Evaluation of different turbulence models in simulating the subsonic flow through a turbine blade cascade

Mohsen Mesbah¹, Vladimir Georgievich Gribin¹ and Kambiz Souri²

¹Department of Steam and Gas Turbines, National Research University "Moscow Power Engineering Institute", Krasnokazarmennaya Street, Building No. 14, Moscow, 111250, RU
²Peoples’ Friendship University of Russia, Miklukho-Maklaya Street, Building No. 6, Moscow, 105005, RU

mo.mesbah64@gmail.com

Abstract. Numerical simulations of the 2D turbulent flow of a viscous compressible gas through a turbine cascade were carried out. The main objective of this work is to evaluate the capability of different turbulence models in predicting the aerodynamic characteristics of a steady flow. Four turbulence models were considered; Spalart-Allmaras, realizable k-epsilon, k-omega SST, and SST transition model. The numerical results of the profile loss coefficient were compared with the available experimental data. The profile loss coefficient calculated using the SST transition model showed the most accurate agreement with the experimental data.

1. Introduction

The flow path through high loaded subsonic and transonic turbine cascades is very complicated due to the variation of flow regime, separation along the blade surface, along with vortex shedding from the trailing edge, and shock waves. Therefore, for a better understanding of these flows, it is important to investigate them experimentally or by reliable numerical simulations.

In recent years, thanks to the availability of high computational capabilities, Computational Fluid Dynamics became an increasingly widespread tool for investigating the aerodynamic characteristics of new machinery designs. Even though a large number of scientific works, such as [1,2], were devoted to the study and improvement of the turbine cascades with emphasis on the aerodynamic losses, all possibilities for their improvement have not yet been exhausted.

In most cases, flows in turbomachinery are three-dimensional, with all three flow regimes (laminar, transition, and turbulent) and this is one of the complex problems in the field of computational simulation of these flows. The equations most often considered for modeling turbulent flows in turbomachines are Reynolds-averaged Navier-Stokes equations (RANS), supplemented by a selected turbulence model.

As noted in [3,4], the numerical simulation of turbomachinery flows significantly depend on the reliable prediction of the turbulence. Over the past decades, a large number of studies have been carried out to consider the turbulent flow through the turbine or compressor cascades.

Different turbulence models were employed by Guo et al. [5], El-Gendi et al. [6], and Sorour et al. [7], to investigate heat transfer and aerodynamic characteristics through a turbine cascade.
A performance assessment of some eddy viscosity and some Reynolds stress turbulence models for modeling the flow through a compressor cascade was presented by Vlahostergios [8], and Zhang et al. [9].

In the present work, the performance of different turbulence models in predicting the aerodynamics characteristics of a steady flow through a subsonic turbine nozzle cascade is investigated. The calculation results were compared with the existing experimental data [1,14], and the most suitable models were recommended for use in similar calculations.

2. Numerical analysis procedures

2.1. Solver and turbulence closures

In this work, numerical studies of a steady-state flow were carried out using the ANSYS FLUENT software package, based on the numerical solution of the Reynolds-averaged Navier-Stokes equations (RANS) taking into account the continuity and energy equations. To close the Reynolds-averaged system of Navier-Stokes equations, the following turbulence models were considered for calculating the flow features in the boundary layer, including the viscous sublayer and buffer layer.

The first turbulence model is the Spalart-Allmaras model. It is a one-equation model that solves the transport equation for the kinetic vortex viscosity [10]. The second model is a two-parameter turbulence model \( k - \omega \) SST (Shear Stress Transport) [11]. The third model is a two-parameter model realizable \( k - \varepsilon \) with enhanced wall treatment [12]. And the fourth model is the SST transition model [13].

The working medium was air and the ideal gas law was used to determine the fluid density. For determination of fluid viscosity Sutherland's law with three default coefficients was used. For all numerical simulations, the solver type and the algorithm of pressure-velocity coupling were chosen pressure-based and coupled respectively. Such an algorithm is suitable for solving gas dynamics problems in the range of Mach number variation from zero to 2 - 3.

