highly affected by the throttling angle. For the $\psi_{th} = 0.4$ [deg] case, the pressure downstream the blade No. -3 to -1 is significantly small compared to that downstream the blade No. 1 to -3. The situation is opposite for the $\psi_{th} = 1.6$ [deg] case.

The blade loading is also highly affected by this non-periodic pressure distribution. Figure 8 shows the $C_p$ distributions among five centre blades for the $\psi_{th} = 0.4$ and $\psi_{th} = 1.6$ [deg] cases. There are significant variations in the blade loading and shock position for these cases, compared to the $\psi_{th} = 1.2$ [deg] case shown in Fig. 5.

This result is qualitatively consistent with the findings by Lepicovsky et al [5], and it also implies that there is an optimal throttling angle for realizing the best periodicity for every different tailboard angle.

In order to confirm the existence of optimal throttling angle for the best periodicity, the periodicity in static pressure is quantified by parameters $\Delta p / p_1$ and $TV$ in Eqs. (1-2). The optimum conditions for these indicators are $\Delta p / p_1 = 0$ and the minima of $TV$. Figure 9 shows the change in $\Delta p / p_1$ and $TV$ for different tailboard and throttling angles. Each line is obtained by sweeping $\psi_{th}$ while keeping $\psi_{th}$ constant. The zero-points of the $\Delta p / p_1$ and the local minima of $TV$ can be found on each tailboard angles. Therefore, it can be confirmed that the throttling angle for the best periodicity depends on the tailboard angle.

**Controlling pitchwise periodicity and its importance**

In the choked cascade with fixed supersonic inflow, all the aerodynamic characteristics are dependent variables of the outlet pressure. Here, the effect of periodicity on the measured aerodynamic characteristics is discussed in detail.

Figure 10 shows the relationships between spatial averaged outlet pressure $\overline{p_2} / p_1$ and outflow angle $\beta_2$. The averaging is conducted over two centre passages along ML2 (i.e. $-s \leq \xi_2 \leq s$). In the wind tunnel results, the measured
characteristics under the fixed tailboards depends on the tailboard angle, and none of them is consistent with the cascade under ideal periodicity. On the other hand, although the difference of approximately 0.2 degrees exists, the curve under $\Delta \beta = 0$ shows similar trend with the ideal case.

From this result, it can be concluded that realizing the best pitchwise periodicity is quite important for obtaining appropriate aerodynamic characteristics of ideal cascades.

UNSTEADY FLOW RESULTS AND DISCUSSION

The aerodynamic influence coefficients (AICs) are obtained with oscillating the centre blade [12] in both the wind tunnel and ideal configurations. The cascade is operated on the baseline condition shown in the previous section. The wind tunnel computation directly employs the tunnel geometry, while 15 blades are prepared for the ideal case. In order to investigate wide range of oscillation condition, the blade frequency is varied from 50Hz to 1500Hz. All the computation are conducted with constant blade amplitude of $\bar{h} = 0.5$ mm.

Aerodynamic Influence Coefficients (AICs)

Figure 11 shows the comparison of frequency dependency of the AICs on the three centre blades between wind tunnel and ideal configurations. Since the AICs are expressed by complex number, their amplitude and phase...
angle is shown. The trend of two different configurations are qualitatively similar both in amplitude and phase angle. However, significant difference in amplitude can be seen in the frequency ranging from 400Hz to 800Hz.

**Blockage effect of the tailboards**

In order to find out where the difference in the amplitude of AICs originates, unsteady flow field around the cascade is investigated. Figure 12 shows pressure amplitude around the cascade for both ideal and wind tunnel configurations. Four frequency levels from 250Hz to 700Hz are highlighted in order to see the increasing process of the difference.

In the ideal cascade shown in Fig. 13 (a), pressure amplitude continuously increases at downstream of the blade -2 to -1 as the frequency increases. In other words, the pressure amplitude becomes higher on the downstream of lower blades.

