Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-30640

NUMERICAL STUDY ON ROTOR-STATOR INTERACTION OF A SUPERSONIC REACTION TURBINE FOR A LIQUID ROCKET ENGINE

Kaname Kawatsu
Japan Aerospace Exploration Agency
2-1-1 Sengen, Tsukuba-shi, Ibaraki, Japan

Naoki Tani
Japan Aerospace Exploration Agency
2-1-1 Sengen, Tsukuba-shi, Ibaraki, Japan

Nobuhiro Yamanishi
Japan Aerospace Exploration Agency
2-1-1 Sengen, Tsukuba-shi, Ibaraki, Japan

ABSTRACT

For an open cycle liquid rocket engine, such as the expander bleed cycle[1,2], the mass flow rate of turbine driving gas should be small, especially to improve rocket engine performance. However, work output must be high as possible. As a result, pressure ratio of the turbine becomes high, and Mach number at both nozzle exit and rotor inlet becomes supersonic[3]. As a result, strong shock wave interaction can be generated between nozzle exit and rotor inlet, and this interaction affects the turbine aerodynamic performance[4]. However, this rotor-stator interaction of supersonic turbine has not yet been clarified. Therefore, as the first step, it is important to clarify the structure of the flow field and to evaluate the accuracy of CFD method as practical engineering tool for liquid rocket engine design.

In the present study, quasi 3-D RANS simulations were applied to the NACA supersonic turbine[5] and the numerical results were compared with the experimental ones to evaluate numerical methodology. Turbulence models and rotor/stator interface modeling method were compared, and their impacts to the turbine aerodynamic performance estimation were evaluated. In addition to these points, the flow field between nozzle and rotor region and the turbine efficiency were investigated. The present results clarify some features of rotor-stator interaction. The shock wave, which is generated near the nozzle exit caused by encounter of nozzle exit flow, reflects at the neighbor nozzle wall and affects the rotor region. At the same time, the shock wave from the rotor leading edge impinges the nozzle cascade, and these shocks interact with each other. The present results showed that Mach number at nozzle outlet becomes different due to each turbulence and rotor/stator interface models. This difference of Mach number influences the shape of detached shock wave at the leading edge of rotor blade, and changes the entire rotor region flow field such as static pressure profile of rotor region. Thus, turbine efficiency may be influenced by these different features of flow field.

INTRODUCTION

In order to achieve high engine performance, a high pressure ratio turbine was chosen for an open cycle liquid rocket engine, because the driving gas is bleeding without contribution to the main chamber combustion. Furthermore, low weight and high performance are required for component of rocket engine, thus work output of turbine has to be high with lower stage number. Consequently low reaction-impulse turbine is adopting, hence the rotor inlet flow speed increase due to high pressure ratio to achieve high work output. As a result, supersonic flow field is formed between nozzle exit and rotor inlet and strong shock wave interaction can be generated between nozzle exit and rotor inlet, and this interaction affects the turbine aerodynamic performance.

Therefore characteristic features of this interaction should be clarified to enhance the accuracy of prediction for performance of supersonic turbine. A high accuracy is needed for prediction of supersonic turbine performance, because the turbine performance is key parameter to determine the engine cycle performance.

NOMENCLATURE

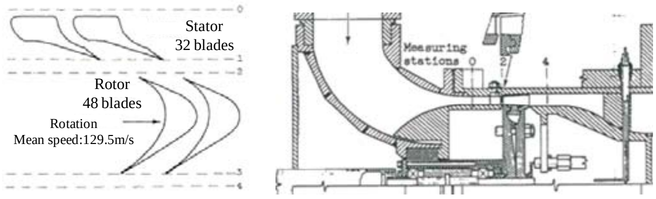
BPF	blade passing frequency
C_P	constant pressure specific heat
h	entropy
P	pressure
T	temperature
y^+	non-dimensional wall distance
(Greek letters)	
γ	ratio of specific heat
(Subscripts)	
IN	inlet of turbine
OUT	outlet of turbine
s	static
t	total
TS	adiabatic static

NUMERICAL ANALYSIS OBJECT

The object of numerical analysis of this study is NACA supersonic turbine[5] since results and configurations of turbine blades have been published. Figure 1 shows configuration of turbine blades and experimental conditions.

