Amirkabir University of Technology
(Tehran Polytechnic)
Vol. 47, No. 1, Summer 2015, pp. 1- 3

Amirkabir Journal of Science & Research (Mechanical Engineering)
(AJSR - ME)

Investigation of Different Turbulence Models Performance on High-turning Turbine Blade Loading Calculations

R. Saberi[1], M. Fathali[2*]

1- MSc student, Department of Aerospace Engineering, K.N. Toosi University ofTechnology, Tehran, Iran
2- Assistant professor, Department of Aerospace Engineering, K.N. Toosi University ofTechnology, Tehran, Iran

(Received 13 July, 2011, Accepted 31August, 2014)

ABSTRACT

In this research, the performances of different turbulence models for simulating flow field in turbine stator have been numerically investigated. To this end, incompressible fluid flow around a high-turning stator blade in Reynolds 2.23×10^5 has been simulated using FLUENT CFD software. Navier-Stokes equation is discretized on a Hybrid computational grid based on the finite volume approach. Considered turbulence models in this research are "Spalart-Allmaras", "Standard k-ε", "Realizable k-ε", "RNG k-ε" , "SST k-ω" and "five-equation Reynolds-Stress Model (RSM)". The performances of these models are evaluated by comparing the pressure coefficients obtained from numerical simulations with corresponding experimental data at four different stator regions. It has been observed that the ability of a turbulence model to predict flow field is not uniform throughout the stator blade. Moreover, all models show relatively poor performance in flow field regions with intense velocity gradients. Comparing the overall accuracy of different models, SST k-ω and RSM turbulence models show the best agreement with the experimental data.

KEYWORDS

Gas Turbine, Turbulence Model, High-turning Blade, Blade Pressure Distribution.

* Corresponding Author, Email: manifat@yahoo.com

1- INTRODUCTION

The gas turbine is widely used for the power generation in power plants, marine and aerial propulsion systems. Therefore, increasing turbines efficiencies is crucially important from industrial and economical point of views. Among the other parameters, increasing the overall efficiency of a gas turbine through improving the profile of turbine blades has attracted a lot of attention [1, 2].

Since experimental investigations in gas turbines are highly expensive, using the numerical simulation in gas turbine investigation has been considered as an attractive alternative. Obviously, due to the relatively high Reynolds number of the flow field in a gas turbine, one of the most important issues for flow field simulation is appropriate turbulent flow modeling [3]. Although a variety of Reynolds Averaged Navier-Stokes equations (RANS) turbulent models have been developed for turbulent flow simulations, none of them are specifically designed for simulating turbo machinery flows.

Therefore, one of the main issues which remains open to discussion is to quantify the performances of different turbulent models in simulating turbo- machinery flows. Closely related to this question, this research aims at two nontrivial goals as follow. First, we examine the ability of the FLUENT CFD software for simulating extremely complex turbo-machinery fluid dynamics. Second, the overall performances of six most widely used turbulence models have been compared by considering relative errors at critical flow field regions.

2- METHODOLOGY

As mentioned above, in this research the performances of five most widely used turbulent models for predicting the dynamics of an incompressible turbulent flow around a high-turning stator blade in a low-pressure gas turbine is investigated. Quantification has been carried out by comparing the numerical results with the experimental data of S. W. Lee *et. al.* [4]. Considered turbulent models in this research are one-equation model (Spalart-Allmaras), four different two-equation models (Standard k-ε, Realizable k-ε, RNG k-ε and SST k-ω) and a five-equation Reynolds-stress model (RSM).

Navier-Stokes equation for a two-dimensional, viscous, incompressible flow with Reynolds number 2.23×10^5 based on the inlet velocity and the blade chord has been numerically solved. Governing equations are discretized based on the second-order upwind finite volume approach. The computational domain consists of a stator blade with the periodic boundary conditions at the top and bottom of the domain.

Further, velocity, turbulent intensity, and integral length scale of the velocity perturbations are imposed at the inflow boundary condition based on the experimental data of [5]. At the downstream, the static pressure is fixed as the outflow boundary condition. The computational domain is discretized using 29302 irregular hybrid structured grid cells: in the boundary layer regions, the structured grid is used while for the outer regions, an unstructured grid is used. Using this grid cells results in an independent grid solution for the pressure coefficient. Discretized equations are implicitly solved based on the SIMPLE algorithm using FLUENT 6.3.26 software package.

Moreover, the overall performances of different models are quantitatively measured by considering the relative error of the pressure coefficient. This relative error is computed at four critical flow field zones as follow. The zone (1) is the stagnation point where the pressure coefficient should be equal to one. The zone (2) is near the trailing edge on the suction side at L/C=1 where the turbulent intensity is considerably high. The zone (3) is located at the separation region on the pressure side at L/C=0.06 where flow field starts to separate. Finally, the zone (4) is at L/C=0.06 on the suction side where flow field shows intense velocity gradients.

3- RESULTS

The results are presented based on the two quantities. First, the pressure coefficients at the above-mentioned regions are compared with the experimental data. Pressure coefficient is the most important parameter in this research, which is closely related to the turbine efficiency. This quantity is defined as $C_p= (P_s-P_\infty)/q_\infty$ where P_s and P_∞ are respectively static pressure on the blade surface and the inlet static pressure. Moreover, q_∞ is the inlet dynamic pressure.

