Evaluation of RANS turbulence models in simulating the corner separation of a high-speed compressor cascade

Haideng Zhang\textsuperscript{a,b}, Yun Wu\textsuperscript{a*} and Yinghong Li\textsuperscript{a}

\textsuperscript{a}College of Aeronautical and Aerospace Engineering, Airforce Engineering University, Xi’an, China; \textsuperscript{b}College of Energy and Propulsion Engineering, Beihang University, Beijing, China

(Received 2 October 2014; final version received 2 July 2015)

A typical high-speed compressor cascade flow is simulated using an eddy viscosity model and a Reynolds stress model. Based on comparisons with experimental results, the simulation accuracies of the two turbulence models are evaluated, and some recommendations are provided for further research. Inside the cascade flow passage, the simulation accuracies of both turbulence models decrease because of the corner effect. Downstream of the blade, the wake mixing influence, and thus the flow loss, is under-predicted by both turbulence models. Compared with the linear eddy viscosity model, the modification of the simulation accuracy with regard to endwall flow prediction using the Reynolds stress model is limited, which only occurs at a highly positive incidence angle. The corner separation strength and wake mixing influence predicted by the eddy viscosity model are surpassed by those of the Reynolds stress model. At a negative incidence angle, the simulation result of the Reynolds stress model is proved to be physically unreasonable according to the flow loss prediction.

\textbf{Keywords:} turbulence model; compressor cascade; corner separation; endwall flow; secondary flow

1. Introduction

High axial compressor performance and operability are key factors and considerations in the design process of high-performance aero-engines. Experimental testing of three-dimensional turbulent flows in high-speed and highly-loaded axial compressors is rather expensive (Place, 1997). Therefore, the reliance of axial compressor design processes and research on computational fluid dynamics (CFD) is ever increasing. As noted by Denton (2010) and Menter, Langtry, and Hansen (2004), the reliable prediction of turbulence is critical for turbomachinery aerothermal design. The state of the art for the development and challenges of turbulent flow CFD in turbomachinery is summarized by Tucker (2013). Although the Reynolds-averaged Navier-Stokes (RANS) method has proven to be rather insufficient in modeling complex turbulent flow in turbomachinery, its future in engineering projects of turbomachinery aerodynamics is still promising because of its robustness and relatively low computational resource consumption (Tucker, 2011, 2013).

In axial compressor stator blade flow passages, cross flow on the shroud/hub gives rise to the accumulation of low-energy fluid at conjunctions between the blade suction surface and shroud/hub endwall, which causes a three-dimensional corner separation or even corner stall under the influence of adverse pressure gradients (Dong, Gallimore, & Hodson, 1987; Joslyn & Dring, 1985; Lei, Spakovszky, & Greitzer, 2008; Schulz, Gallus, & Lakshminarayana, 1990). Because of its dramatic impact on compressor performance, corner separation has been studied both experimentally and numerically with the progress of axial compressor design techniques using compressor cascade models (Chen, Qiao, Liesner, & Meyer, 2014; Evans, Hodson, Hynes, & Wakelam, 2010; Gbadebo, Cumpsty, & Hynes, 2005, 2008; Zachos, Grech, Charnley, Pachidis, & Singh, 2011).

With the enormous development in computer science, the large eddy simulation (LES) method has been used in compressor cascade flow simulation in recent years, utilizing a large amount of computer resources (Matsuura & Kato, 2007; McMullan & Page, 2011; Teramoto, 2005; Zaki, Wissink, Durbin, & Rodi, 2009). Modeling three-dimensional corner separation requires the CFD code to be very robust. Therefore, most of the LES work is typically focused on the two-dimensional flow characteristics in compressor cascade flow passages, and the relatively more influential three-dimensional corner separation has been neglected. Recently, the three-dimensional flow structures in a compressor cascade were simulated using the detached eddy simulation (DES) method (Z. Wang & Yuan, 2013), but the blade airfoil NACA-0009 studied is lightly loaded, which is not representative of modern highly loaded axial compressors. Therefore, the RANS method is widely used in detailed numerical work on
three-dimensional corner separation. In order to improve the simulation accuracy of the RANS method in corner separation modeling, modification of current RANS models was performed (Liu, Lu, Fang, & Gao, 2011; D. Wang, Lu, & Li, 2009). This, in turn, extended the application of RANS methods in the research on axial compressor corner separations.