In the experiment, heated air of which total pressure is 0.52MPa and total temperature is 444.3K was used as driving fluid, so the ideal gas assumption was adopted.

The turbine consist with 32 stator blades and 48 rotor blades, thus turbine flow paths with two stator blades and three rotor blades had been choose as a flow field of numerical calculations.



Experimental conditions

Driving fluid	Air
Inlet total pressure	0.520MPa
Inlet total temperature	444.3K
Pressure ratio	30

FIGURE 1. CONFIGURATION OF TURBINE AND EXPERIMENTAL CONDITIONS [5]

NUMERICAL METHOD

In this study, FLUENT 6.3.26[6] was used for the numerical simulation of a flow field in turbine flow paths. Three-dimensional (3-D) Reynolds averaged Navier-Stokes(RANS) equations were solved with local time step implicit or explicit method. The convective fluxes were evaluated with AUSM scheme and 2nd space accuracy was applied.

In the present study, evaluation of numerical models, which are often used in CFD for design phase, was carried out. To clarify the influence of numerical models, turbine efficiency was compared.

As the objects of comparison for turbulence model, Realizable k-ε model with EWT (Enhancement Wall Treatment)[6], which is widely used to solve practical engineering problems, and SA (Spalart and Allmaras) model[7], which is mainly used to solve supersonic problems.

Meanwhile, the rotor/stator interface modeling methods were evaluated by comparing two methods. One is a Frozen Rotor model in which the rotating system is treated with static system, so the calculations were carried out easily. Another one is a Mixing Plane model in which flow fields of rotational direction are circumferentially averaged, so this model is useful to get approximate time averaged results with low calculation costs.

Passage height change is small and the hub-tip ratio is high in the present object, therefore 3-D effect of flow field in the turbine is weak. In the present study, two-dimensional (2-D) calculation is appropriate to capture flow field characteristics. The computational domain, consisted of two stator blades and three rotor blades per row, was meshed with 150,000 hexahedra unstructured grids, and the whole region averaged wall y^+ was less than 5. The boundary conditions are described in Fig 2. The wall was treated as adiabatic and no-slip wall.

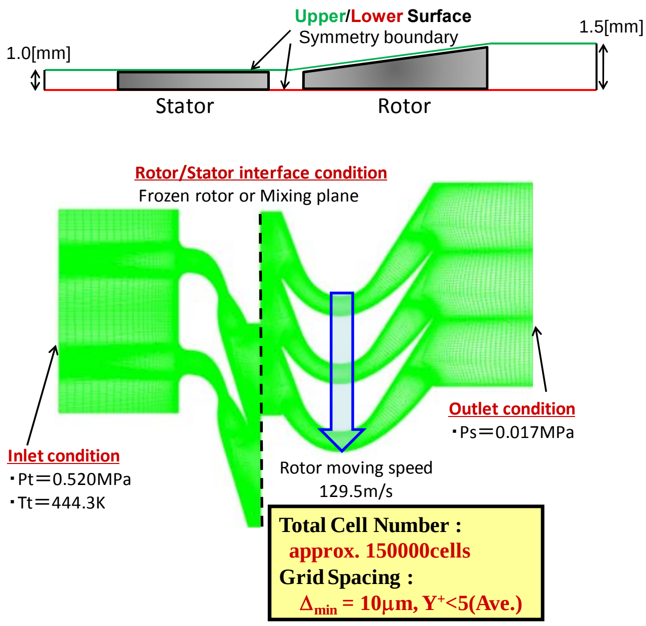


FIGURE 2. COMPUTATIONAL DOMAIN AND BOUNDARY CONDITIONS

STEADY STATE NUMERICAL RESULTS

CHARACTERISTICS OF FLOW FIELD

Figure 3 shows a contour of static pressure and a schematic drawing shock waves which are generated between nozzle exit and rotor inlet. In this section, a result of the Realizable k-ε model with Frozen Rotor model was used.