Second-order upwind discretization was used to obtain accurate solutions. Since such a solution is unstable at the first iterations, therefore, the first several tens of iterations were carried out using the first order of discretization, then second-order discretization was used.

The solution was considered complete if the following conditions were met:
- If the values of residuals of continuity and other variables were less than then \( 10^{-3} \) and the residual of the energy equation is lower than \( 10^{-6} \).
- The difference between the flow rates of the working medium between the inlet and outlet sections is no more than 1% of the total flow rate.

2.2. Investigated cascade and boundary conditions

In this paper, a turbine stator cascade C9012A was considered. The blade coordinates can be found in [1]. The main geometric parameters of the cascade are summarized in table 1. As shown in figure 1, the domain includes 1 nozzle guide vane, inlet zone located at a 1 axial chord length upstream of the leading edge, and outlet located at a 1 axial chord length from the trailing edge. All blade walls are set with the no-slip condition and are adiabatic. The top and bottom borders along the cascade were specified as periodic boundaries. At the inlet boundary, total pressure, total temperature, and turbulence parameters are given, and at the outlet boundary, the static pressure was set. Table 2 presents the boundary conditions.
Table 1. The main geometric parameters of the cascade.

| Parameter                  | Value | Unit   |
|----------------------------|-------|--------|
| Chord                      | 50    | mm     |
| Axial chord                | 26.58 | mm     |
| Installation angle         | 33.2  | degree |
| Relative spacing           | 0.75  | -      |
| Leading-edge radius        | 3     | mm     |
| Trailing edge radius       | 0.25  | mm     |
| Incidence angle            | 0     | degree |
| The effective flow exit angle | 12   | degree |

Figure 1. Scheme of the computational domain

2.3. Computational grid
The reliability of the results of numerical modeling highly depends on the quality of the constructed mesh. A structured quadrilateral mesh was created with the commercial grid generator ICEM CFD using a multi-block meshing technique. Around the blade surfaces, O-grid was employed to get a highly orthogonal grid for the most accurate simulation of the boundary layer. As low Reynolds formulation of turbulence models were used to accurately capture flow features in the entire boundary layer including viscous sublayer and logarithmic layer, the $y^+$ value for all numerical simulations was approximately equal to 1, and there were about 28 cells in the boundary layer. To achieve this, a dimensional height of $y_1 = 1.49 \mu m$ is employed for the first cells at solid walls. Four mesh densities were created to check the grid independent of numerical results. Numerical results of the profile loss coefficient were compared for these four mesh densities using the SST transition model. It showed that the difference between the numerical results of loss coefficients for the finest and coarsest mesh is less than 0.17%. In figure 2 the final version of the computational grid, consisting of 42108 cells is presented.
Table 2. Boundary conditions.

| Parameter                              | Value  | Unit  |
|----------------------------------------|--------|-------|
| Inlet total pressure, $P_{01}$         | 152    | kPa   |
| Inlet total temperature, $T_{01}$      | 436    | K     |
| Outlet isentropic Mach number, $M_{2is}$| (0.22 – 0.8) | - |
| Outlet Reynolds number, $Re_2$         | (2.2 – 7.75) $10^5$ | - |
| Inlet turbulent intensity, $Tu$        | 3      | %     |
| Turbulent length scale, $l$            | 2.625  | mm    |

Figure 2. The final computational grid used for numerical simulations
Figure 3. The profile loss coefficient for various exit isentropic Mach number.

Figure 4. The Mach number contours for $M_{2is} = 0.5$. 
3. Results and Discussion

Figure 3 shows the experimental and computational profile loss coefficients for various exit Mach numbers. The computational results using the SST transition model were in good agreement with the experimental data. As the reason for more discrepancy of other models, it can be noted that the loss of kinetic energy in fully turbulent flows is greater than in transitional flows.