Most of the past studies on compressor cascade corner separation were performed in the low-speed regime (with inflow Mach numbers smaller than 0.3) to reveal the flow separation mechanisms and determine an efficient control method to suppress the flow separation. However, it is not always easy to transfer these research results to engineering applications of high-speed flow (Tiedemann, Heinrich, & Peitsch, 2012). CFD application to a high-speed compressor cascade is more difficult because of the more severe corner separations and fully developed flow unsteadiness. Even so, studies on high-speed compressor cascades have greatly benefited from the robust RANS methods (Chen et al., 2014; Gmelin, Thiele, & Liesner, 2011; Hergt, Meyer, & Engel, 2013; Hergt, Meyer, Liesner, & Nicke, 2011; Weber, Schreiber, Fuchs, & Steinert, 2002). The turbulence models applied in the previous high-speed compressor cascade studies were carefully selected eddy viscosity models, which can easily diverge. However, by neglecting the anisotropic nature of the turbulence, the linear eddy hypothesis produces apparent deviations between simulations and experiments. Hence, to investigate high-speed compressor cascade flows, improving the simulation accuracy of the eddy viscosity model is necessary and meaningful. The efforts of D. Wang et al. (2009) and Liu et al. (2011) provided promising guidelines for modifications to eddy viscosity models to better simulate three-dimensional corner separation. However, the present encouraging results are obtained in the low-flow regime and constrained to the one-equation Spalart-Allmaras model. Non-linear eddy viscosity models have also been used, with better simulation accuracy (Balabel & El-Askary, 2011; Craft, Launer, & Suga, 1996; Gatski & Speziale, 1993; Kulisa & Dano, 2006; Mani, Babcock, Winkler, & Spalart, 2013; Mehdizadeh, Temmerman, Tartiville, & Hirsch, 2012). However, the shortcomings of non-linear eddy viscosity models are also unavoidable (Tucker, 2013). Through a careful numerical investigation, it is concluded that a linear turbulence model could provide good results in a simple jet flow, whereas in a complicated jet flow, a non-linear model would be more favorable (Balabel & El-Askary, 2011). In contrast, compared with linear turbulence models, it has also been pointed out that no significant modifications are observed when using the non-linear model to simulate a turbine flow (Kulisa et al., 2006). Thus, it is indeed necessary to use the stress-strain relationship provided by a full Reynolds stress model to obtain accurate predictions. Although the Reynolds stress model is always expected to give improved accuracy, this is not always the case (Tucker, 2013). To achieve closure of equations, the Reynolds stress model, in practice, contains substantial empiricism. Therefore, to apply a Reynolds stress model to high-speed compressor cascade CFD research, careful calibrations with experiments must be applied first.

As previously discussed, to study the corner separation of axial compressors using RANS turbulence models, as a first step, it is essential to evaluate the accuracy of RANS turbulence models in simulating three-dimensional corner separation, especially in the high-speed flow regime. However, because experimental tests of corner separations in the high-speed flow regime are very expensive, researchers are seldom able to perform such evaluations. Therefore, few references are available in this research field.

In this paper, an eddy viscosity model is applied to simulate a high-speed compressor cascade flow. The accuracy of the eddy viscosity model is discussed in detail. Because of the unavoidable deviations between the simulation results of the eddy viscosity model and the results of experiments, a Reynolds stress model is introduced with the expectation of higher simulation accuracy. A better understanding of the advantages and disadvantages of the eddy viscosity model and Reynolds stress model in simulating three-dimensional corner separation is obtained, which is useful for promoting axial compressor corner separation CFD research, especially in the high-speed flow regime.

2. RANS equations

With the introduction of averaged and fluctuating quantities in the original unsteady Navier-Stokes equations, the RANS equations are produced as shown in Equation (1):

\[ \frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}(\tau_{ij} - \rho \frac{\partial U_i \vec{U}}{\partial x_j}) + S_M \]

(1)

Compared to the direct numerical simulation (DNS) method, computational resources are greatly reduced by the simulation of RANS equations. Therefore, RANS equations have been widely adopted for practical engineering applications. However, the averaging procedure introduces additional unknown terms \((\rho \bar{u} \bar{u})\), which contains products of the fluctuating quantities \((\vec{u})\). The unknown terms \(\rho \bar{u} \bar{u}\), referred to as Reynolds stress, act like additional stresses in the fluid and are difficult to determine directly. So additional equations are necessary to model the Reynolds stress with known quantities in order to achieve “closure” and the turbulence model type is defined by the equations used to close the system.

2.1. Eddy viscosity model

The eddy viscosity model is based on the hypothesis that the Reynolds stress can be related to the mean velocity gradients and eddy viscosity by the gradient diffusion
hypothesis, in a manner similar to the relationship between stress and strain tensors in laminar Newtonian flow:

\[-\rho \bar{u}_i \bar{u}_j = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left( \rho k + \mu_t \frac{\partial U_k}{\partial x_k} \right)\]  

(2)

where \( \mu_t \) is the turbulent viscosity and \( k \) is the kinetic energy.