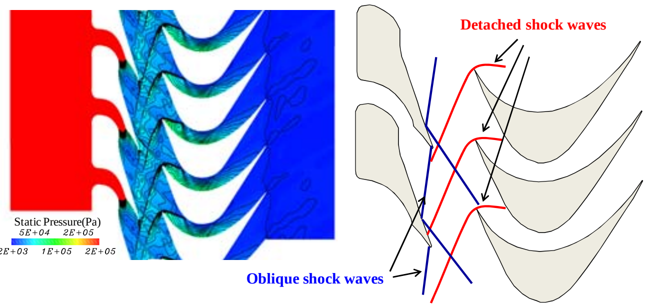


FIGURE 3. STATIC PRESSURE CONTOURS AND SCHEMATIC OF SHOCK WAVES

Oblique shock waves are generated with the encounter of flows at the nozzle trailing edge. The oblique shock waves reflect at the stator wall and impinge to rotor inlet. At the same time, the detached shock waves, which are generated at the leading edge of rotor blade, impinge to stator blade.

As previously described, a complicated flow field that contains reflection and inflection of shock waves is created between nozzle exit and rotor inlet of supersonic turbine.

EFFECT OF TURBULENCE MODEL

Figure 4 shows plots of static pressure of experimental and numerical results. The static pressure of numerical results at rotor region is circumferentially averaged. To evaluate the effect caused by difference of turbulence models, Realizable k-ε and SA were compared and Frozen Rotor model was used as the rotor/stator interface modeling in these cases.

According to the plots of static pressure, the numerical results show qualitative agreement to experimental results in both cases. At the same time, a difference of distribution of static pressure is presented at the region drawing with red dot-line in Fig. 4 near the incident point of detached shock wave.

Figure 5 shows contours of Mach number of numerical results with the Realizable k-ε or SA model. In these results, the difference of Mach number at rotor inlet is observed and the shock wave angle becomes smaller with the Realizable k-ε model. This difference appears as the distinction of oblique shock waves near the leading edge of rotor. Therefore, the difference of distribution of static pressure is expected as the result of difference of rotor inlet Mach number.

