Journal of the Korean Society of Marine Engineering, Vol. 35, No. 7, pp. 938～945, 2011(ISSN 1226-9549)
http://dx.doi.org/10.5916/jkosme.2011.35.7.938

On the Significance of Turbulence Models and Unsteady Effect on the Flow Prediction through A High Pressure Turbine Cascade

M. M. El-Gendi[1] · Sang-Wook Lee[2] · Chang-Ho Son†

(Received November 15, 2011; Revised November 22, 2011; Accepted November 23, 2011)

Abstract : Unsteady flow simulations through a transonic turbine vane were carried out for an isentropic Mach number of 1.02 and a Reynolds number of 10^6. The main objective of the study is to investigate the effect of unsteadiness due to vortex shedding on the flow in transonic regime. The steady and the time-averaged unsteady results by employing three different turbulence models: shear stress transport (SST), *k-ω*, and *ω* Reynolds stress models were compared. The comparisons were emphasized on the isentropic Mach number along the blade and total pressure loss at the cascade exit. The results showed that both steady and unsteady calculations have good agreement with experimental data along the blade surface. However, at cascade exit, the unsteady calculations have much better agreement with experimental data than steady calculations. Based on these, we conclude that the unsteady flow calculations are essential for these types of problems.

Key words : Turbine cascade, Unsteady flow, Turbulence model, Transonic flow

1. Introduction

Flow patterns through a high loaded subsonic and transonic turbine cascades are very complicated. These flows are usually combined by a transition and separation along the blade surface, vortex shedding from the trailing edge, and shock waves. Due to these phenomena, the flow is completely unsteady. Therefore, for optimized blade design and better understanding of the flow physics, it is important to investigate these flow patterns and obtain an accurate numerical predication. Arts et al. [1] studied experimentally a high loaded turbine cascade at different flow conditions. They studied subsonic and transonic flows at different Mach and Reynolds numbers with different turbulent intensities. They also studied both aerodynamic and aero-thermal behavior of the flow. Liu [2] checked the validity of a new numerical method (modified implicit flux-vector-splitting) through this flow pattern from aerodynamic point of view. He found that the method is able to capture most of physics of this flow pattern and that its convergence is very rapid. Gehrer and Jericha [3] studied the effect of several turbulent models (Baldwin-Lomax model and several versions of low Re *k-ε* model) on the prediction of the heat transfer inside the turbine cascade. They noticed that the heat transfer prediction near the leading edge is poor. Larsson

† Corresponding Author(School of Mechanical Engineering, University of Ulsan, Ulsan, South Korea, E-mail : chson@ulsan.ac.kr, Tel:052-259-2588
1 School of Mechanical Engineering, University of Ulsan, Ulsan, South Korea / Mechanical Power and Energy Department, Faculty of Engineering, Minia University, Minia, Egypt
2 School of Mechanical Engineering, University of Ulsan, Ulsan, South Korea

This paper is extended and updated from the short version that apeared in the Proceedings of the International symposium on Marine Engineering and Technology (ISMT 2011), held at BEXCO, Busan, Korea on October 25-28, 2011.

[4] made a comparison between the results of k-ε and k-ω turbulence models. He noticed that both turbulent models have a problem in the leading edge region. In addition, it was noticed that the incoming turbulence level has no effect on the transition.

Although this problem is inherently unsteady due to the vortex shedding, we notice that all numerical calculations in the literature considered only steady state solutions.

In the present study, we investigated the effect of unsteady calculations on predictions of isentropic Mach number and total pressure loss. The calculations were carried out using three different turbulence models to predict the flow around the blade and in the wake from aerodynamic point of view. The comparison includes the results of our steady and unsteady calculations and steady turbulent calculation of Liu [2].

2. Investigated cascade

The investigated cascade is a transonic high pressure turbine guide vane. The blade coordinates can be found in Arts et al. [1] and the cascade dimensions are shown in Table 1.