Inspired by previous work on high-speed compressor cascades (Chen et al., 2014; Hergt et al., 2011; Hergt, Meyer, & Engel, 2006), the Wilcox k-omega model (Wilcox, 1988) is applied in this paper. The turbulent viscosity \( \mu_t \) is linked to the turbulence kinetic energy \( k \) and turbulent frequency \( \omega \) via the relation

\[ \mu_t = \rho \frac{k}{\omega} \]  

(3)

For the two-equation Wilcox k-omega model, transport equations for turbulent kinetic energy \( k \) and turbulent frequency \( \omega \) are as follows:

\[ \frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_i k)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \]

\[ + \rho P_k - \beta' \rho k \omega \]  

(4)

\[ \frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho U_i \omega)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \omega}{\partial x_j} \right] \]

\[ + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 \]  

(5)

In Equation (5), \( P_k \) is the turbulence production due to viscous forces, and is modeled with Equation (6):

\[ P_k = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \left( \frac{3}{2} \mu_t \frac{\partial U_i}{\partial x_k} + \rho k \right) \]  

(6)

For a special simulating case, definition of the modeled constants should be based on rich experience. In this paper, having referred to the advice of commonly used CFD software, the modeled constants applied are set as below:

| Constants | \( \beta' \) | \( \alpha \) | \( \beta \) | \( \sigma_k \) | \( \sigma_\omega \) |
|-----------|-------------|-------------|-------------|-------------|-------------|
| Value     | 0.09        | 5/9         | 0.075       | 2           | 2           |

2.2. Reynolds stress model

Unlike the eddy viscosity model, the Reynolds stress model (Lauder, Reece, & Rodi, 1975; Speziale, Sarkar, & Gatski, 1991) is based on transport equations for all components of the Reynolds stress tensor and the dissipation rate. A separate transport equation must be solved for each of the six Reynolds stress components \( \rho \bar{u}_i \bar{u}_j \).

The omega-based Reynolds stress model is applied in this paper and the modeled equations for Reynolds stress can be written as follows:

\[ \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial t} + \frac{\partial (\rho U_i \bar{u}_j)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_2} \right) \frac{\partial \bar{u}_i}{\partial x_j} \right] \]

\[ + \rho P_{ij} - \beta' \rho \bar{u}_i \omega_j + \Phi_{ij} + \frac{\partial}{\partial x_k} \left( \mu + \mu_t \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_k} \right) \]  

(7)

3. Experimental procedure

The typical high-speed cascade NACA65-K48 (Chen et al., 2014; Hergt et al., 2006, 2011, 2013) was studied. The design Mach number is 0.67, with a design Reynolds number of \( 4.5 \times 10^5 \) based on the blade chord length, and the pitch to chord ratio is 0.55. Its main geometry parameters are shown in Table 1.

The experimental results in this paper were obtained in the Dalian Maritime University high-speed linear cascade wind tunnel. The highest inlet Mach number reaches about 1.1, thus being qualified for high subsonic compressor cascade experiments. During the experiment, the inlet Mach number was set as 0.7 and the blades were confined to a rotary plate. Through rotating the plate, the inlet flow incidence can be changed continuously. To keep periodicity, seven blades were utilized in the experiment. Through measuring the transverse distribution of the total pressure loss coefficient downstream of the central three blades, periodicity of the flow field is testified (as shown in Figure 1).

The measurement system employed in the experiments consisted of dynamic probes, a high-frequency pressure transducer, a high-accuracy pressure scanning module (DAS3217) and a high-accuracy thermocouple temperature measurement module (DTS 3250).

4. Numerical method

To reduce the numerical errors, a second-order accurate approximation was applied in the discretion processes, and the robustness was ensured using high-quality calculation
grids. The time step in the simulation was set as 5e-5s, with which fluid is expected to flow pass the cascade blade in about 5 steps.

The calculation grid model is shown in Figure 2. With a symmetrical boundary at mid-span, the cascade flow passage of half span was modeled to reduce the computation cost. The grid element number was varied to test the corresponding influences on the simulation results in preparation for the formal work of this paper. Finally, a grid model of 1.5 million was chosen as optimal, with high numerical accuracy being the first optimizing goal and low computational cost being the second. Near solid walls, grids were refined and the dimensionless variable $y^+$ is restricted under 1, as the low-Re treatment is applied for near wall turbulence.

The inlet boundary was set 1.2 chord-lengths upstream of the blade leading edge, and the outlet boundary was placed 2 chord-lengths downstream of the blade trailing edge. At the inlet, total pressure distributions in the span-wise direction together with a fixed total temperature from experiments were imposed. For the outlet, an average static pressure boundary was imposed. During the simulations, solid walls were assumed to be adiabatic and non-slip.

5. Endwall flow

The static pressure coefficient $C_p$ is defined as

$$C_p = \frac{p_{2s} - p_{1s}}{p_1 - p_{1s}},$$

where $p_1$ and $p_{1s}$ are the total pressure and static pressure of inlet flow, and $p_{2s}$ is the local static pressure. The distribution of the static pressure measurement points in the experiments is shown in Figure 3. Experimentally obtained and simulated static pressure distributions from locations L1 to L12 are compared in Figure 4 to evaluate the simulation accuracy of the turbulence models.

