Wright State University

CORE Scholar

Mechanical and Materials Engineering Faculty
Publications

Mechanical and Materials Engineering

2004

CFD Analysis of Unsteady Separated Transonic Oscillation Cascade Aerodynamics

Xuedong Zhou

J. Mitch Wolff

Wright State University - Main Campus, mitch.wolff@wright.edu

Follow this and additional works at: <https://corescholar.libraries.wright.edu/mme>

Part of the [Materials Science and Engineering Commons](#), and the [Mechanical Engineering Commons](#)

Repository Citation

Zhou, X., & Wolff, J. M. (2004). CFD Analysis of Unsteady Separated Transonic Oscillation Cascade Aerodynamics. *International Journal of Turbo and Jet-Engines*, 21 (3), 143-153.
<https://corescholar.libraries.wright.edu/mme/5>

This Article is brought to you for free and open access by the Mechanical and Materials Engineering at CORE Scholar. It has been accepted for inclusion in Mechanical and Materials Engineering Faculty Publications by an authorized administrator of CORE Scholar. For more information, please contact library-corescholar@wright.edu.

CFD Analysis Of Unsteady Separated Transonic Oscillation Cascade Aerodynamics

Xuedong Zhou and J. Mitch Wolff

Department of Mechanical & Materials Engineering, Wright State University, Dayton, OH, 45324, USA

Abstract

With an improved algebraic mesh-deforming algorithm, STAR-CD, a commercial computational fluid dynamics (CFD) solver is employed for the numerical analysis of a transonic oscillating linear cascade of advanced design blades. The center blade oscillates 0.6-degrees about the middle cord. The numerical simulation is conducted for a frequency range from 200 Hz to 500 Hz. A hybrid grid, which utilizes a structured O-grid around the airfoil and an unstructured grid everywhere else is employed. The Spalart-Allmaras (S-A), one equation turbulence model, along with other two equation $k-\epsilon$ models, are also utilized for the steady state simulation. The S-A turbulence model provided significantly better steady state results in the separated flow region than the $k-\epsilon$ model. The agreement between the experimental data and CFD prediction was better for the $M=0.8$ unsteady results than the $M=0.5$ on the suction surface. This could be attributed to a larger unsteady variation in the reattachment location for the $M=0.5$ results than the $M=0.8$. Finally, the time average of the unsteady skin friction coefficient is significantly different than the steady state value indicating nonlinear unsteady aerodynamics are significant for this analysis.

Nomenclature

C	Blade chord
Cp	First harmonic pressure coefficient $P/(\rho U^2 \alpha)$
	Steady pressure coefficient $(P - P_{\infty}) / (0.5 \rho U^2)$
k	Semi-chord reduced frequency $\omega C / 2V$
M	Mach number
n	Airfoil numbering, $n=0, \pm 1, \dots, \pm 4$
P	Airfoil surface pressure
P_{in}	Inlet pressure
S	Airfoil spacing
V	Velocity
x	Axial location
α_1	Amplitude of pitching motion, 0.6°
γ	Stagger angle
ρ	Air density
μ	Viscosity
β	Interblade phase angle
ω	Oscillation frequency

Introduction

Turbomachinery flutter/forced response is a physical phenomena strongly related to the unsteady aerodynamics of the engine which can result in a high cycle fatigue structure failure. To accurately simulate these physical phenomena, a fully coupled structural and fluid analysis is required to predict the blade behavior. As an initial step toward this goal, research has been accomplished utilizing a prescribed motion of the blades. Experimentally, this has been accomplished primarily in linear cascade facilities and is extremely challenging and costly. Computationally, the cost is significantly lower but the question of the fidelity of the analysis is of serious concern. Traditional CFD analysis assumes the surface to be stationary. To analyze a blade row with a prescribed motion requires the movement of the computational boundaries. Hence, for turbomachinery simulations with a moving boundary, CFD tools, besides being fast and robust, need to be able to handle the following issues.