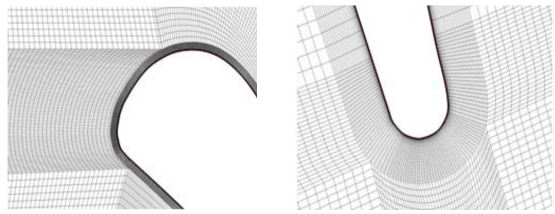
Table 1: Dimensions of cascade and boundary conditions

Name	Symbol	Dimension	Unit
chord	C	67.647	mm
axial chord	C_{ax}	36.462	mm
pitch	g	57.500	mm
stager angle	λ	55.000	deg
inlet total pressure	P_{o1}	159.600	kPa
inlet total temperature	T_{o1}	415.000	K
exit Mach number	$M_{2,is}$	1.020	

3. Numerical Method

3.1 Computational Grid

Figure 1 shows the grid topology around the leading and trailing edges. The O-H type grid was used in all cases. The O type grid around the blade provides good resolution and orthogonality. The H type grid applied in the remaining of the domain to provide good resolution in the wake region and to decrease the skewness. The total number of grid nodes is about 1.24×10^5.



(a) (b)

Figure 1: Computational grids (a) leading edge (b) trailing edge

3.2 Boundary Conditions

All studied cases have the same exit isentropic Mach number ($M_{2,is} = 1.02$) and exit isentropic Reynolds number ($Re_{2,is} = 1.0 \times 10^6$), based on chord length. In this study, both total pressure and total temperature are imposed at the inlet and static pressure is imposed at the exit, (see Table 1). The no slip condition was implemented on the surface of the blade, where the wall is adiabatic. The periodic boundary condition was used in the pitch direction.

3.3. Numerical Scheme

The Navier-Stokes equations were solved by using a commercial code, ANSYS-CFX(ANSYS, Inc, Canonsburg, PA). The cell centered and mesh-vertex finite volume method was used for spatial discretization. The transient scheme was the second order backward Euler method and a high resolution scheme [5] was used for treating the advection term.

3.4. Turbulent Models

Three turbulent models were tested in this study: shear stress transport model (SST) [6], k-ω model [7], and *ω* Reynolds stress model (*ω*RS). Both SST and *k-ω* turbulence models are based on eddy viscosity hypothesis. On the other hand, *ω*RS turbulent model is based on transport equations for all components of Reynolds stress tensor and the dissipation rate. SST turbulent model combines the advantage of *k-ω* turbulent model near the surface and *k-ε* model in the outer regions. In addition, SST turbulent model accounts for the transport of turbulent shear stress.

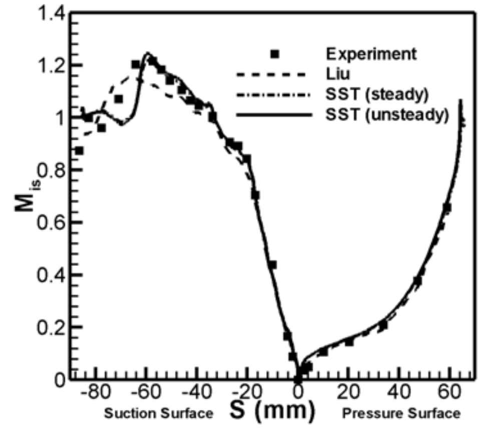
4. Results and Discussion

4.1 Pressure Distribution along the Blade

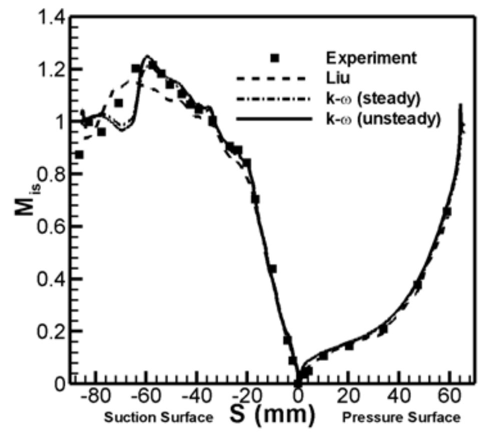
Figure 2 shows the distributions of the isentropic Mach number along the blade in all studied cases and numerical result of Liu [2] as well as the experimental data [1]. The abscissa S represents the coordinate along the blade surface. The absolute value of S means the distance from the stagnation point on the leading edge, and the sign of S means the side of the blade. A positive value means along the pressure surface, and a negative value means along the suction surface. The isentropic Mach number is calculated by the following equation:

$$M_{is} = \sqrt{\frac{2}{\gamma - 1}\left[\left(\frac{P}{P_{o1}}\right)^{\frac{1-\gamma}{\gamma}} - 1\right]}$$