According to Figure 4, in both the streamwise and pitchwise directions, the general variance law of the endwall static pressure is well predicted by Reynolds-averaged turbulence models. Along the streamwise direction, the static pressure increases first and then decreases. Along the pitchwise direction, the static pressure increases from the suction side to the pressure side. Near the blade suction surface, the onset of corner separation is denoted by the constant static pressure distribution along the pitchwise direction in the separation region. During the experimental process, measurement points were located several millimeters away from the blade suction surface, and the onset of corner separation was not observed. For the design and positive incidence angles, when corner separation is fully developed, constant static pressure distribution regions were measured by experiments at locations L8, L7 and L6 respectively, as shown in Figures 4(a)–4(c). Based on comparisons between simulated and experimentally obtained
Figure 4. Endwall static pressure distributions.
static pressure variance laws along the pitchwise direction at these locations, the onset of corner separation is inferred to be reasonably predicted by the Reynolds-averaged turbulence models. For a negative incidence angle, as shown in Figure 4(d), the corner separation is constrained to a small scale and was not measured in the experiments.

Inside the cascade flow passage, a great challenge to the accuracy of turbulence model simulations is imposed by the corner effect, as illustrated in Figure 4(a). At conjunctions between the blade surfaces and endwall, the turbulence anisotropy, non-equilibrium, and energy backscattering are enhanced by the transport and accumulation of the boundary layer, as well as the emergence of corner separation. Thus, both turbulence models fail to predict the fluid state variance properly, as shown by the results at locations from L1 to L10. Compared with the results of experiments, the most severe deviations of the simulated static pressure distributions around the corner are observed at the design incidence angle, as shown in Figure 4(a). However, according to Figures 4(b)–4(d), at both positive and negative incidence angles, the influences of the corner effect are weakened, as indicated by smaller deviations between the turbulence model predictions and experiments, especially for the highly positive incidence angle. Near the blade trailing edge, at locations L11 and L12, the corner effects are almost wiped out, and the static pressure distributions in the experiments are well predicted by the eddy viscosity model, except at the highly positive incidence angle condition, as shown in Figure 4(c). A limited improvement in the endwall flow prediction accuracy is achieved by the Reynolds stress model, although obvious differences between the simulation results with the eddy viscosity model and the Reynolds stress model are observed at a negative incidence angle, as shown in Figure 4(d), and a highly positive incidence angle, as shown in Figure 4(c).

At the design incidence angle, as shown in Figure 4(a), the simulation results with the eddy viscosity model and the Reynolds stress model are almost coincident. Both models behave poorly at the corner inside the cascade flow passage and provide rather satisfying predictions at locations L11 and L12 near the blade trailing edge.

As the incidence angle increases and becomes positive, differences between the simulation results for both turbulence models appear. For a moderately positive incidence angle (shown in Figure 4(b)), differences are mostly observed near the blade trailing edge. Under this circumstance, solving the Reynolds stress transport equations is inferior to choosing the eddy viscosity hypothesis, because according to Figure 4(b), at locations L11 and L12, the static pressure distributions are better predicted by the eddy viscosity model. According to Figure 4(c), at a highly positive incidence angle, the corner effect greatly diminishes inside the cascade flow passage, and obvious differences between simulation results for the two turbulence models are found at both the front part and outside of the cascade flow passage at L12. At the front part, the simulation results of the Reynolds stress model are a little closer to the experimental results, but not apparently so. Outside the cascade flow passage at L12, when the flow is totally free of the blade restriction, the static pressure is more accurately predicted by the Reynolds stress model.

From Figure 4(d), at a negative incidence angle, the static pressure predicted by Reynolds stress model is smaller than that of the eddy viscosity model. At locations L1, L2, L3, L4, L5, L6, and L7, the simulation results of the Reynolds stress model are closer to the results of the experiments. In contrast, at locations L9, L10, L11 and L12, the simulation results of the Reynolds stress model become much smaller than the experimental results, which agree with the simulation results of the eddy viscosity model.

6. Downstream flow loss

To evaluate the flow loss, the total pressure loss coefficient $\zeta$ is defined as

$$\zeta = \frac{p_1 - p_2}{p_1 - p_{1s}},$$

where $p_2$ is the local total pressure.

To better evaluate the flow loss at different blade heights, the pitch-averaged total pressure loss coefficient $\zeta_t$ is defined as

$$\zeta_t = \frac{1}{n} \sum_{i=1}^{n} \left( \zeta_i \cdot \Delta t_i \right).$$

Figure 5 shows comparisons between the flow loss predictions by the turbulence models and the experimental results. Data were obtained at 60% of the chord length downstream of the blade trailing edge. The variance law of flow loss along the spanwise direction predicted by the turbulence models agrees very well with the experimental results, which accounts for the popularity of turbulence models in engineering applications.