In figures 4 and 5 are shown the Mach number and static pressure contours for different turbulence models at $M_{2is} = 0.5$ respectively. It is seen that velocity and pressure distributions qualitatively and quantitatively are similar in the cases of using realizable $k - \varepsilon$, $k - \omega$ SST, and SST transition models. However, the one-parameter Spalart-Allmaras model cannot be recommended for determining the flow characteristics in a nozzle cascade with a turbulent flow regime.

Figure 6 shows the contours of the turbulent kinetic energy (TKE) obtained by modeling the fully turbulent (realizable $k - \varepsilon$, $k - \omega$ SST) and transitional (SST transition model) flows. It can be seen that the TKE growth begins in the middle of the blade suction surface when using realizable $k - \varepsilon$ and $k - \omega$ SST (figure 6, a and b). But when considering the transitional flow, the TKE growth occurs approximately at the trailing edge zone (figure 6, c). Therefore, when simulating the flow through the turbine cascade using the SST transition model, the flow over most of the airfoil surfaces remains almost laminar. Since the kinetic energy loss in laminar flows is less than turbulent flows, consequently, the error of computational results using the transition SST model is less than when using other turbulence models.

![Figure 5. Static pressure contours for $M_{2is} = 0.5$](image-url)
4. Conclusion
The numerical simulation of the flow field in a turbine stator cascade was carried out. The effect of four turbulence models on the flow was investigated.

The computational results of profile loss coefficients have shown that the SST transition model produces reasonable agreement with experimental data. The results of Mach number and static pressure contours for three models of realizable $k-\varepsilon$, $k-\omega$ SST, and SST transition were similar. However, the Spalart-Allmaras model cannot be recommended for determining the flow characteristics in a turbine cascade with a turbulent flow regime.

References
[1] Deich M E 1996 Gas Dynamics of Turbomachinery Cascades (Moscow: Energoatomizdat)
[2] Zaryankin A E 2019 Mechanical of Incompressible and Compressible Fluids textbook for universities (Moscow: MPEI Publishing House 592 p)
[3] Denton J 2010 Some limitations of turbomachinery CFD ASME Turbo Expo GT2010-22540
[4] Menter F R, Langtry R and Hansen T 2004 CFD simulation of turbomachinery flows – verification, validation, and modeling Proceeding of ECCOMAS
[5] Guo L, Yan Y and Maltson J D 2012 Performance of 2D scheme and different models in predicting flow turbulence and heat transfer through a supersonic turbine nozzle cascade International journal of heat and mass transfer 55 6757-6765
[6] El-Gendi M M, Lee K U, Chung W J, Joh C Y, and Son C H 2012 Evaluation of turbulence models in a high-pressure turbine cascade simulation J. Comput. Fluids Eng. 17 53-58
[7] Sorour M M, Teamah M A and Abbas A M, 2010 Evaluation of different turbulence models and numerical solvers for a transonic turbine blade cascade Proceedings of ICFD 10
[8] Vlahostergios Z 2018 Performance assessment of Reynolds stress and eddy viscosity models on a transitional DCA compressor blade J. Aerosp. 5(4) 120
[9] Zhang H, Wu Y and Li Y 2015 Evaluation of RANS turbulence models in simulating the corner separation of a high-speed compressor cascade J. Eng. Appl. Comput. Fluid Mech. 9(1) 477-489
[10] Spalart P R and Allmaras S R 1994 A one-equation turbulence model for aerodynamics flows *La Recherche Aerospatial* **1** 5-21

[11] Mentler F R 1994 two-equation eddy-viscosity turbulence models for engineering applications *AIAA Journal* **32** 1598-1605

[12] Kader B 1981 temperature and concentration profiles in fully turbulent boundary layers *Int. J. Heat and Mass Transfer* **24** 1541-1544

[13] Mentler F R, Langtry R and Volker S 2006 Transition modeling for general purpose CFD codes *Flow Turbulence and Combustion* **77** 277-303

[14] Lazarev L Y, Stepanova T N, Ryakhovskaya N V and Fadeev V A 2004 *Geometric and Energy Characteristics of the Profiles of Turbine Blades of Constant Cross-section* (Moscow: MPEI Publishing House 51 p)