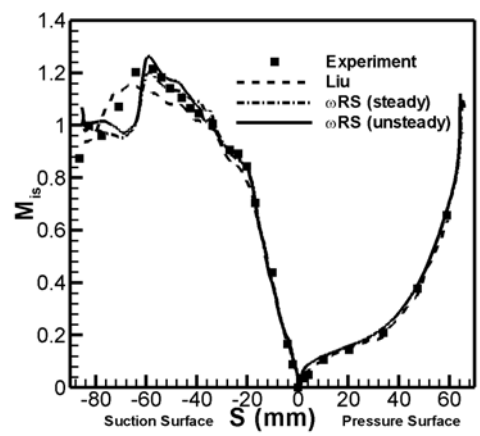
where, P_{o1} is a total pressure at the inlet, P is pressure along the blade surface, and γ is specific heat ratio. All numerical results show good agreement with that of the experimental data [1] along the pressure surface. On the other hand, along the suction surface, there is a deviation on the rear part. The deviation begins from S - 40 mm for Liu's steady calculations [2] and from S - 60 mm for the present steady and unsteady calculations.



(a)

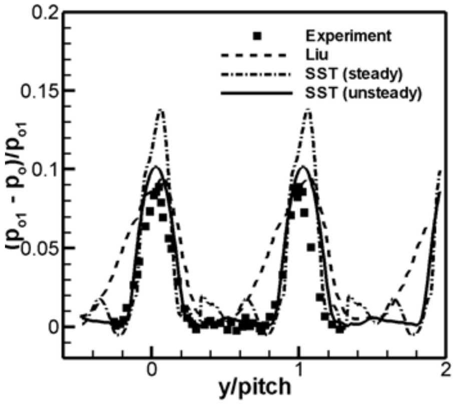


(b)

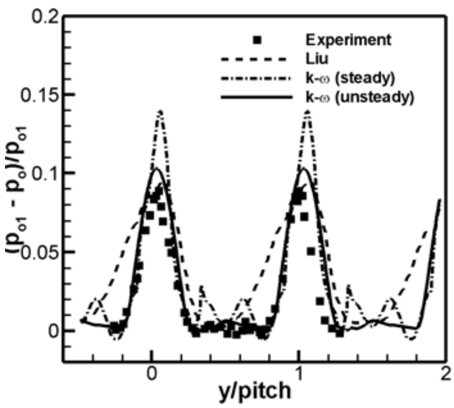


(c)

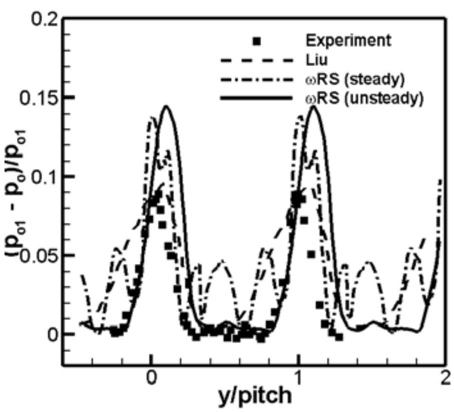
Figure 2: Isentropic Mach number distribution along the blade (a) SST (b) *k-ω* (c) *ω*RS model



(a)



(b)



(c)

Figure 3: Total pressure loss for steady and unsteady calculations (a) SST (b) *k-ω* (c) *ω*RS model

For all turbulent calculations, the difference between the steady and unsteady calculations is almost negligible.