In order to evaluate the simulation accuracy of the two turbulence models, the cascade flow passage is divided into three zones along the spanwise direction – the endwall zone (E-zone), the mid-span zone (M-zone) and the intermediary zone (I-zone), between the E-zone and M-zone.

At the design and positive incidence angles, as shown in Figures 5(a)–5(c), in the E-zone, the increase and decrease process of the flow loss along the spanwise direction is well predicted by both turbulence models. However, the flow loss is obviously under-predicted by the eddy viscosity model while the Reynolds stress model provides flow loss predictions with a higher accuracy, especially for the regions closest to the endwall. At a negative incidence angle, as shown in Figure 5(d), in the E-zone, the flow loss predictions by the Reynolds stress model
are not physically reasonable because of the sudden flow loss increase phenomenon. Although the flow loss is still under-predicted by the eddy viscosity model, the decrease process of flow loss along the spanwise direction predicted agrees very well with the results of experiment.

As shown in Figure 5(a), at the design incidence angle, the flow loss is under-predicted by both turbulence models in the I-zone. According to Figure 5(b), at a moderately positive incidence angle, flow loss over-prediction by the Reynolds stress model is partly observed in the I-zone. For a highly positive incidence angle, as shown in Figure 5(c), the I-zone flow loss is mostly over-predicted by the Reynolds stress model and partly over-predicted by the eddy viscosity model. However, the process whereby the flow loss first increases and then decreases in the spanwise direction is again well predicted by both turbulence models, as shown in Figures 5(a)–5(c). Similar to the E-zone, at a negative incidence angle, the simulation results of the eddy viscosity model in the I-zone are closer to the experimental results compared with those of the Reynolds stress model.

In the M-zone, as shown in Figures 5(a) and 5(b), at the design and moderately positive incidence angles, the flow loss predicted by both turbulence models agrees very well with the experimental results. At a highly positive incidence angle, as shown in Figure 5(c), the M-zone flow loss is greatly over-predicted by the Reynolds stress model but properly predicted by the eddy viscosity model. At a negative incidence angle, as shown in Figure 5(d), the flow loss is obviously over-predicted by both turbulence models, but the eddy viscosity model is slightly more accurate.

The M-zone flow is almost undisturbed by the three-dimensional corner separation, and mainly behaves in a two-dimensional way. Thus, the anisotropic nature of the turbulence is not dominant, which results in the relatively high prediction accuracy for both turbulence models, except for negative and highly positive incidence angle conditions, where the two-dimensional separation is further deteriorated. For both the I-zone and E-zone, flow mixing and three-dimensional corner separation coexist. Different scales of vortexes and turbulence structures give rise to prominent anisotropy, non-equilibrium, and energy
backscattering in flow interactions. Under this circumstance, both turbulence models fail to predict the flow structures accurately, and the flow loss under-prediction at negative, design, and moderately positive incidence angles, as well as the flow loss over-prediction at highly positive incidence angles, was also observed in previous studies (Kožulovic & Röber, 2006; Lewin, Kožulovic, & Stark, 2010). Through solving the non-linear Reynolds stress transport equations, flow loss prediction accuracy is marginally enhanced by the Reynolds stress model at design and positive incidence angles and even reduced at negative incidence angles. Therefore, the gains are far from satisfying in consideration of the triple computational cost compared to the eddy viscosity model.

7. Secondary flow
7.1. Secondary flow at the design incidence angle
The secondary flow structures at the design incidence angle obtained from the experiments and the simulations are shown in Figures 6 and 7, respectively. According to Figures 6(a) and 7(a), inside the cascade flow passage, at the 70% chord length transverse plane, the passage vortex (PV) first originates from the E-zone, which is observed in the experiment as well as predicted by both turbulence models. Because of geometric restrictions from the blades in the cascade flow passage, the region close to the blade surfaces is not measured in the experiment. As shown in Figure 6(b), at the 90% chord length plane, in contrast with the simulation results, shown in Figure 7(b), additional topological structures (A-TSs) associated with blade suction side separation are observed in the experiment, and the small-scale corner separation vortex (CV) is only observed in the simulation.