For the sake of the comparison, relative error lower than 1% is considered as the excellent quality, between 1% till 10% as good quality, between 10% till 35% as the average quality and higher than 35% as the poor quality. Overall, it is found that almost all six turbulence models show poor qualities for simulating flow fields in regions (2) and (4). Moreover, the simulation error is higher at the pressure side of the blade compared to the suction side. The results and the qualities of different models in simulating the flow fields at different regions are presented in tables 1 and 2.

As the further indication of the performance of the turbulence models, the velocity in the vortical flow field region is considered. This quantity can represent the essential features of the wake region in which the velocity gradients and the turbulent intensity are considerably high. The SST k-ω, due to including the effects of the Reynolds stresses, shows the lowest relative error in predicting the vortical velocity.

Next, the RSM model shows the highest accuracy. The lower accuracy of this model compared to the SST k-ω mode is originated from its sensitivity to the near-wall region. The Standard k-ε model shows the lowest accuracy. This poor performance is improved in k-ε RNG by including the effect of the flow field small-scale structures. The quality of the k-ε family models in predicting vertical velocity is further improved in k-ε Realizable by considering the effect of vorticity in the governing equation of ε. Interestingly, despite the relatively simple structure of the Spalart-Allmaras model compared to the k-ε family models, this model shows considerably better performance in predicting the vortical velocity. This better performance can be attributed to its inherent character of the method in ability of predicting high-turning flows.

4- CONCLUSION

In this research, the performances of six widely used turbulence models, namely Spalart-Allmaras, k-ε families (Standard k-ε, Realizable k-ε, RNG k-ε, and SST k-ω) and the Reynolds-stress model RSM, for turbo machinery simulations are investigated. The test case is considered as the flow field around a high-turning turbine blade.

All simulations are carried out at the Reynolds number 2.23×10^5, using FLUENT 6.3.26 software package. The governing equations are discretized on a 29302 irregular hybrid structured grid cells, using second-order upwind finite volume approach.

In results, the pressure coefficients at four critical flow field regions as well as the velocity in the vortical flow field region are considered. To quantify the accuracy of these turbulence models, these quantities are compared to the experimental data [5].

Although the accuracies of these models change form point to point throughout the flow field, an overall comparison among these models can be carried out. The results reveal that the SST k-ω and RSM turbulence models are generally in good agreement with the experimental results.

However, the computational overhead of the five-equation RSM model is considerably higher than that in the two-equation SST k-ω turbulence. Moreover, the Standard k-ε model shows the lowest accuracy. The rest of the models show performances between these two limits. Therefore, SST k-ω and RSM models can be considered as the most suitable candidates for turbo machinery simulations.

5- REFERENCES

[1] S. Burguburu, C. Toussaint, G. Leroy, “Numerical optimization for turbomachinery blades aerodynamic design using a gradient method coupled with a Navier-Stokes solver”, ISABE 1117, 2001.

[2] G. Brereton, T. Shih, “Turbulence modeling in simulation of gas-turbine flow and heat transfer”, ANNALAS of the New York Academy of Sciences journal of Heat Transfer in Gas Turbine Systems, Vol. 934: pp. 52-63, May 2001.

[3] F. Menter, “Performance of popular turbulence models for attached and separated adverse pressure gradient flows”, AIAA Journal 30(8): pp. 2066-2072, 1992.

[4] S. W. Lee, B. J. Chae, “Effects of squealer rim height on aerodynamic losses downstream of a high- turning”, Elsevier journal of Experimental Thermal and Fluid Science, vol. 2, No 38, 2008

[5] D. Lastiwka, “Influence of rotor blade scaling on the numerical simulation of a high pressure gas turbine” M.A. Thesis, Ottawa-Carleton Institute for Mechanical and Aerospace Engineering University of Ottawa, Ottawa, Ontario, Canada, May 14, 2009.

Table 1. Comparison of Cps calculations by turbulence models at critical regions with experimental results.

Turbulence model	Zone 1		Zone 2		Zone 3		Zone 4	
	Cps	% error	Cps	% error	Cps	% error	Cps	% error
Experimental data	1	---	-0.45	---	0.62	---	-1.24	---
Spalart-Allmaras	1.11	11	-0.71	57.8	0.53	14.5	-0.80	35.5
Standard k- ε	1.35	35	-0.74	64.4	0.43	30.6	-0.60	51.6
k-ε Realizable	1.01	1	-0.62	37.8	0.54	12.9	-0.79	36.3
k-ε RNG	1.12	12	-0.61	35.6	0.52	16.1	-0.78	37.1
SST k-ω	1.01	1	-0.60	33.3	0.62	0	-0.85	31.4
RSM	1.01	1	-0.58	28.9	0.57	8	-0.85	31.4

Table 2. A qualitative comparison of the turbulence models in critical regions.

Turbulence model	Zone 1	Zone 2	Zone 3	Zone 4
Spalart-Allmaras	ok	Poor	ok	ok
Standard k- ε	poor	Poor	poor	poor
k-ε Realizable	good	Ok	ok	ok
k-ε RNG	ok	Ok	ok	ok
SST k-ω	good	Ok	Excellent	ok
RSM	good	Good	good	ok