17th Australasian Fluid Mechanics Conference
Auckland, New Zealand
5-9 December 2010

Prediction of free-stream turbulence variation at the leading edge of an axial compressor blade

S. C. T. Perkins, A. D. Henderson, G. J. Walker and J. E. Sargison

School of Engineering
University of Tasmania, Tasmania 7001, Australia

Abstract

Free-stream turbulence is known to have a large effect on laminar-turbulent transition and the development of boundary layers on axial-flow compressor blades. It is well known that boundary layers in the leading edge region of compressor blades are very sensitive to changes in free-stream turbulence intensity. The paper investigates how a commercial Reynolds Averaged Navier Stokes (RANS) code predicts flow in the leading edge region of a controlled diffusion (CD) compressor blade with a circular arc leading edge using two widely accepted turbulence models. The blade profile is typical of CD stators used in modern axial flow aero gas turbine engines. Simulations were performed using the two-equation Menter Shear Stress Transport (SST) model and the Speziale, Sarkar and Gatski (SSG) Reynolds Stress Model (RSM). Of particular interest is how free-stream turbulence intensity is modelled in areas of high strain-rate and how the turbulence intensity varies as the blade leading edge is approached. The computational study is undertaken to assist in the design of validation experiments which were undertaken in a large-scale 2-D cascade wind tunnel. The findings show that both the SST and the RSM models agree in their prediction of turbulence intensity around the leading edge region and that the RSM model predicts an anisotropic variation of turbulence as the flow travels around the leading edge.

Introduction

Computational fluid dynamics (CFD) has been a key tool in the aerodynamic design of the blading in gas turbine engines for decades and is being relied on more and more every day. Performance predictions during the design process are critically dependent on the boundary layer development and transition to turbulent flow. Many of the key determining factors required in CFD can either be measured directly or modelled accurately using modern computational methods, except for free-stream and wake turbulence which is seldom known in many practical situations [2]. This leads the CFD user into making many, often crude, assumptions and the use of empirical turbulence models. In the leading edge region of a compressor blade, the boundary layer is extremely thin, laminar and highly receptive to free-stream turbulence, especially when laminar separation bubbles form. A number of detailed studies have shown the importance of the leading edge region and how sensitive it is to small changes in geometry and flow parameters [6, 4]. Changes in levels of free-stream turbulence intensity have a significant impact on boundary layer development and thus on blade performance, which in turn can have a significant impact on engine performance [10, 18, 5, 17]. Free-stream turbulence also has a large effect on the mixing out of non-uniformities in enthalpy, entropy and temperature.

It is possible to modify low-speed research facilities to better represent flow inside a multistage machine by installing upstream turbulence generating grids, fitting boundary layer trips to the hub and casing and by installing grids downstream of a stator row to replicate the potential flow field of a downstream stage. Place et al. [12] showed that out of the previously mentioned changes elevating the free-stream turbulence by means of an upstream turbulence generating grid had the largest impact on performance, leading to a 1.8% rise in stage efficiency. Camp and Shin [1] conducted an experimental investigation into the turbulent flow field inside embedded stages of three different low-speed four-stage compressors. They showed that the turbulence intensity at the inlet to stator varied from 3.6 - 7.0% and the turbulence length scale location varied from 3.6 - 7.1 % of mean blade chord. It is expected that these levels would further increase inside real engines.

The price of computing power is decreasing dramatically and the amount of computing power used early in the design process stages is ever increasing, which has lead to a vast reduction in numerical errors. However CFD predictions are still subject to large errors when incorrect models are selected within a particular solver. Hobson [7, 8] showed, experimentally, that at medium to high incidence angles the free-stream turbulence intensity (Tu) increased by an order of magnitude just ahead of and in the leading edge region of a compressor cascade. This phenomenon was attributed not to streamwise diffusion, but to the high levels of shear as the flow undergoes strong acceleration around the leading edge. In regions of adverse pressure gradient Soranna et al. [15] showed, using PIV measurements, that the compressed streamwise component of velocity fluctuation is enhanced and the surface normal components, are stretched and suppressed. The opposite trend was seen in regions of accelerating flow, which is consistent with rapid distortion theory (RDT). It was also shown that in an area either side of peak suction where the local conditions were non-equilibrium, the wake turbulence was reduced by a rapid change in orientation of the compressive and extensional strains. This leads to the turbulence production rate becoming negative, which in turn leads to a decrease in turbulent kinetic energy. The above findings clearly show that the turbulence field in any blade row is going to be highly anisotropic and non-homogeneous. Sarkar et al. [14] indicated that the surface normal fluctuations are more effective than the streamwise fluctuations in initiating transition. It is clear that if the performance of a blade is to be predicted accurately the above mentioned phenomena must be modelled correctly.