As shown in Figures 6(c), 6(d) and 7(c), downstream of the blade, wake mixing of the pressure side and suction side flow interacts with the passage vortex and makes the secondary flow structures more complicated. Compared with the simulation results, the influences of wake mixing (WM-I) on the secondary flow structures are more prominent in the experiments. At the 140% chord length transverse plane, as shown in Figure 6(d), topological structures corresponding to wake mixing are still observable in the experiment. In addition, further downstream at the 160% chord length transverse plane, as shown in Figure 6(e), only the passage vortex is measureable, which testifies to the almost complete dissipation of wake mixing influences. However, according to the simulation results, the obvious influences of wake mixing are only visible at the 110% chord length transverse plane (shown in Figure 7(c)) for both turbulence models, and the topological structures corresponding to passage vortex and wake mixing interactions are also different from those in the experiments. The fast dissipation of wake mixing during the simulation processes results in the rapid disappearance of its influence on the flow field. As shown in Figure 7(d), at the 120% chord length transverse plane, wake mixing is predicted to dissipate almost completely by both turbulence models. Thus, downstream of the cascade blade, the differences between the simulated and experimentally measured secondary flow structures are dramatic in relation to the wake mixing factor. At the 160% chord length transverse plane (shown in Figures 6(e) and 7(f)), where the wake mixing influences are also wiped out in the experiments, the secondary flow structures predicted by the turbulence models agree well with those in the experiments, just like inside the cascade flow passage.

Although turbulence models are capable of predicting the existence of main corner separation vortexes, predicting the interactions between different flow structures
Figure 7. Simulated secondary flow structure at design incidence angle.
is out of their reach. Under the design incidence angle condition, the corner separation and wake mixing influences are both under-predicted by the eddy viscosity model. As a result, the flow losses of both the E-zone and I-zone are also under-predicted. Near the endwall, the Reynolds stress model behaves slightly better in accurately predicting the flow loss, which is in accordance with the more properly predicted corner separation strength. However, the non-linear characteristics of turbulence are greatly enhanced by the introduction of wake mixing, which imposes some unmet challenges on the empirically established Reynolds stress transportation equations. For this reason, the Reynolds stress model behaves poorly in predicting the flow loss at the edge of the E-zone and in the I-zone, where wake mixing interacts severely with the passage vortex.

7.2. Secondary flow at off-design incidence angles
Secondary flow structures at the 160% chord length transverse plane from the experiments and simulations at off-design incidence angles are shown in Figures 8–10. From Figure 8, with an incidence angle of 4°, the wake mixing influence is still under-predicted by both turbulence models. At the 160% chord length transverse plane, where the influence of wake mixing is still obvious according to the experimental results, only the existence of corner vortices is predicted by both the eddy viscosity model and the Reynolds stress model. Thus, as shown in Figure 5(b), the flow loss in the I-zone is still mostly under-predicted. On the other hand, apart from the passage vortex, the corner separation vortex is only found at the 160% chord length transverse plane in the simulation results for the Reynolds stress model, which indicates a stronger corner separation compared with the simulation results of the eddy viscosity model and the experimental results. For this reason, the flow loss in the I-zone is partly over-predicted by the Reynolds stress model, as shown in Figure 5(b).

For highly positive incidence angles, the corner separation strength predicted by the turbulence models is greatly enhanced, especially for the Reynolds stress model (shown in Figure 9) and, as shown in Figure 5(c), over-prediction...
of the flow loss is thus observed. However, according to Figure 9, the wake mixing influence is still dramatically under-predicted by the eddy viscosity model, which is in accordance with the large-scale flow loss under-prediction by the eddy viscosity model in Figure 5(c). In contrast, wake mixing plays a more important role in the secondary flow structure shown by the simulation results of the Reynolds stress model, and as a result, the under-prediction of the flow loss in Figure 5(c) is very minor. As shown in Figure 10, for negative incidence angles, the wake mixing influence is wiped out both in the experiment and the simulations, and only the passage vortex can be observed at the 160% chord length transverse plane. However, according to Figure 5(d), the simulation results of the eddy viscosity model are more physically reasonable.

8. Conclusions
The main characteristics of endwall flow are well predicted by both the eddy viscosity model and the Reynolds stress model. Inside the cascade flow passage, the corner effect is aroused at solid surface conjunctions, and the simulation accuracies of the turbulence models are greatly reduced. Compared with the design incidence angle, at negative or positive incidence angles, the corner effect is weakened and the simulation accuracy of the turbulence models increases in terms of predicting the endwall flow. However, in spite of the corner effect, the onset of corner separation inside the cascade flow passage is still reasonably predicted by both turbulence models. Outside the cascade flow passage and with the absence of the corner effect, a relatively high prediction accuracy for the endwall flow is achieved by the turbulence models at the design and moderately positive incidence angles. At negative or highly positive incidence angles, there are still apparent deviations between the simulation and experimental results. In a test involving the solution of the non-linear Reynolds stress transport equations, the simulation accuracy in predicting the endwall flow is marginally improved by the Reynolds stress model, but only at highly positive incidence angles.