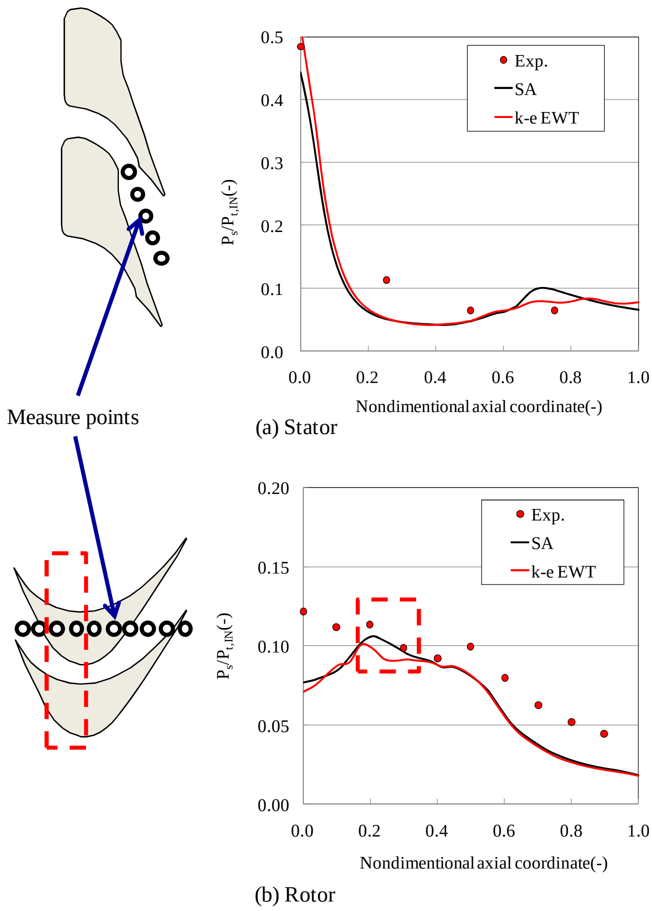


FIGURE 4. STATIC PRESSURE DISTRIBUTIONS

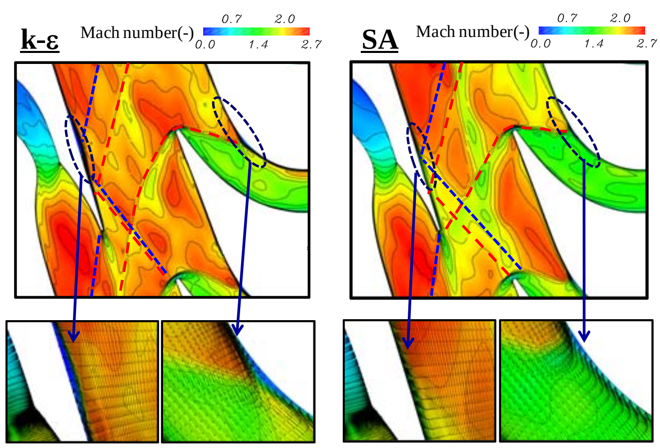
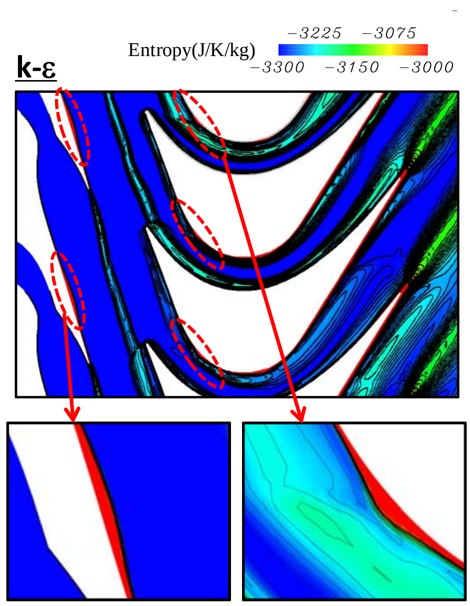
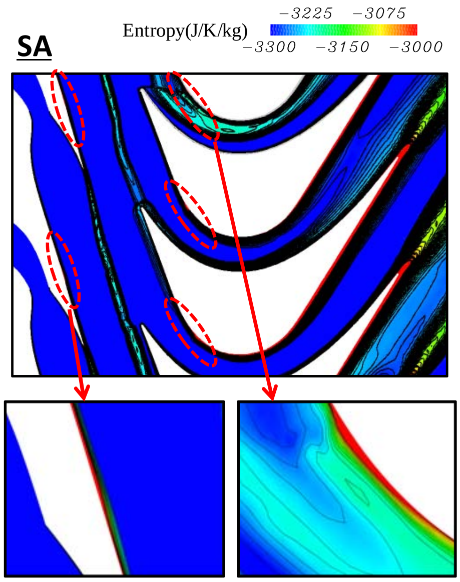


FIGURE 5. MACH NUMBER CONTOURS AT ROTOR/STATOR INTERFACE



(a) Realizable k-ε model case



(b) SA model case

FIGURE 6. ENTROPY CONTOURS AT ROTOR/STATOR INTERFACE

The most noticeable difference by the turbulence model is a separation region caused by the shock wave reflection. The size of separation at stator and rotor wall in the regions, which are indicated with black dot-line in Fig. 5, is different. The Realizable k-ε model predicts lager separation region than the SA model.

It can be confirmed with these results that we have to select an appropriate turbulence model to predict the flow field which contains shock waves and separations of boundary layer such as flow paths of supersonic turbine.

To evaluate an aerodynamic loss of turbine, entropy contours are compared in Fig. 6. The entropy distribution contours show clear correlation to the Mach number contours. As shown in Fig. 5, separation region which is predicted by Realizable k-ε model is larger than the one by SA model and high entropy area in Fig. 6 can be seen at these separation regions. The present results show that small boundary layer separation may affect higher turbine efficiency.

EFFECT OF ROTOR/STATOR INTERFACE MODEL

Figure 7 shows a contour of Mach number by Mixing Plane case with Realizable k-ε model. To keep the stability of the calculation, expanding regions drawn with red dot-line were added between the stator outlet and rotor inlet. The regions were adopted, because the inflection of shock waves to interface region between stator and rotor causes instability of the calculation. As a result of circumferential averaging at rotor/stator interface, the Mach number of inlet flow is predicted lower than that of Frozen Rotor.

Figure 8 shows plots of static pressure of experimental and numerical results. The static pressure of numerical results at rotor part is circumferentially averaged. The effect of averaging can be also observed as a difference of static pressure plots as shown Fig. 8. The prominent pressure rise can be observed in Mixing Plane case, and this is a result of the influence of detached shock wave from neighboring rotor blade. In the Mixing Plane case, inlet Mach number becomes higher. Thus, lager pressure rise is caused near the leading edge part.