In this study the authors investigate how two widely accepted turbulence models predict the flow leading up to and around the leading edge of a compressor blade. Particular attention is focussed on the prediction of the free-stream turbulence field. The turbulence models for this application need to cover a range of scenarios from initial design calculations through to calculations aimed at investigating detailed flow phenomena. Many numerical codes currently used for design work in industry, such as the quasi-three dimensional flow solver MISES [3], utilise an isotropic 'frozen' turbulence assumption.

This paper begins with a brief description of the CFD and experimental set-ups and the turbulence models used. The results and discussion section presents a detailed comparison of the differ-

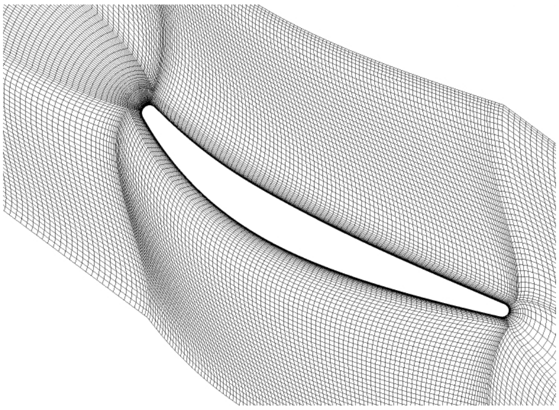

Figure 1: Computational mesh created in ANSYS ICEM with 1.3×10^6 elements

ent test cases presented. Final conclusions are drawn as to the applicability of different turbulence models to predict the turbulent flow field in the leading edge region of a compressor blade.

Numerical Approach

A large scale, low-speed, two-dimensional (2D) compressor cascade, based in the Whittle Laboratory at the University of Cambridge, was also used in this investigation to provide experimental validation of the CFD results. The model was based on this cascade and Table 1 shows some of the key parameters.

Table 1: Compressor Cascade Details

Chord	[mm]	285
Cascade Span	[mm]	610
Cascade Pitch	[mm]	281
Reynolds Number (chord)		2.6×10^5
Inlet Flow Angle (to axial)	[deg]	49

The model domain consists of single blade element with periodic boundary conditions above and below. The full span of the cascade was modelled (0.61 m) with symmetry boundary conditions applied to each side plane. The inlet boundary was located a distance of 0.322 m axially upstream from the leading edge and 0.322 m downstream from the trailing edge. The turbulence intensity and length scale at the inlet plane of the model were chosen to be 8.2% and 0.016 m respectively based on the dimensions of the actual turbulence generating screen used in the cascade and correlations for turbulence decay behind grids [13].

An O-mesh was chosen for the grid type due to its simplicity when applied to 2D geometries. O-meshes are also an essential requirement for accurate modelling of the near wall flow region. ANSYS ICEM was used to create the mesh and the final mesh consisted of 13×10^6 elements for a single passage (Figure 1). This is a very fine grid for a single passage simulation. However, as free-stream turbulence was an important parameter a fine grid was required throughout the three dimensional domain. In the boundary layer region y^+ was less than 1 for the nodes nearest the blade surface.

k-ω SST Model

The $k-\omega$ based Shear Stress Transport (SST) turbulence model is a blend of the $k-\varepsilon$ and $k-\omega$ models [11]. The model does not directly compute the transport of turbulent shear stresses as turbulent kinetic energy k and turbulent frequency ω are being convected in the flow. Bradshaw's assumption is used to account for their transport effects. The $k-\omega$ SST model has been reported to behave well in non-equilibrium boundary layers compared with the popular $k-\varepsilon$ and $k-\omega$ models, and predicts boundary layer separation well.

Reynolds Stress Model

Reynolds Stress Models solve transport equations to directly compute the Reynolds stresses throughout a flow field. It is often referred to as a Second Order Closure and is considered a high order and elaborate turbulence model used in detailed CFD studies. These models are expected to give better results in flows with anisotropic turbulence. The SSG Reynolds Stress model developed by [16] is considered to give superior accuracy than the LLR version developed by [9] in most flows and was chosen for this investigation.