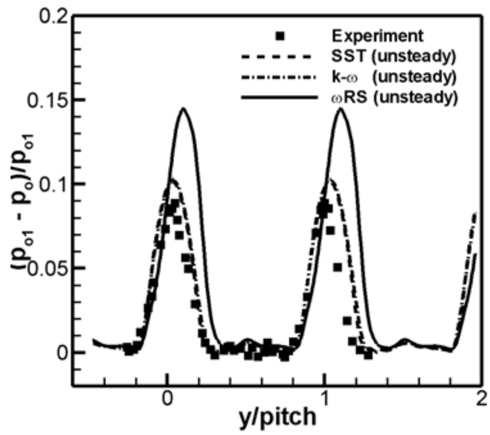


Figure 4: Comparison between the turbulence models

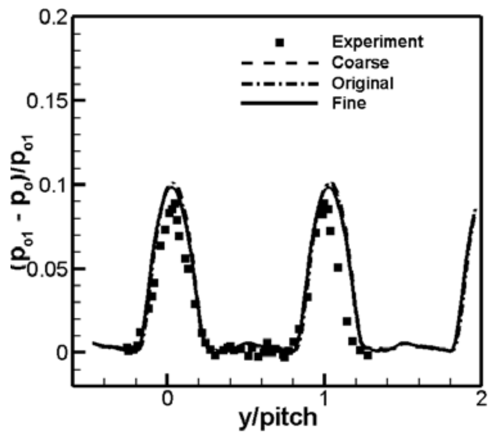


Figure 5: Comparison of time-averaged total pressure loss based on different number of grid

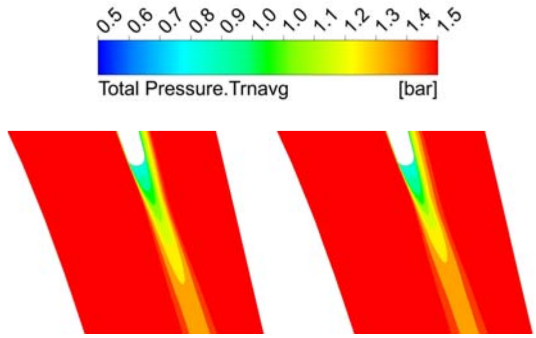
4.2 Total Pressure

The distribution of dimensionless total pressure loss at the exit (x/C_{ax} = 1.4) is shown in Fig. 3 as well as the numerical result of Liu [2] and the experimental data [1]. The *x* coordinate originates from the stagnation point on the leading edge. For all turbulent model calculations, the difference between the steady and unsteady calculations is obvious. In particular, for SST and *k-ω* model, unsteady calculations are much better than steady calculations. On the other hand, for *ω*RS model, the results for unsteady calculation have a negligible improvement over that of the steady calculation. Comparing steady calculations, results of Liu [2] have the best agreement with experimental data for the peak value. For the remainder of the results, the present steady calculation shows much better agreement with experimental data than Liu's steady calculation [2], where Liu's results show excessively broad.

To compare the performances of the turbulence models in unsteady calculations, we drew a single figure combining the results of the three turbulence models. Figure 4 shows the distribution of time-averaged dimensionless total pressure loss for all turbulent models and experimental data [1] at the exit. The results of SST and *k-ω* model are almost identical and have better agreement with experimental data than with *ω*RS model.

Figure 5 shows the distribution of the time-averaged dimensionless total pressure loss at the exit using three different grids for SST turbulence model and the experimental data [1]. The total number of grid points for coarse, original, and fine grid systems are about 0.66×10^5, 1.24×10^5, and 1.7×10^5, respectively. All grids have y^+ less than one. The current study was carried out using the original grid. Figure 5 shows that the results are almost identical and independent of considered grid systems. Therefore, the original grid is valid for the present study.

The time-averaged total pressure loss of SST and *k-ω* turbulent models have much better agreement with experimental data [1] than that of *ω*RS turbulent model, and the distribution for SST and *k-ω* turbulent models is almost identical as shown in Figure 4. However, the time-averaged total pressure contours of these two turbulent models show subtle differences even though they are quite small. The time-average total pressure contours of SST and *k-ω* turbulent models in the wake region are shown in Figure 6.