Figure 9 indicates the difference of the distribution of entropy between Mixing Plane and Frozen Rotor cases. This difference is also caused by averaging. The flow is averaged circumferentially at rotor/stator interface and entropy at the rotor inlet becomes higher than the Frozen Rotor case.

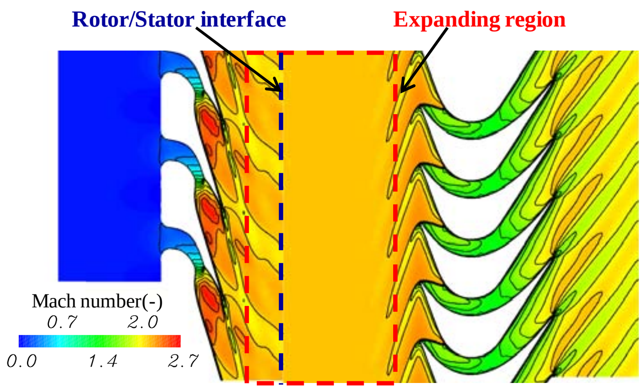
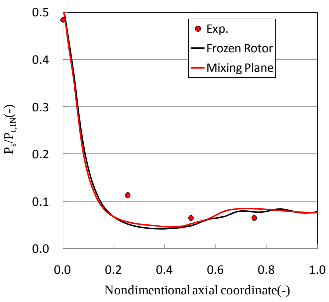
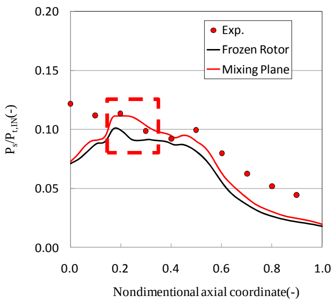


FIGURE 7. MACH NUMBER CONTOUR IN MIXING PLANE CASE

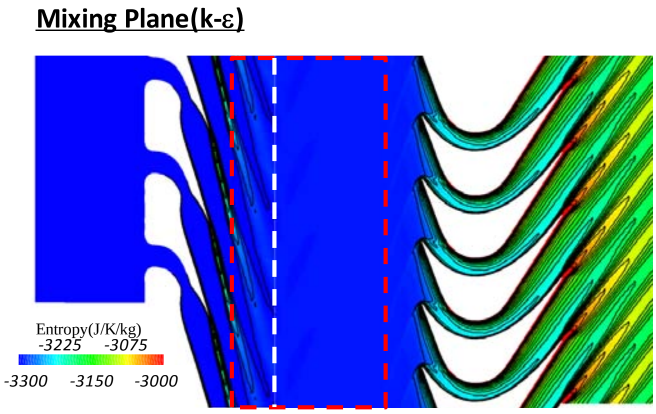


(a) Stator

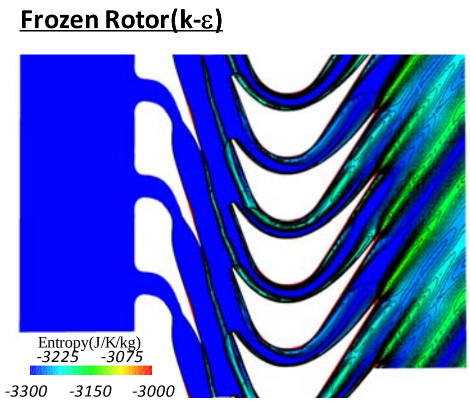


(b) Rotor

FIGURE 8. COMPARISON OF STATIC PRESSURE DISTRIBUTIONS



(a) Mixing Plane case



(b) Frozen Rotor case

FIGURE 9. COMPARISON OF ENTROPY CONTOURS

EVALUATION OF TURBINE EFFICIENCY

As the final step, effects of numerical models were evaluated on the difference of turbine efficiency. The adiabatic static efficiency η_{TS} is defined as follows:

$$\eta_{TS} = \frac{ht_{IN} - ht_{OUT}}{CpTt_{IN}\left(1-\left(Ps_{OUT}/Pt_{IN}\right)^{(\gamma-1)/\gamma}\right)} \qquad (1)$$

This parameter is important for engine cycle analysis, especially for feasibility study step. The efficiency comparison between numerical analysis and experimental result are shown in Fig. 10. From this result, the efficiency is different about 16% in maximum. The minimum efficiency is predicted with Mixing Plane rotor/stator model case because of the rising of entropy caused by circumferential averaging at rotor/stator interface. From an engineering point of view, less than 5% prediction error is desired. However, present difference is much higher than the desirable value. This results means that turbulence model should be carefully selected for CFD of supersonic turbine.