Downstream of the blade, at the design and positive incidence angles, the wake mixing influence is mostly under-predicted by the turbulence models. As a result, under-prediction of the flow loss is present. With an increase in the incidence angle, the corner separation strength predicted by the turbulence models is increased, and some over-prediction of the flow loss is observed, in spite of the wake mixing influence under-prediction. Generally, at the design and positive incidence angles, the corner separation strength and wake mixing influence predicted by the eddy viscosity model are surpassed by those of the Reynolds stress model. When the incidence angle is increased to be highly positive, obvious over-prediction of flow loss by the Reynolds stress model is observed, while the flow loss is still mostly under-predicted by the eddy viscosity model. At negative incidence angles, the wake mixing influence is reduced and the secondary flow structure is more properly predicted by the Reynolds-averaged turbulence model. However, the simulation results of the Reynolds stress model are proved to be physically unreasonable according to the flow loss prediction.

9. Recommendation and future work
In this paper, to evaluate the accuracy of RANS turbulence models in simulating the corner separation of a high-speed compressor cascade, the corresponding simulation and experimental results were analyzed and compared in detail.

Based on the conclusion of this paper, the following suggestions are provided to researchers carrying out similar projects:

(1) Commonly used RANS methods can be treated as useful tools in simulating the main structures of high-speed compressor cascade corner separation, which are rather unstable.
(2) To simulate a high-speed compressor cascade flow, special attention should be paid to the corner effect inside the cascade passage and wake mixing downstream of the blades.

(3) The Reynolds stress model is not superior to the eddy viscosity model in simulating high-speed compressor cascade corner separations of moderate strength. In simulating high-speed compressor cascade flows under near-stall conditions, the Reynolds stress model may be an alternative choice, but the simulation results should be carefully evaluated in order to avoid physically unreasonable results.

However, it should be mentioned that the experimental methods discussed in this paper are incapable of capturing small-scale and transient high-speed compressor cascade flow structures. Thus, in the future, in order to perform finer evaluations of the simulation accuracy, more advanced experimental methods such as particle image velocimetry (PIV) should be applied. On the other hand, many factors that influence the simulation accuracy, such as the numerical scheme, should be considered in future studies – and with the development of computer technology, some LES simulation work will be quite helpful in studying three-dimensional corner separations in the high-speed regime.

Acknowledgements

The authors thank Prof. Meyer of DLR for the airfoil data.

Disclosure statement

No potential conflict of interest was reported by the authors.

Funding

This work was supported by the National Natural Science Foundation of China [51336011, 50906100], the Science Foundation for the Author of National Excellent Doctoral Dissertation of China, and China Scholarship Council.

References

Balabel, A., & El-Askary, W. A. (2011). On the performance of linear and nonlinear k-ε turbulence models in various jet flow applications. *European Journal of Mechanics - B/Fluids*, 30, 325–340.

Chen, P. P., Qiao, W. Y., Liesner, K., & Meyer, R. (2014). Location effect of boundary layer suction on compressor hub-corner separation (ASME Paper No. GT 2014-25043).

Craft, T. J., Launder, B. E., & Suga, K. (1996). Development and application of a cubic eddy-viscosity model of turbulence. *International Journal of Heat and Fluid Flow*, 17(2), 108–115.

Denton, J. (2010). *Some limitations of turbomachinery CFD* (ASME Paper No. GT2010-22540).

Dong, Y., Gallimore, S. J., & Hodson, H. P. (1987). Three-dimensional flows and loss reduction in axial compressors. *Journal of Turbomachinery*, 109, 354–361.

Evans, S., Hodson, H., Hynes, T. P., & Wakelam, C. (2010). Flow control in a compressor cascade at high incidence. *Journal of Propulsion and Power*, 26(4), 828–836.

Gatski, T. B., & Speziale, C. G. (1993). On explicit algebraic stress models for complex turbulent flows. *Journal of Fluid Mechanics*, 234(1), 59–78.

Gbadebo, S. A., Cumpsty, N. A., & Hynes, T. P. (2005). Three-dimensional separations in axial compressors. *Journal of Turbomachinery*, 127, 331–339.

Gbadebo, S. A., Cumpsty, N. A., & Hynes, T. P. (2008). Control of three-dimensional separations in axial compressors by tailored boundary layer suction. *Journal of Turbomachinery*, 130, 011004–011004-8.

Gmelin, C., Thiele, F., & Liesner, K. (2011). Investigations of secondary flow suction in a high speed compressor cascade (ASME Paper No. GT2011-46479).

Hergt, A., Meyer, R., & Engel, K. (2006). Experimental investigation of flow control in compressor cascades (ASME Paper No. GT 2006-90415).

Hergt, A., Meyer, R., & Engel, K. (2013). Effects of vortex generator application on the performance of a compressor cascade. *Journal of Turbomachinery*, 135, 021026–021026-10.

Hergt, A., Meyer, R., Liesner, K., & Nicke, E. (2011). A new approach for compressor endwall contouring (ASME Paper No. GT2011-45858).