In the wind-tunnel configuration, the upper tailboard exists where the pressure amplitude increases in the ideal configuration. Therefore, there can be some interaction between the wind tunnel wall and acoustic wave propagating toward minus-direction of the blade number. The acoustical reflection on the tunnel wall potentially cause the observed amplitude difference. More detailed interaction phenomenon and the consistency with the theoretical backgrounds could not figure out however, it can be said that the unsteady flow field can be grasped in a qualitative sense by the developed CFD code.

**Acoustic cut-on in the duct**

Although the actual frequency is very high compared to the past flutter experiments (e.g., up to 550Hz and 730Hz for the standard configurations 5 and 7, respectively [1]), the acoustic cut-on in the exit duct appears in the present computation. The lowest cut-on frequency of the acoustic wave propagating downstream inside a rectangular duct can be derived as follows.

\[
f_{cuton} = \frac{a}{2D} \sqrt{1 - M^2}
\]

In the present baseline case, the duct width, sound speed and Mach number are \(D = 87.6\) mm, \(a = 327\) m/s, and \(M = 0.693\), respectively. The resultant cut-on frequency is \(f_{cuton} = 1350\)Hz.

Figure 13 shows pressure amplitude distribution inside the duct. Fig. 13 (a) and (b) correspond to cut-off (1100Hz) and cut-on (1500Hz) case, respectively. The pressure amplitude decays toward downstream for the cut-off case, while the wall pressure become higher entire in the streamwise direction for the cut-on case.

Figure 14 shows the amplitude of wall pressure against blade frequency sampled near the leading edge of the throttle. The pressure amplitude gradually increases as the frequency increases. Then, the amplitude drastically jumps when the blade frequency reaches the cut-on frequency. From this result, it is confirmed that the present method can capture global unsteady phenomena like acoustic cut-on.

**CONCLUSION**

A numerical method and its application for an assessment of the flow field inside a wind tunnel was presented. In order to simulate geometrically complex configurations, the overset mesh technique was implemented to our structured CFD solver. The mesh of the wind tunnel was created by assembling all component mesh using distance-based blanking and iterative donor search. This methodology enabled to simplify the mesh generation and parametric study processes.

Applying the developed method, steady and unsteady simulations of the whole wind tunnel were conducted. The upstream and downstream periodicity of the cascade, effect of the tunnel wall on the unsteady flow field, and duct acoustic characteristics were focused on. The findings are summarized as follows.

1. The periodicity of downstream pressure distribution and blade loading is highly affected by the setting of tailboard and throttle. An optimum throttle position for the best periodicity exists for each tailboard angle, which provides appropriate aerodynamic characteristics of ideal cascades in the wind tunnel environment.

2. The difference in the influence coefficients between ideal and wind tunnel configurations becomes large when the pressure amplitude increases on the downstream of lower blades.
(3) The present method can capture global unsteady phenomena like acoustic cut-on.

The developed CFD solver will be helpful for future design activities or assessments of flow field inside various experimental facilities.

NOMENCLATURE

| Symbol | Description |
|--------|-------------|
| $\beta$ | [deg] flow angle relative to the cascade |
| $\theta_{ts}$ | [deg] setting angle of the test section |
| $\theta_{tb}$ | [deg] setting angle of the tailboards |
| $\theta_{th}$ | [deg] opening angle of the throttle |
| $\theta_s$ | [deg] stagger angle |
| $\xi$ | [m] coordinate on the measurement line |
| $a$ | [m/s] sound speed |
| $c$ | [m] chord length |
| $C_l$ | [-] steady lift coefficient |
| $C_{lh}$ | [-] unsteady lift coefficient |
| $C_p$ | [-] steady pressure coefficient |
| $C_{ph}$ | [-] unsteady pressure coefficient |
| $D$ | [m] duct height downstream the cascade |
| $f$ | [Hz] frequency |
| $h_0$ | [m] oscillation amplitude of the blade 0 |
| $l$ | [m] span length |
| $M$ | [-] Mach number |
| $n$ | [-] direction vector normal to the chord line |
| $p_t$ | [Pa] total pressure |
| $s$ | [m] pitch length |
| $T$ | [sec] time period of blade oscillation |
| $t$ | [sec] time |
| $v_s$ | [m/s] suction velocity on the transonic wall |
| $x$ | [m] chordwise distance from the leading edge |