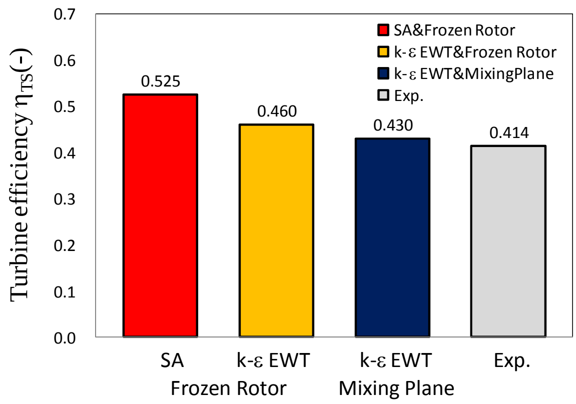


FIGURE 10. ADIABATIC STATIC EFFICIENCY OF TURBINE

UNSTEADY NUMERICAL RESULTS

In this section, unsteady results are described and discussed. The Realizable k-ε is used as a turbulence model for unsteady case. A fluctuation of the efficiency is shown in Fig. 11. There are 6 cycles per a pitch, which consists with 2 stator blade and 3rotor blades. Figure 12 shows spectrum of the efficiency and the frequency was normalized with stator blade passing frequency (BPF). The strong amplitude can be observed at Frequency/BPF=3. This spectrum associates that the unsteady tendency of performance is dominated by the rotor passing frequency.

The instantaneous contours of Mach number are shown in Fig. 13. In these figures, points (A) and (B) were select as viewpoints. The difference can be seen in the shapes of detached shock waves at rotor leading edges, which are also observed at static pressure distributions as shown in Fig. 14. At point (B) the angle of detached shock waves, which are indicated with red dot-line, is lower than that at point (A). The shape of detached shock wave affects to a boundary layer separation which influents to an aerodynamic loss. At point (B), the separation becomes lager than the one at point (A). As a result, the efficiency is estimated lower. The fluctuation of the efficiency is caused by this difference of detached shock wave shape. The difference of shape is caused by the change of rotor inlet Mach number, which is determined by a relative location of rotor/stator and interaction of detached shock waves at rotor leading edge. If the Mixing plane was used as a model of rotor/stator interface, these unsteady phenomena cannot be predicted with the averaging at rotor/stator interface.

A contour of root mean square (RMS) of static pressure is shown in Fig. 15. This result shows that, variations of flow field are observed near the detached shock waves at leading edge of rotor. Strong variation occurs at the boundary layer separation point, which is caused by interaction of the detached shock wave, and rotor wall boundary layer. From these results, the shape of the detached shock wave at rotor leading edge dominates the unsteady characteristic of supersonic turbine. Thus, the important point to estimate the unsteady performance of supersonic turbine is the prediction of the separation caused by shock wave interaction at rotor blade wall. It suggests that appropriate turbulence model has to be selected.

The difference of the efficiency difference between the Mixing Plane and the unsteady case is about 1%. This difference is small compared to 5% error criteria for engineering design.