1. **Mesh deforming algorithm:** Since the boundary of blade surface is moving, then the computational mesh needs to move in a time-

marching manner. Algorithms, such as an algebra method, a spring-mass system method and an elastic system have been typically employed.

2. **Geometric Conservation Law (GCL):** Typically CFD models solve the governing equations in the computational domain and apply boundary conditions in physical domain. Since most codes are dedicated to a fixed surface analysis, a simplification is employed to only address a fixed boundary surface analysis. Therefore, an artificial source term will be generated for models with a moving computational mesh.
3. **Non-Reflection Boundary Conditions:** For turbomachinery, the unsteady flow field will generate various propagating waves. These waves are classified as either sub-resonant or super-resonant depending on various parameters. Sub-resonant waves decay exponentially with distance from the unsteady source while super-resonant waves propagate unattenuated from the source. These propagating waves can cause serious errors if proper far-field boundary conditions (i.e. non-reflecting) are not employed.

Once these issues have been effectively addressed, then the CFD model can be modified to include a structural model to determine the blade surface motion (i.e. fluid/structure interaction). Therefore, it is vital that the above issues be properly validated before developing a fluid/structure interaction model.

In this paper, the issues related to a moving grid CFD model for turbomachinery applications are addressed. Two-dimensional linear theory can be applied to an oscillating cascade of flat plate airfoils to obtain an analytic solution, which can be compared to a high fidelity CFD model. A three-dimensional helical fan rotor of twisted flat plates can also be utilized to validate the 3-D CFD model. After validation of the deforming grid algorithm, the unsteady CFD model is utilized to model a transonic cascade of advanced fan tip airfoils, which have significant flow separation.

Moving Boundary and Mesh

There are several different methods for updating the computational mesh for moving surfaces, such as a mass-spring model /1/, an elastic structure FEA model /2/ and an algebra model /3/. These methods have drawbacks, either high computational expense or insufficient robustness. Melville /4/ proposed that each mesh point obtain a reference surface point by searching for a reference triangle plane on the wetted surface, and move accordingly to the new position of the reference point. The basic procedure, shown in Figure 1, is as follows:

1. Globally search for the closest point, N_1 , on moving surface for each mesh point, R , in the flow domain.
2. Identifying the first edge of the reference triangle, $\overline{N_1 N_2}$, which is most aligned with the projection of the $\overline{N_1 P}$.
3. Locating the second reference triangle edge, $\overline{N_1 N_3}$, which is most orthogonal to $\overline{N_1 N_2}$.
4. Find reference point, R , by projecting P on the reference triangle plane.
5. Move P according to the new position of reference point R .

However, the method has problems finding $\overline{N_1 N_2}$ and $\overline{N_1 N_3}$ if the mesh is skewed or has large aspect ratios. An alternative method is proposed.

Instead of using the 2nd and 3rd steps above to find the two edges, one can simply find those two edges by picking two nodes adjacent to N_1 in the same cell surface. Therefore, the reference point can

Fig. 1: Target Point Projection

be located with respect to the new reference triangle. With this modified method, the grid motion is more robust and computationally efficient.

Linear Theory Validation

Validation of STAR-CD for a moving boundary cascade simulation is obtained by comparing numerical results with linear theory [5] for both a 2-D and a 3-D oscillating flat plate configuration.

2-D Linear Flat Plate

A structured H-mesh is constructed for up to two flat plate blade passages. The flat plates are simulated with a baffle cell, which has zero thickness with wall boundary conditions on both sides of the cell. For all the results presented the inlet Mach number is 0.7, the stagger angle is 45 degrees and the solidity is $\sqrt{2}$, Figure 2. The flat plate blades are moved in a prescribed plunging motion of 0.5% chord to ensure the linear theory assumption is valid. The inflow and outflow boundary plane is located 3 chords up and down stream of the flat plates to reduce any unsteady wave reflections.