Subscripts and accents

| Subscript | Description |
|-----------|-------------|
| 1         | quantities on ML1 |
| 2         | quantities on ML2 |
| $^*$       | pitchwise average |

REFERENCES

[1] Fransson, T. H. and Verdon, J. M., “Updated Report on "Standard Configurations for Unsteady Flow Through Vibrating Axial-Flow Turbomachine-Cascades"”, http://www.energy.kth.se/proj/projects/Markus.Joecker/STCF/, 1991

[2] Ren, W., Seeley, C. E., Zhang, X., Mitchell, B. E., and Ju, H., “Investigations of Flutter and Aero Damping of A Turbine Blade Part 2: Numerical Simulations”, ASME Paper GT2016-57935, 2016

[3] Vogt, D. and Fransson, H. T., “Experimental Investigation of Mode Shape Sensitivity of an Oscillating Low-Pressure Turbine Cascade at Design and Off-Design Conditions”, J. Eng. Gas Turbines Power 129(2), 2006, pp 530-541

[4] Vogt, D., “Experimental Investigation of Three-Dimensional Mechanisms in Low-Pressure Turbine Flutter”, Royal Institute of Technology Ph.D Thesis, 2005, pp 80-92

[5] Lepicovsky, J., McFarland, E. R., Chima, R.V., and Wood, J.R., “On Flowfield Periodicity in the NASA Transonic Flutter Cascade”, J. Turbomach 123(3), 2000, pp 501-509

[6] Buffum, D. H. and Fleeter, S., “Wind Tunnel Wall Effects in a Linear Oscillating Cascade”, J. Turbomach 115(1), 1993, pp 147-156

[7] Buffum, D. H. and Fleeter, S., “Effect of Wind Tunnel Acoustic Modes on Linear Oscillating Cascade Aerodynamics”, J. Turbomach 116(3), 1994, pp 513-524

[8] Ott, P., Norryd, M., and Böls, A., “The Influence of Tailboards on Unsteady Measurements in a Linear Cascade”, ASME Paper 98-GT-572, 1998

[9] Aotsuka, M., Watanabe, T., and Machida, Y., “Role of Shock and Boundary Layer Separation on Unsteady Aerodynamic Characteristics of Oscillating Transonic Cascade”, ASME Paper GT2003-38425, 2003

[10] Watanabe, T. and Aotsuka, M., “Unsteady Aerodynamic Characteristics of Oscillating Cascade With Separation Bubble in High Subsonic Flow”, ASME Paper GT2005-68665, 2005

[11] Kazawa, J. and Watanabe, T., “Active Suppression of Cascade Flutter with Piezoelectric Device”, ASME Paper GT2006-90645, 2006

[12] Hanamura, Y., Tanaka, H., and Yamaguchi, K., “A Simplified Method to Measure Unsteady Forces Acting on the Vibrating Blades in Cascade”, Bulletin of JSME, Vol.23, No.180, 1980, pp 880-887

[13] Tateishi, A., Watanabe, T., Himeno, T., Aotsuka, M., and Murooka, T., “Verification and Application of FSI and Modal Identification Technique to Cascade Flutter Simulations”, Proceedings of International Gas Turbine Congress 2015, WePM1F.3, 2015

[14] Shima, E., “Role of CFD in Aeronautical Engineering (No.14) -AUSM Type Upwind Schemes-”, NAL-SP30, Proceedings of 13th NAL Symposium on Aircraft Computational Aerodynamics, 1996, pp 41-46

[15] Spalart, P. R. and Allmaras, S. R. “A One-Equation Turbulence Model for Aerodynamic Flows”, Recherche Aerospatiale 1, 1994, pp 5-21

[16] Chan, W. M., Kim, N., and Pandya, S. A., “Advances in Domain Connectivity for Overset Grids Using the X-rays Approach”, Proceedings of the Seventh International Conference on Computational Fluid Dynamics, ICCFD7-1201, 2012

[17] Quon, E. W. and Smith, M. J., “Advanced Data Transfer Strategies for Overset Computational Methods”, Computers & Fluids 117, 2015, pp 88-102