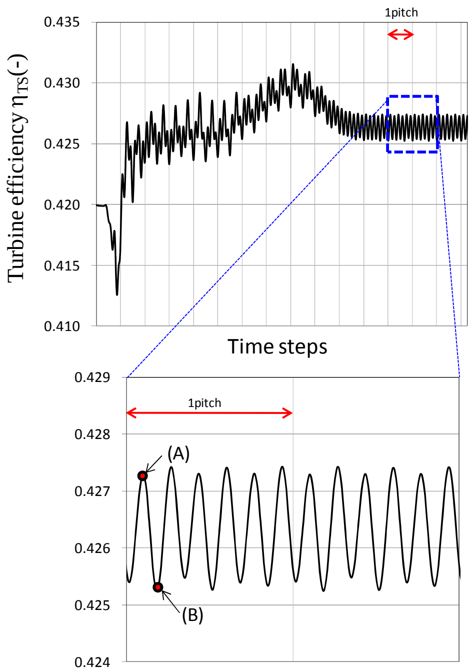


FIGURE 11. FLUCTUATION OF TURBINE EFFICIENCY IN UNSTEADY CASE

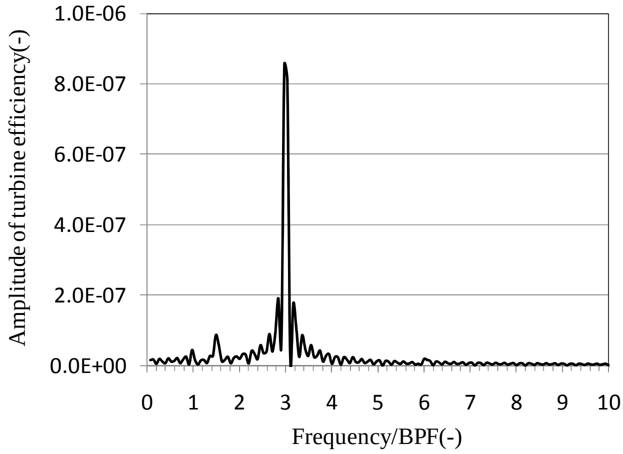


FIGURE 12. SPECTRUM OF TURBINE EFFICIENCY IN UNSTEADY CASE

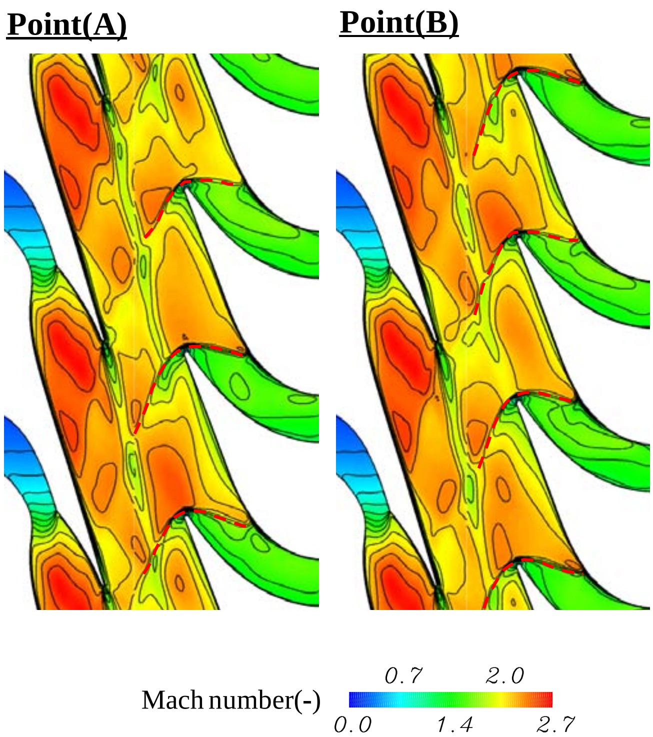


FIGURE 13. INSTANTANEOUS MACH NUMBER CONTOR IN UNSTEADY CASE

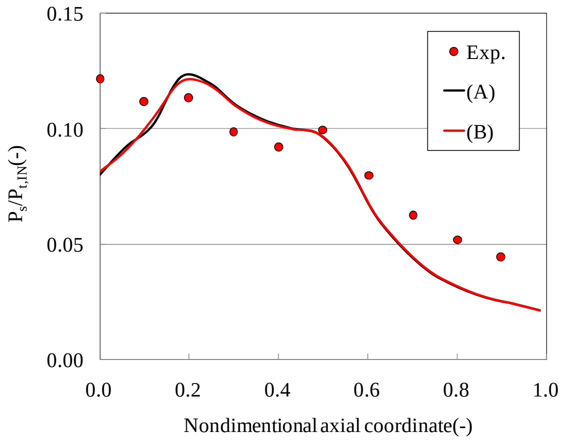


FIGURE 14. STATIC PRESSURE DISTRIBUTIONS AT EACH TIME

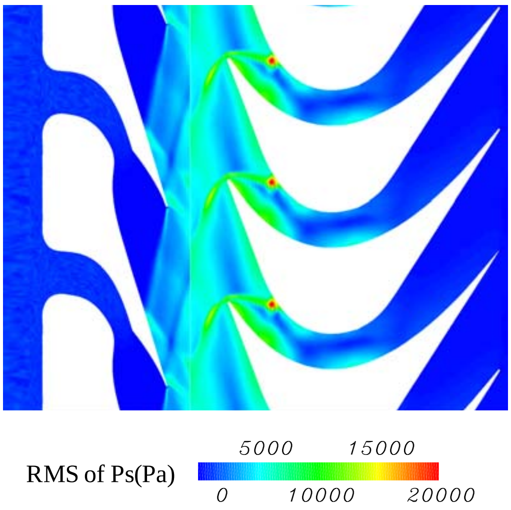


FIGURE 15. RMS OF STATIC PRESSURE

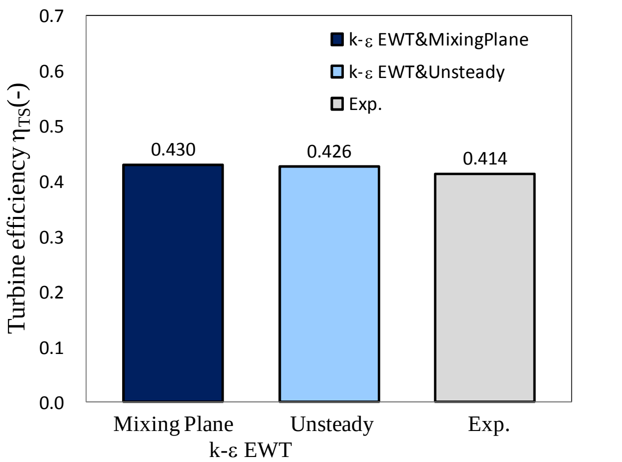


FIGURE 16. COMPARISON OF TURBINE EFFICIENCY

CONCLUSION

In this study, quasi 3-D RANS simulations were applied to the NACA supersonic turbine to evaluate engineering numerical approach, and the numerical results were compared with the experimental results. Investigations are done on the flow field between nozzle and rotor region and the turbine efficiency. Turbulence models and rotor/stator interface modeling methods were compared, and its impact to the turbine aerodynamic performance was evaluated. The results are summarized as follows.

- The Realizable k-ε model predicted lager separation, caused by shock inflection at the stator and rotor blades, which was compared to SA model. This may affect the turbine efficiency.

- In the case of using Mixing Plane as a rotor/stator interface, the Mach number of inlet flow becomes lower and entropy at the rotor inlet becomes higher than these of Frozen Rotor. The predicted turbine efficiency with Mixing Plane model was lower than with Frozen Rotor model.