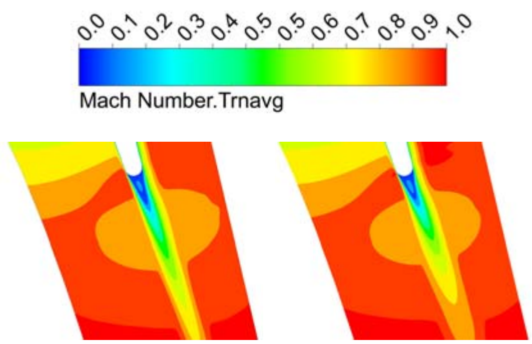


(a) (b)

Figure 6: Time-averaged total pressure contours (a) *k-ω* (b) SST model

4.3 Mach Number and pressure contours

Because the unsteady calculation of *ω*RS turbulent model has poor agreement with experimental data, we did not present the results here. Also, because of the similarity in the results between *k-ω* and SST turbulent models, we present only the contours of SST turbulent model henceforth.



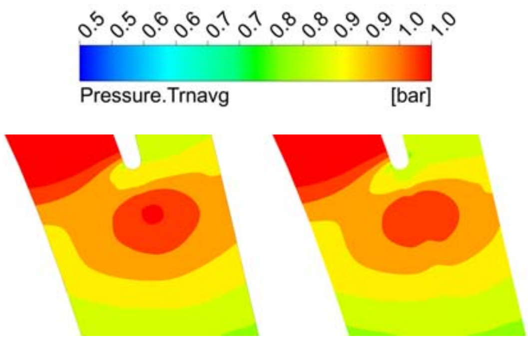
(a) (b)

Figure 7: Mach number contours for SST model (a) steady (b) unsteady (time-averaged)

Since the total pressure is a function of Mach number and static pressure, we present Mach number and the static pressure contours in Figs. 7 and 8 respectively.

Figure 7 shows the time-averaged Mach number contours for unsteady calculations and steady calculation. There is a minor difference between the time-averaged Mach number contours for unsteady case and for steady case. For example, the far wake is wider in the unsteady calculation than in the steady calculation.

Also, near the trailing edge, low Mach number region in the steady case is larger than that in the unsteady case. Inside this region, there is a little higher Mach number region which is larger in the unsteady case than steady case.

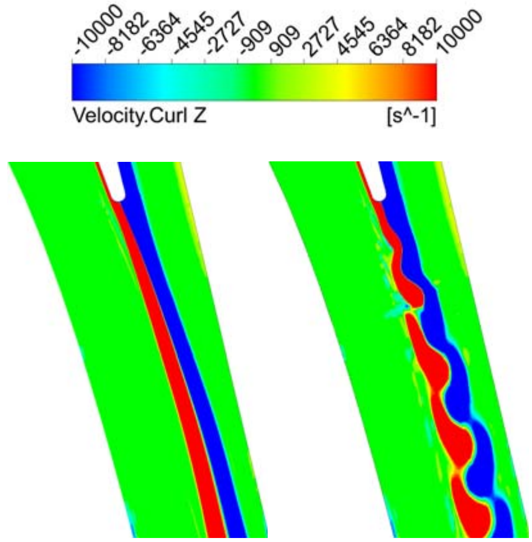


(a) (b)

Figure 8: Static pressure contours for SST case (a) steady (b) unsteady (time-averaged)

In Figure 8, the static pressure distributions for unsteady calculations (time-averaged) and for steady calculation are shown for comparison. They exhibit a small but substantial difference in that steady calculation shows larger high pressure region in the wake area.

Since both Mach number and static pressure between the steady and the unsteady case (time-averaged) are different, we can expect that total pressure would also have difference between the steady and unsteady case.