Fig. 2: Schematic of the flat plate model

To determine the effects of any unsteady waves generated by the oscillating airfoils, two configurations are calculated 1.) a sub-resonant and 2.) a super-resonant. A sub-resonant case, where propagating waves decay exponentially, is obtained for the previously described cascade of flat plates when the interblade phase angle is 180° with a semi-chord reduced frequency, k , of 0.33. Figure 3 shows the magnitude and phase of the first harmonic of the surface unsteady pressures compared to the linear theory results. Excellent agreement is shown in terms of the unsteady magnitude and phase. A super-resonant case, where propagating waves do not decay, is obtained when the interblade phase angle is 0° with a semi-chord reduced frequency, k , of 0.75. Figure 4 contains both the STAR-CD and linear theory results. Excellent agreement is shown to linear theory, therefore the validation of the 2-D implementation of the deforming grid algorithm into STAR-CD has been shown.

Fig. 3: First harmonic ΔC_p 2-D flat plate in torsion with $M=0.7$ $\beta=180^\circ$ $k=0.33$

Fig. 4: First harmonic ΔC_p 2-D flat plate in torsion with $M=0.7$ $\beta=0^\circ$ $k=0.75$

3-D Helical Fan of Flat Plate Blades

The helical fan, Figure 5, consists of flat plate blades with zero thickness enclosed within a rigid cylindrical duct with no gap between the blade tip and duct. The blades are twisted to maintain a zero incidence at all sections. The inflow relative Mach number at mid-span is 0.7 with the free stream axial Mach number being 0.495. The stagger angle at mid-span of the helical fan is 45° , and the gap to chord ratio is unity for a rotor with 24 blades and a diameter of 8.448, with a hub to tip ratio of 0.8.

The helical fan configuration is a closer model to an actual turbomachinery configuration since it is a three-dimensional problem; therefore it can be used to further verify the ability of the STAR-CD model to simulate vibrating blade row unsteady aerodynamics. The computational mesh was obtained from Bakhle /6/ It is a structured sheared

Fig. 5: Helical fan geometry

H-grid mesh with 141 points in the stream wise direction, 11 grid points in the span wise direction and 41 points in the blade-to-blade direction. Both inlet and exit planes are 0.7 chord length, respectively.

Results from a pitching motion of 0.1° about the mid-chord are presented here. Again, both a sub-resonant and a super-resonant configuration was analyzed. A time step dependence study was conducted and 200 time steps per blade pass obtained time step independence. For both cases, starting from a steady state solution, a periodic solution was obtained in 4 blade passages. For the sub-resonant case, an interblade phase angle of 180° and a reduced frequency of 0.5 were analyzed. Figure 6 shows the comparison between STAR-CD and linear theory. The agreement is excellent for this sub-resonant case. Figure 7 contains the results for the super-resonant case of 0° interblade phase angle and $k=0.75$. There is a slight over-prediction of the magnitude and the phase. This could be the result of reflections from the inlet and exit boundaries since they are at only 0.7 chords and STAR-CD does not have non-reflecting boundary conditions. We have been told that the next release of STAR-CD will have a non-reflecting boundary condition option. These results validate that the improved mesh deforming method in STAR-CD is capable of

handling the GCL and unsteady wave propagation issues.

Fig. 6: First harmonic ΔC_p 3-D flat plate in torsion with $M=0.7$ $\beta=180^\circ$ $k=0.5$

NASA Oscillating Transonic Cascade

Facility Description:

Details of the NASA transonic oscillation cascade facility [7,8] at Glenn Research Center are

Fig. 7: First harmonic ΔC_p 3-D flat plate in torsion with $M=0.7$ $\beta=0^\circ$ $k=0.5$

given in Figures 8 and 9 which show both the geometry of the facility and blade numbering convention. Table 1 contains the detailed design information of the advanced fan blade airfoil test article. The cascade is a 9 blade linear cascade with the capability of moving the blades in a prescribed pitching motion. For the results presented in this paper, only the middle blade is oscillated about the