Solution Method

A second order accurate descritisation scheme was used and all solutions were converged to a RMS residual level of 10^{-6}, two orders of magnitude below the default level.

Experimental Measurements

Particle Image Velocimetry (PIV) was used in the suction surface leading edge region of the instrumented cascade blade. The laser sheet was fired in the blade-blade plane at 50% span, whilst the camera was aimed along the blade in a spanwise direction. The PIV interrogation window measured 52×36.25 mm and covered the first 7.5% of blade chord. The PIV results produced both instantaneous and time-averaged information about the velocity flow field as well as key properties such as 2D turbulence intensity and Reynolds stresses. Each time averaged image consisted of approximately 1000 image pairs, which was deemed to be sufficient in order to obtain reliable statistics on the turbulent flow field. Upstream and downstream, of the cascade, three-hole probe traverses were performed on the cascade to check flow periodicity and two-dimensionality. These tests showed that the total pressure varied by less than 1.3% based on free-stream velocity, and that the inlet flow angle varied by less than 0.5^o over the pitch centered about the instrumented blade. Static pressure distributions were also used to confirm the inlet flow angle.

It should be noted that in all the experimental plots the x and y axes are aligned approximately parallel and normal respectively to the blade suction surface immediately downstream of the leading edge circle.

Results and Discussion

Time-averaged velocity flow field

As the flow approaches the leading edge stagnation zone it is forced to decelerate before undergoing rapid acceleration as it is deflected around the circular arc of the leading edge. Where the circular arc meets the main body of the suction surface there is a discontinuity in surface curvature, which causes a velocity over-speed or spike. Following this spike in velocity is a short region of extremely adverse pressure gradient, which can often cause the laminar boundary layer to separate from the blade surface. Downstream of the leading edge spike the flow continues to accelerate gradually until peak suction which assists in maintaining a stable and thin boundary layer. Figure 2 and Figure 3 show that the velocity field is well predicted using the SSG-RSM model and the important features mentioned above can clearly be seen.

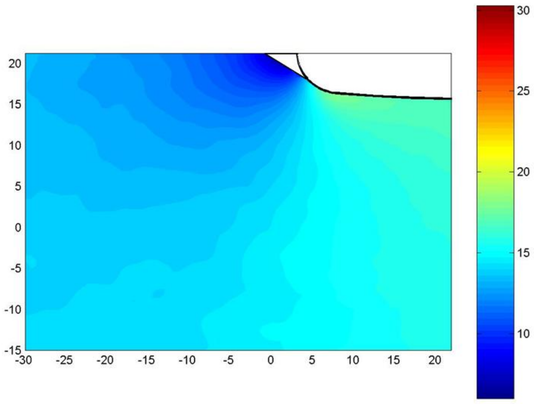

Figure 2: Measured velocity field (PIV)