- The difference of predicted turbine performance between Mixing Plane analysis and unsteady analysis was small. Thus, Mixing Plane maybe applicable to practical applications from engineering design point of view.

REFERENCES

[1] M. Kojima, et al., “Preliminary Design and Analysis for the LE-X Engine Components”, 46th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, AIAA 2009-5485, 2009.

[2] G. P. Sutton, “Turbopumps, a Historical Perspective”, 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, AIAA 2006-5033, 2006.

[3] C. D. Colclough, “Design of Turbine Blades Suitable for Supersonic Relative Inlet Velocities and the Investigation of Their Performance in Cascades: Part 1 Theory and Design”, Journal Mechanical Engineering Science, Vol.8 No.1, 1966.

[4] M. Tanaka, et al., “Investigation of the Two-dimensional Performance of Supersonic Impulse Turbine Blade Cascades (2nd Report, Flow and Losses in Blade Passage and Some Design Criteria)”, Bulletin of JSME, Vol. 27, No.255, 1984.

[5] T. P. Moffitt, “Design and Experimental investigation of a Single-Stage Turbine with a Rotor Entering Relative Mach Number of 2”, NACA RM E58F20a, 1958.

[6] http://www.ansys.com/products/fluid-dynamics/fluent/

[7] P. R. Spalart, and S. R. Allmaras, "A One-Equation Turbulence Model for Aerodynamic Flows," Recherche Aerospatiale, No. 1, pp. 5-21, 1994.