Fig. 8: Cascade Configuration C, 20/24

Fig. 9: Airfoil and cascade coordinate system

Table 1
Airfoil and cascade parameters

Blade chord	C	89.2mm
Leading edge camber angle	ν	29.5 degree
Maximum thickness	t_{max}	0.045 C
Location of maximum thickness	ϕ_{max}	0.625 C
Stagger angle	γ	60.0 degree
Number of blades in the cascade		9
Blade pitch	S	58.4 mm
Cascade solidity	C/S	1.53
Pitching axis	ϕ	0.5 C
Blade height	h	95.9 mm

mid-cord. Lepicovsky /9/ conducted the experiments at a pitching amplitude of 0.6 degree, frequencies of 200, 300, 400 and 500Hz and inlet Mach numbers of 0.5 and 0.8. For the 0.8 Mach case, the flow is

transonic since the leading edge region goes supersonic. The airfoil has negative camber at the leading edge, which causes a large separation region on the suction side starting at about 5% chord and extended up to 40%. Currently, experimental data are available on the suction surface of blade 6 and the pressure surface of blade 4 for the configuration with blade 5 oscillating.

Previously, Buffum /7/ conducted experiments with all nine blades oscillating at a particular inter-blade phase angle. There are several numerical comparisons for this configuration, which have shown good agreement /10/. However, most of the comparisons are for incompressible flow ($M=0.2$) with minimal separation.

Computational Grid

A hybrid computational grid, with a structured grid around the airfoil surface and an unstructured grid everywhere else, is employed. The benefit of the structured grid is to accurately capture the flow characteristics near airfoil surface by providing better control of y^+ values for turbulence modeling. The unstructured grid eases the overall grid generation effort by removing problems of high grid skewness for high stagger angle and low solidity cascade geometries like the NASA advanced fan airfoils. Figure 10 shows the computational grid for

Fig. 10: Hybrid Computational Grid

one flow passage with exploded views of the leading and trailing edge regions. The nominal y^+ value for the computation grid was 1.0.

Turbulence Models

Due to the negative camber in the leading edge of the airfoil, a large separation region starting at 5% chord and extending to about 40% chord is measured. Several researchers [10,11] have done similar calculations and shown that the Spalart-Allmaras one equation turbulence model is the best model to capture separation for an airfoil application. However, most of the work has been done for incompressible flow such as 0.29 Mach. Therefore, calculations will be performed with different turbulence models to determine which model is best for turbomachinery unsteady aerodynamics modeling with strong separation regions. The available turbulence models in STAR-CD are a linear, quadratic and cubic $k-\epsilon$ model and the one-equation Spalart-Allmaras model.

Steady State Results

Initially, 2-D steady state calculations were obtained of the NASA Transonic Cascade configuration for all nine stationary blades for $M=0.5$ and $M=0.8$. Figures 11 and 12 show the steady C_p distribution on blade #4 for both Mach numbers, respectively. For the subsonic case ($M=0.5$) only the cubic $k-\epsilon$ model results are compared to the experimental data because both the linear and quadratic results are basically identical to

Fig. 11: Steady C_p for 0.5 Mach Number

Fig. 12: Steady C_p for 0.8 Mach Numbers

the cubic results. The Spalart-Allmaras turbulence model provides a significantly better prediction than the $k-\epsilon$ model in the strong separation region on the suction surface. The differences between the two turbulence models and the experimental data outside the separated flow region is minimal, therefore the overall agreement of the S-A results is excellent. For the transonic results (Figure 12), the S-A model still does a significantly better job than the $k-\epsilon$ model, but it does not compare as well to the experimental data as for the subsonic case. The predicted reattachment point was significantly off which could be because of the shock wave formed near the leading edge. Based on the steady state results only the S-A model will be utilized for the unsteady analysis.