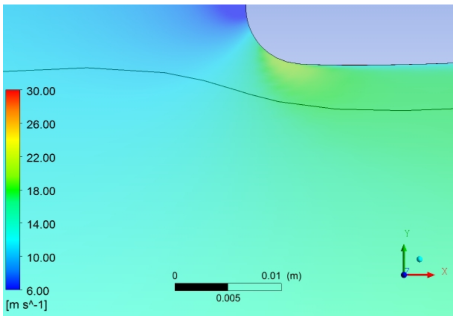

Figure 3: Computed velocity field for SSG-RSM model

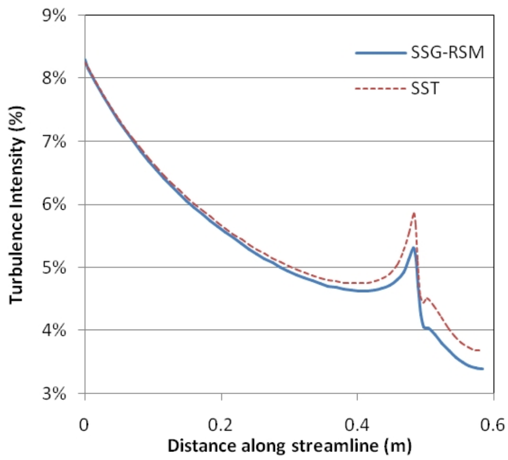

Figure 4: Variation of computed turbulence intensity along a streamline

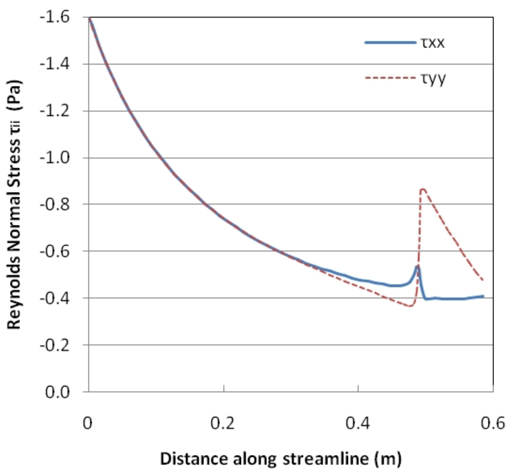

Figure 5: Variation of computed Reynolds normal Stress components $\tau_{xx} = -\rho\overline{u'u'}$ and $\tau_{yy} = -\rho\overline{v'v'}$ along a streamline (SSG-RSM model)

Turbulent flow field

The time-averaged experimental results show a number of interesting and important trends in the free-stream turbulence field, which numerical codes need to predict if the boundary layer development and transition process is to be modelled accurately, such as the variation of turbulence intensity Tu and the development of turbulence anisotropy.

Figure 4 shows the variation of turbulence (Tu) along a streamline starting from the inlet boundary of the model and travelling around the leading edge as shown in Figure 3. Figure 4 shows Tu decays substantially to a level of approximately 4.5% before approaching the region shown in Figure 3. As the leading edge is approached the turbulence intensity rises rapidly. This occurs when the speed decreases relative to the disturbance level, leading to a spike in turbulence intensity near the leading edge. The turbulence level then falls as the flow accelerates around the suction surface of the blade. Both SSG-RSM and SST models give reasonable agreement of this turbulence intensity variation, and the trends have been validated using experimental PIV. In addition, both models predict a free-stream value of just over 4% in the blade passage which is in close agreement with the turbulence intensity measured using a hot-wire probe. This shows that both models are accurately predicting the decay rate.

The high strain-rate of the flow around the leading edge is also shown to cause an anisotropic variation of turbulence properties. Figure 6, from experimental PIV data, shows that the time averaged velocity fluctuations are strongly anisotropic. The relative variation of the velocity fluctuations in the flow field is greatest in regions of high shear strain where energy is transferred from the mean flow to the fluctuating components. The well known turbulence production term $\overline{(u_i u_j)}\frac{\partial U_i}{\partial U_j}$ appears in equations for both the mean kinetic energy and the mean fluctuating kinetic energy, however they have opposite signs and therefore whatever its effect on the kinetic energy of the mean, its effect on the kinetic energy of the fluctuations must be opposite.

The $k-\omega$ SST turbulence model assumes isotropic eddy viscosity throughout the flow domain, and being a two-equation model is known to have poor accuracy in regions of rapidly changing strain-rate. The RSM model, on the other hand, calculates the individual components of the time-averaged velocity fluctuations and is expected to more accurately determine anisotropy within a complex turbulent flow field (Figures 7 and 5). Comparing these plots with Figure 6 shows that the RSM model predicts the anisotropy with varying degrees of success. Figure 7 clearly shows the fluctuating component normal to the direction of the mean flow acceleration being amplified while the component parallel to it is damped. This agrees with RDT but does not match the experimental results in Figure 6.

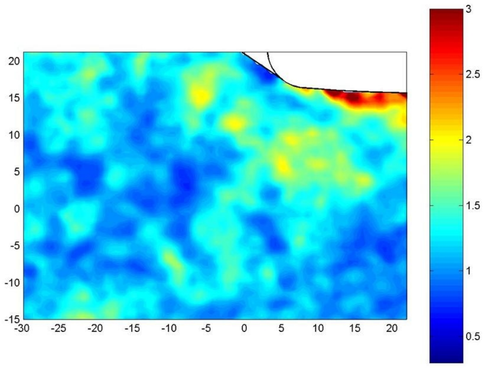

Figure 6: Time average PIV contour plot of u'_{rms}/v'_{rms}

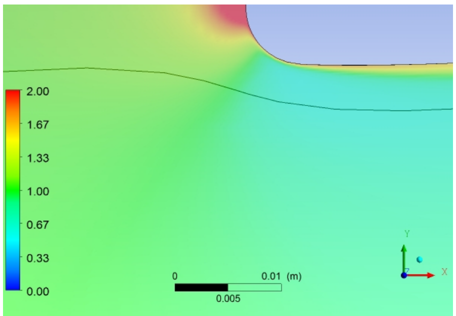

Figure 7: Predicted u'_{rms}/v'_{rms} from the SSG-RSM model

Conclusions

This study has shown the flow around the leading edge of a compressor blade to produce anisotropic variations in turbulence properties. While both $k-\omega$ SST and RMS models agree in their prediction of the turbulent intensity, including the sharp rise as the flow approaches the leading edge, the rapidly changing strain-rate decreases the ability of two-equation turbulence models to accurately model the flow in this region.