Joslyn, H. D., & Dring, R. P. (1985). Axial compressor stator aerodynamics. *Journal of Engineering for Gas Turbines and Power*, 107, 485–493.

Kožulovic, D., & Röber, T. (2006). Modelling the streamline curvature effects in turbomachinery flows (ASME Paper No. GT2006-90265).

Kulisa, P., & Dano, C. (2006). Assessment of linear and non-linear two-equation turbulence models for aerothermal turbomachinery flows. *Journal of Thermal Science*, 13(1), 14–26.

Launder, B. E., Reece, G. J., & Rodi, W. (1975). Progress in the development of a Reynolds-stress turbulence closure. *Journal of Fluid Mechanics*, 68, 537–566.

Lei, V. M., Spakovsky, Z. S., & Greitzer, E. M. (2008). A criterion for axial compressor hub-corner stall. *Journal of Turbomachinery*, 130(3), 031006–031006-10.

Lewin, E., Kožulovic, D., & Stark, U. (2010). Experimental and numerical analysis of hub-corner stall in compressor cascades (ASME Paper No. GT 2010-22704).

Liu, Y. W., Lu, L. P., Fang, L., & Gao, F. (2011). Modification of Spaltart-Allmaras model with consideration of turbulence energy backscatter using velocity helicity. *Physics Letter A*, 375, 2377–2381.

Mani, M., Babcock, D. A., Winkler, C. M., & Spalart, P. R. (2013). Predictions of a supersonic turbulent flow in a square duct (AIAA Paper No. 2013-0860).

Matsuraka, K., & Kato, C. (2007). Large Eddy Simulation of three-dimensional separations in axial compressors by turbulence modeling. *Proceedings of ECCOMAS, Juvás-kylä*. 219–225.

McMullan, W. A., & Page, G. J. (2011). Large eddy simulation of a controlled diffusion compressor cascade. *Flow, Turbulence and Combustion*, 86, 207–230.

Mehdizadeh, O. Z., Temmerman, L., Tartinvile, B., & Hirsch, C. (2012). Applications of EARSM turbulence models to internal flows (ASME Paper No. GT2012-68886).

Menter, F. R., Langtry, R., & Hansen, T. (2004). *CFD simulation of turbomachinery flows* - verification, validation, and modelling. *Proceedings of ECCOMAS, Juvás-kylä*. 219–225.
Place, J. M. M. (1997). *Three-dimensional flow in core compressors* (PhD thesis). University of Cambridge.

Schulz, H. D., Gallus, H. E., & Lakshminarayana, B. (1990). Three-dimensional separated flow field in the endwall region of an annular compressor cascade in the presence of rotor-stator interaction: Part 1-quasi-steady flow field and comparison with steady-state data. *Journal of Turbomachinery, 112*(4), 669–678.

Speziale, C. G., Sarkar, S., & Gatski, T. B. (1991). Modelling the pressure-strain correlation of turbulence: An invariant dynamical systems approach. *Journal of Fluid Mechanics, 227*, 245–272.

Teramoto, S. (2005). *Large-eddy simulation of a transonic compressor cascade with boundary layer Transition* (AIAA Paper No. 2005-3639).

Tiedemann, C., Heinrich, A., & Peitsch, D. (2012). *A new linear high speed compressor stator cascade for active flow control investigations* (AIAA Paper No. 2012-3251).

Tucker, P. G. (2011). Computation of unsteady turbomachinery flows: Part 1-progress and challenges. *Progress in Aerospace Sciences, 47*, 522–545.

Tucker, P. G. (2013). Trends in turbomachinery turbulence treatments. *Progress in Aerospace Sciences, 63*, 1–32.

Wang, D. H., Lu, L. P., & Li, Q. S. (2009). *Improvement on S-A model for predicting corner separation based on turbulence transport nature* (AIAA Paper No. 2009-4931).

Wang, Z. N., & Yuan, X. (2013). *Unsteady mechanisms of compressor corner separation over a range of incidences based on hybrid Les/RANS* (ASME Paper No. GT 2013-95300).

Weber, A., Schreiber, H.-A., Fuchs, R., & Steinert, W. (2002). 3-D Transonic flow in a compressor cascade with shock-induced corner stall. *Journal of Turbomachinery, 124*, 358–366.

Wilcox, D. C. (1988). Multiscale model for turbulent flows. *AIAA Journal, 26*(11), 1311–1320.

Zachos, P. K., Grech, N., Charnley, B., Pachidis, V., & Singh, R. (2011). Experimental and numerical investigation of a compressor cascade at highly negative incidence. *Engineering Applications of Computational Fluid Mechanics, 5*(1), 26–36.

Zaki, T. A., Wissink, J. G., Durbin, P. A., & Rodi, W. (2009). Direct computations of boundary layers distorted by migrating wakes in a linear compressor cascade. *Flow, Turbulence and Combustion, 83*(3), 307–322.