Inlet/Exit Boundary Plane Study

Since STAR-CD currently does not have non-reflective boundary conditions, the effect of the location of the inlet and exit plane boundaries was completed for both the steady and unsteady calculations studied. As expected, for the steady state results no effect was seen with changes in the boundary locations. However, the length of extension does play an important role in the unsteady simulation. Figure 13 presents the comparison of the time history of the blade #4 pressure coefficient at 30% chord for 0.8 Mach and 500 Hz pitching frequency. It can be clearly seen that increasing the inlet/exit plan from 4.5 to 6.0

Fig. 13: Inlet/Outlet boundary plane location influence on Cp magnitude

chords has practically no effect on the magnitude of the unsteady solution, but at 1.8 chords the solutions is significantly lower. Therefore, all unsteady calculations were completed with the inlet/exit boundaries at six chords.

Unsteady Results

The NASA Transonic Oscillating Cascade facility was used with the center (blade #5) moving in a prescribed pitching motion of 0.6° about the mid-chord. The 2-D STAR-CD analysis modeled this configuration including all 9 blades in the cascade with only the 5th blade oscillating 0.6° about the mid-chord. Currently, experimental data is only available on the two stationary blades adjacent to the center oscillating blade. Figure 14 is the pressure

Fig. 14: Time history for experiment data and predicted solution

time trace on the pressure surface (16.6% chord) of blade #4 for 0.8 inlet Mach number and 500Hz oscillation frequency. There is a DC shift in the predicted results compared to the experimental data that represents the difference in the steady state results. Removing this off-set results in a very good agreement between the STAR-CD prediction and the experimental data.

Since the experimental data (see Figure 14) is nearly sinusoidal, the data is processed with an FFT (fast Fourier transform) to obtain the magnitude and phase of the first harmonic. The computational data was analyzed in the same manner and compared to the experimental results at all chord locations.

Figure 15 shows the amplitude and phase comparison with chord for an inlet Mach number of 0.8 and 300Hz oscillation frequency on the suction surface of blade #6. The comparison is excellent for both magnitude and phase of the 1st harmonic pressure coefficient. The largest differences are in the phase data near the leading and trailing edges of the airfoil. But the magnitude is quite small at these locations; therefore, any small error in the magnitude components (i.e. the real and imaginary components) can result in an exaggerated phase error because the phase is the ratio of the real and imaginary components. For an inlet Mach of 0.5 with a 400Hz frequency, the 1st harmonic response of blade #6's suction surface distribution is given in Figure 16. Again, there is overall an excellent agreement between the STAR-CD and experimental data. The phase comparison is almost exact except for the leading edge location, which again has a small amplitude. The magnitude is under-predicted in the 30-40% chord region of the suction surface. This is the region where the experimental data shows reattachment taking place. Therefore, an investigation of the STAR-CD reattachment prediction was investigated.

To better investigate the separated flow region of the suction surface, the skin friction coefficient, C_f , was calculated and plotted versus chord. To capture the unsteady nature of the analysis the maximum and minimum C_f values in a period of oscillation were record along with the time average of the unsteady solution. These three values plus the steady

Fig. 15: 1st Harmonic ΔC_p comparison at $M=0.8$, $f=300\text{Hz}$ for the Suction Surface of Blade #6

Fig. 16: 1st Harmonic ΔC_p comparison at $M=0.5$, $f=400\text{Hz}$ for the Suction Surface of Blade #6

state C_l values were plotted versus the suction surface chord.

Figure 17 contains both the 0.5 and 0.8 Mach number results for blade #5. The time average of the unsteady results does not equal the steady state results for both Mach numbers. Therefore, the unsteady aerodynamics is introducing some nonlinear flow phenomena, which is critical for properly modeling this problem. The unsteady

solution's reattachment point is always earlier than the steady state reattachment, which is in the direction of providing a closer agreement with the experimental data. The amount of variation in the reattachment point is greater in the $M=0.5$ results than in the $M=0.8$ results, which could explain the larger difference in the $M=0.5$ 1st harmonic magnitude results shown previously.