(a) (b)

Figure 9: Vorticity contours for SST case (a) steady (b) unsteady (instantaneous)

4.3 Instantaneous results

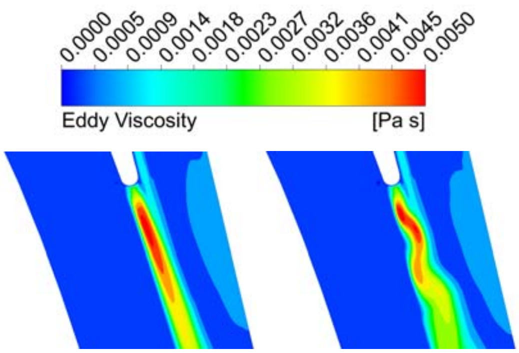
Figure 9 demonstrates the vorticity distributions of steady and unsteady calculations. The steady calculation shows a pair of straight and thin high vorticity strips. On the other hand, the coherent structure of the vortex shedding is clear for the unsteady case, as shown in Fig. 9. Therefore, periodically repeated vortex shedding can be viewed as the main source of the differences between the unsteady and the steady calculations for the time-averaged total pressure, static pressure and Mach number as shown in Figures. 3, 7 and 8 respectively.

This vortex shedding of unsteady calculations promotes the turbulent mixing and faster turbulent kinetic energy decay. Figure 10 compares the eddy viscosity between the steady and the unsteady calculations for SST turbulence model. For steady calculation, the high eddy viscosity region is concentrated in the thin wake center. On the other hand, unsteady calculation shows wider and wavy, but relatively lower eddy viscosity region.

5. Conclusion

The numerical simulations of the flow field in a high pressure turbine cascade were carried out. The effect of unsteadiness due to the vortex shedding was considered. The results of steady and unsteady calculations of three turbulence models were compared.

The computation showed that calculation results with all three turbulent models are in reasonably good agreement with experimental data along the blade surface. There was only a small deviation between the predicated and experimental results on the rear part of the suction side for both steady and unsteady calculations. At the exit, the results of unsteady calculations of SST and k-ω turbulence models, in particular, have much better agreement with experimental data than steady calculations



(a) (b)

Figure 10: Eddy viscosity contours for SST case (a) steady (b) unsteady (instantaneous)

References

[1] Arts T., Rouvroit L. D. and Rutherford A. W., "Aerothermal investigation of a highly loaded transonic linear turbine guide vane", VKI Technical Note 174, 1990.

[2] Liu Y., "An efficient numerical method for highly loaded transonic cascade flow", Eng. Math, vol. 60, pp. 115-124, 2008.

[3] Gehrer A. and Jericha H., "External heat transfer predictions in a highly-loaded transonic linear turbine guide vane cascade using an upwind biased Navier-Stoke solver", In Seminar and Workshop on Turbomachinery Flow Prediction VI, 1998.

[4] Larsson J., "Turbine blade heat transfer calculations using two-equations turbulence models", J. of Power and Energy, vol. 211, pp. 153-262, 1997.

[5] Barth T. J. and Jespersen D. C., "The design and application of upwind schemes on unstructured meshes", AIAA paper 89-0366, 1989.

[6] Menter F. R., "Two-equation eddy viscosity turbulence models for engineering applications", ALAA Journal, vol. 32, P-8, pp. 1598-160, 1994.

[7] Wilcox D. C., "Turbulence modeling for CFD", DWC industries, INC, 2000.

Author Profile

M. M. El-Gendi

He received his B.Sc. and M.Sc. degrees in Mechanical Engineering from Minia University, Egypt in 1998 and 2003, respectively. He then received his Ph.D. degree in Aerospace Engineering from Nagoya University, Japan in 2010. Since 2010, he has been an assistant professor in Minia university. In 2011, he had a Post-doc position in University of Ulsan, South Korea. His research interests are in fluid mechanics and turbomachinery.

Sang-Wook Lee

He received his B.S. and M.S. degree in Naval Architecture and Ocean Engineering from Seoul National University in 1992 and 1994. He then received his Ph.D. degree in Mechanical Engineering from the University of Illinois at Chicago in 2005. Currently, he is an assistant professor of the School of Mechanical Engineering at the University of Ulsan.

Chang-Ho Son

He received his B.S. degree in Mechanical Design from Seoul National University in 1978 and then M.S degree in Mechanical Engineering from Korea Advanced Institute of Science and Technology in 1980. Then he received his Ph.D. degree in Mechanical Engineering from State University of New York at Stony Brook in 1992. Currently, he is an associate professor of the School of Mechanical Engineering at the University of Ulsan.