Recent studies [6, 19] have shown that the boundary layer at the leading edge is the principal receptivity site for transition processes on the suction surface of compressor blades, and the results from this study further emphasises the need for accurate modelling of free-stream turbulence in this region of flow.

Acknowledgements

Staff of the Whittle Laboratory, Cambridge University are thanked for provision of experimental facilities, helpful discussions, and hospitality extended to the authors.

References

[1] Camp, T. R. and Shin, H. W., Turbulence intensity and length scale measurements in multistage compressors, *Journal of Turbomachinery*, **117**, 1995, 38 – 46.

[2] Denton, J. D., Some limitations of turbomachinery CFD, ASME Paper GT2010-22540, 2010.

[3] Drela, M. and Youngren, H., A user's guide to MISES 2.53, Technical report, MIT Computational Aerospace Sciences Laboratory, 1998.

[4] Goodhand, M. N. and Miller, R. J., The impact of real geometries on three-dimensional separations in compressors, ASME Paper GT2010-22246, 2010.

[5] Gostelow, J. P., Blunden, A. R. and Walker, G. J., Effects of free stream turbulence and adverse pressure gradient on boundary layer transition, *Journal of Turbomachinery*, **116**, 1994, 392–404.

[6] Henderson, A. D. and Walker, G. J., Observations of transition phenomena on a controlled diffusion compressor stator with a circular arc leading edge, *Journal of Turbomachinery*, **132**, 2010, 1 – 9.

[7] Hobson, G. V. and Shreeve, R. P., Inlet turbulence distortion and viscour flow development in a controlled - diffusion compressor as very high incidence, *Journal of Propulsion and Power*, **9**, 1993, 397 – 404.

[8] Hobson, G. V., Wakefield, B. E. and Roberts, W. B., Turbulence amplification with incidence at the leading edge of a compressor cascade, *Journal of Rotating Machines*, **2**, 1999, 89 – 98.

[9] Launder, B., Reese, G. and Rodi, W., Progress in the developments of reynolds-stress turbulence closure, *Journal of Fluid Mechanics*, **68**, 1975, 537 – 566.

[10] Mayle, R., The role of laminar - turbulent transition in gas turbine engines, *Journal of Turbomachinery*, **113**, 1991, 509–536.

[11] Menter, F. R., Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA Journal*, **32**, 1994, 1598–1605.

[12] Place, J. M. M., Howard, M. A. and Cumpsty, N. A., Simulating the multi-stage environment for single-stage compressor experiments, *Journal of Turbomachinery*, **33**, 1996, 706–716.

[13] Roache, P. J., The generation of nearly isotropic turbulence by means of grids, *Journal of Heat and Fluid Flow*, **8**, 1987, 82 – 92.

[14] Sarkar, S. and Voke, P. R., Large - eddy simluation of unsteady surface pressure over a low - pressure turbine blade due to interaction of passing wakes and inflexional boundary layer, *Journal of Turbomachinery*, **128**, 2006, 221 – 231.

[15] Soranna, F., Chow, Y., Uzol, O. and Katz, J., The effect of inlet guide vanes wake impingement on the flow structure and turbulence around a rotor blade, *Journal of Turbomachinery*, **128**, 2006, 82 – 94.

[16] Speziale, C., Sarkar, S. and Gatski, T., Modelling the pressure-strain correlation of turbulence: an invariant dynamical systems approach, *Journal of Fluid Mechanics*, **277**, 1991, 245–272.

[17] Tain, L. and Cumpsty, N. A., Compressor blade leading edges in subsonic compressible flow, *ProQuest Science Journal*, **214**, 2000, 221–242.

[18] Walker, G. J., The role of laminar - turbulent transition in gas turbine engines: A discussion, *Journal of Turbomachinery*, **115**, 1993, 207–216.

[19] Wheeler, A. P. S., Miller, R. J. and Hodson, H. P., The effects of wake induced structures on compressor boundary layers, *Journal of Turbomachinery*, **129**, 2007, 705–712.