Fig. 17: Skin friction coefficient for suction surface of oscillating blade 5#

Summary and Conclusions

An improved algebraic model for deforming the computational grid has been incorporated into STAR-CD. The enhanced CFD model was then validated for turbomachinery applications by comparison to linear theory for both 2-D and 3-D configurations. The model was then applied to the NASA Transonic Oscillating Cascade advanced design fan blades, which provides a very challenging configuration because of significant separation on the suction surface. The enhanced STAR-CD model overall provided excellent agreement with the NASA experimental data both for steady and unsteady results. The S-A turbulence model provided significantly better results in the separated flow region than the $k - \epsilon$ model. The agreement between the experimental data and CFD prediction was better for the $M=0.8$ unsteady results than the $M=0.5$ on the suction surface. This could be

attributed to a larger variation in the reattachment location for the $M=0.5$ results than the $M=0.8$. Finally, the time average of the unsteady skin friction coefficient is significantly different from the steady state value indicating nonlinear unsteady aerodynamics are significant for this analysis.

Acknowledgements

The authors want to acknowledge Dr. Jan Lepicovsky of the QSS Group Inc. & NASA Glenn Research Center for providing the experimental data. In addition, the support of ADAPCO, Inc. developers of the STAR-CD CFD model, is greatly appreciated. Finally, the financial support of the Dayton Area Graduate Studies Institute (DAGSI) project PR-AFIT-99-07 is most gratefully acknowledged.

References

1. Batina, J.T., "Unsteady Euler Algorithm With Unstructured Dynamic Mesh for Complex - Aircraft Aeroelastic Analysis", AIAA Paper, No 89-1189, April, 1989.
2. Chiandussi, G., Bageda, G. and Onate, E. "A Simple Method for Automatic Update of Finite Element Meshes", *Commun. Numer. Meth. Engng.* 2000, 16.
3. Tsai, H.M., Wong, A.S.F., Cai J., Zhu Y., Liu F. "Unsteady flow calculations with a parallel multiblock moving mesh algorithm", *AIAA Journal*, 39:(6) pp. 1021-1029, Jun 2001.
4. Melville, R. "Dynamic Aeroelastic Simulation of Complex Configurations Using Overset Grid Systems", AIAA Paper, 2000-2341, June 2000.
5. Whitehead, D.S., "Classical Two-Dimensional Methods," AGARD Manual on Aeroelasticity in *Axial-Flow Turbomachines Volume 1 Unsteady Turbomachinery Aerodynamics*, Platzer, M.F. and Carta, F.O., eds, AGARDograph No. 298.
6. Bakhle M., Srivastava R., Keith T., "Aeroelastic Calculations Based on Three dimensional Euler Analysis", AIAA Paper 98-3295, July 13-15, 1998.
7. Buffum D., Capece V., King A., "Experimental Investigation of Unsteady Flows at Large Incidence Angles in a Linear Oscillating Cascade", AIAA Paper 96-2823.
8. Lepicovsky J., McFarland E., Capece V. and Hayden J., "Unsteady Pressures in a Transonic Fan Cascade Due to a Single Oscillating Airfoil", GT-2002-30312, 2002-12-17.
9. Lepicovsky J., McFarland E., Capece V. "Methodology of Blade Unsteady Pressure Measurement In The NASA Transonic Flutter Cascade", NASA Report, 2002.
10. Ekaterinaris, J., Srinivasan, G. and McCroskey, W.J., "Present Capabilities of Predicting Two-Dimensional Dynamic Stall", ARARD CP-552, 1994.
11. Clark, W., and Hall, K., "A Time-Linearized Navier-Stokes Analysis of Stall Flutter", *Journal of Turbomachinery*, July 2002, Vol. 122, pp. 467-476.