

Turbulence modelling of the aerodynamic interaction of OGV wakes and diffuser flow

LI Jing-hua, Page Gary, McGuirk Jim

(Department of Aeronautical and Automotive Engineering, Loughborough University,
Loughborough, Leicestershire, LE11 3TU, United Kingdom)

Abstract: Different turbulence closures were used to predict the flow interaction between the wakes created by compressor outlet guide vanes (OGVs) and a downstream annular pre-diffuser. Two statistical turbulence models were tested based on the classical Reynolds-averaged Navier-Stokes (RANS) approach. Both high- Re and low- Re (Launder-Sharma) versions of the $k-\epsilon$ model were applied to a selected test problem for OGV wake/diffuser flows. The test problem was specifically chosen because experimentally determined inlet conditions and both profile and performance data were available to validate predictions. A preliminary study was also reported of the more advanced large eddy simulation (LES) approach. The LES sub-grid-scale (SGS) model was the basic Smagorinsky eddy viscosity assumption, with a Van-Driest damping function for improved capture of near-wall viscous behaviour. Comparison between the two RANS models showed little difference in terms of velocity contours at OGV trailing edge and diffuser exit. In terms of overall diffuser performance (static pressure recovery and total pressure loss coefficients), the high- Re model was shown to agree well with experimental data. The preliminary LES study indicates the highly unsteady character of the OGV wake flow, but requires improved treatment of inlet conditions.

Key words: outlet guide vane (OGV) wakes; diffuser flow; interaction;
Reynolds-averaged Navier-Stokes (RANS) modelling;
large eddy simulation (LES) modelling

CLC number: TH432

Document code: A

Introduction

Modern aircraft gas turbine design requires that the compressor produces a high pressure rise in a minimum number of stages. As a consequence, the air leaves the compressor with a high axial velocity. Efficient and stable combustion with small total pressure losses can, however, only be achieved with low air velocities. Moreover, the compressor efflux, which feeds the fuel injector and enters the combustor flametube in several locations, should approach the injector and liner air admission ports in a controlled manner; this ensures favourable conditions to achieve the target design values of flow split,

port discharge coefficients, and primary/dilution jet momentum ratios/jet trajectory angles for appropriate depth of jet penetration^[1]. The current best practice for achieving the required flow deceleration is a combination of an annular pre-diffuser behind the compressor outlet guide vane (OGV) row, followed by a sudden expansion dump diffuser. Optimisation of system performance means maximising the velocity reduction within the pre-diffuser, whilst maintaining a stable (i. e. unseparated) flow, before the air enters the dump region — maximising pre-diffuser area ratio at fixed length implies low exit dynamic pressure, hence low dump loss.

However, annular pre-diffuser flow is well

known to be very sensitive to inlet conditions^[1]. This has lead to several investigations of the interaction between the wakes shed by upstream OGVs and pre-diffuser flowfields. Experimental studies by Barker and Carrotte^[2-4] have indicated the beneficial impact that the blade wakes, the associated higher turbulence levels, and other flow structures typically produced by an upstream axial compressor, can have on the performance of a pre-diffuser and the quality of the air supply to the fuel injector. Similarly Walker et al.^[5] have shown how, by adopting an integrated approach (as first suggested in Ref. [6]), the design of the OGV and pre-diffuser may be considered simultaneously, with the OGV wake flow manipulated to advantage in achieving improved pre-diffuser performance. Finally, work on automatic computational fluid dynamics (CFD)-based optimisation of aero-engine OGV/pre-diffuser systems^[7], following the suggestions of Refs. [5-6], has underlined the importance of diffuser inlet conditions, which must be taken into account to gain maximum benefit in reducing total pressure loss in combustor external aerodynamics.

In such CFD-based optimisation, the accuracy of the optimised solution can only be as good as the accuracy of the turbulence modelling used. Since the optimum pressure rise design of diffusers is known to be close to incipient separation, and Reynolds-averaged Navier-Stokes (RANS) statistical turbulence models are known to find accurate prediction of adverse pressure gradient induced separation difficult, it is of interest to study and compare the performance of different RANS turbulence models for OGV wake/diffuser aerodynamic interactions. Equally it may be that the large eddy simulation (LES) approach may give more accurate predictions, and application of this approach to OGV/diffuser flows has not currently been reported. Note, it is not, at present, conceivable for LES models to be used in a CFD-based optimisation methodology such as that described in Ref. [7], since this requires tens to hundreds of CFD runs and

LES would currently be prohibitively expensive. It is these technical aspects which have formed the motivation behind the work presented here.

Different turbulence closures are used to predict the flow interactions between the wakes created by a compressor OGV row and a downstream annular pre-diffuser. Two statistical turbulence models are used based on the classical RANS approach. Both high- Re and low- Re (Launder-Sharma) versions of the k - ϵ model are applied to a selected test problem for OGV wake/diffuser flows. The test problem was specifically chosen such that both experimentally determined inlet conditions and overall system performance data were available^[6]. A preliminary study is also reported of the more advanced LES approach. The LES sub-grid-scale (SGS) model is based on the simple Smagorinsky eddy viscosity assumption, with a Van-Driest damping function for improved capture of near wall viscous behaviour.

1 Numerical methodology

An in-house CFD code, DELTA, has been employed for the current investigation. This code was originally written as a RANS code for aerospace propulsion installation applications (Page et al.^[8]); it was subsequently developed to run also in LES mode (see Refs. [9-11]). A brief description of the DELTA versions used in the current study is presented in the following sections.

1.1 RANS flow solver

The default version of DELTA^[8] constitutes a pressure-based, structured mesh, multi-block, finite-volume solver for compressible flow problems. Accordingly it adopts a mass-weighted (Favre-averaged) approach for all flow variables with exception of density and pressure, which are not mass-weighted. The code solves transport equations for mass, momentum, energy and turbulence model variables. For incompressible flows such as the present problem, solution of the energy equation is unnecessary and a con-

stant density assumption may be made. The conservation equations solved (now considered as Reynolds-averaged) may thus be written:

$$\frac{\partial(\rho \bar{u}_i)}{\partial x_i} = 0 \quad (1)$$

$$\begin{aligned} \frac{\partial(\rho \bar{u}_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho \bar{u}_i \bar{u}_j) = \\ - \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \bar{\tau}_{ij}}{\partial x_j} - \frac{\partial}{\partial x_j}(\rho \bar{u}_i \bar{u}_j) \end{aligned} \quad (2)$$

An overbar indicates Reynolds averaging, $\bar{\tau}_{ij}$ is the time-averaged viscous stress involving the molecular viscosity μ , and the turbulent Reynolds stresses are modeled via:

$$- \rho \bar{u}_i \bar{u}_j = 2\mu_t \left[\frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] - \frac{2}{3} \rho k \delta_{ij} \quad (3)$$

where μ_t is the turbulent viscosity.

The choice of turbulence model is an important aspect for OGV/diffuser aerodynamic interaction modelling. It is important to note that it is not necessary to select a model which has good performance at predicting separation since, for minimum loss, the design will always have to avoid separation. It is the case that the prediction of boundary layer growth in an adverse pressure gradient is one flow characteristic that is important. This argues for a model such as the $k-\omega$ /shear stress transport (SST) model^[12], which is usually regarded as performing well for such flows. The experimental work reported in Refs. [2-3], however, indicates that an even more important flow characteristic that has to be accurately captured is the mixing out of the wake created by the OGV in the diffuser flowfield. This is a free shear flow phenomenon not a near wall flow feature. This argues for a $k-\epsilon$ model^[13], which is usually considered better at capturing free shear flows than $k-\omega$. In the present work, both high- Re and low- Re versions of the $k-\epsilon$ model are considered; predictions with the $k-\omega$ /SST model will also be carried out and reported separately.

1.2 High- Re $k-\epsilon$ turbulence closure

The high- Re model employed in this work is the $k-\epsilon$ model as described by Launder and Spal-

ding^[13]:

$$\mu_t = c_\mu \rho \frac{k^2}{\epsilon} \quad (4)$$

$$\begin{aligned} \frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \bar{u}_i k)}{\partial x_i} = \\ \frac{\partial}{\partial x_j} \left[\frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] + P_k - \rho \epsilon \end{aligned} \quad (5)$$

$$\begin{aligned} \frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial(\rho \bar{u}_i \epsilon)}{\partial x_i} = \\ \frac{\partial}{\partial x_j} \left[\frac{\mu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right] + \frac{\epsilon}{k} (c_{\epsilon 1} P_k - c_{\epsilon 2} \rho \epsilon) \end{aligned} \quad (6)$$

production of turbulence energy P_k is the product of the turbulent component of stress and the mean strain rate tensor:

$$P_k = - \rho \bar{u}_i \bar{u}_j \bar{S}_{ij} \quad (7)$$

$C_{\epsilon 1}, C_{\epsilon 2}, C_\mu, \sigma_k, \sigma_\epsilon$ are the standard empirical constants as defined in Launder and Spalding^[13].

1.3 Launder-Sharma low- Re $k-\epsilon$ turbulence closure

A low- Re turbulence model requires inclusion of viscous diffusion, damping functions, and additional terms to improve the model's ability to capture near-wall behaviour. The model selected here is due to Launder and Sharma^[14]:

$$\begin{aligned} \frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \bar{u}_i k)}{\partial x_i} = \\ \frac{\partial}{\partial x_j} \left[\left(\mu_t + \frac{\mu}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho(\tilde{\epsilon} + D) \end{aligned} \quad (8)$$

$$\begin{aligned} \frac{\partial(\rho \tilde{\epsilon})}{\partial t} + \frac{\partial(\rho \bar{u}_i \tilde{\epsilon})}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu_t + \frac{\mu}{\sigma_\epsilon} \right) \frac{\partial \tilde{\epsilon}}{\partial x_j} \right] + \\ \frac{\tilde{\epsilon}}{k} (c_{\epsilon 1} f_1 P_k - c_{\epsilon 2} f_2 \rho \tilde{\epsilon}) + E \end{aligned} \quad (9)$$

The turbulent eddy viscosity is now written:

$$\mu_t = c_\mu f_\mu \rho \frac{k^2}{\tilde{\epsilon}} \quad (10)$$

The quantity $\tilde{\epsilon}$ and the dissipation rate ϵ are related by:

$$\epsilon = \tilde{\epsilon} + D \quad (11)$$

and the extra terms and damping functions are defined as:

$$E = 2\mu_t \frac{\mu_t}{\rho} \left\{ \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_k} \right\}^2, \quad D = 2\mu_t \left\{ \frac{\partial \sqrt{k}}{\partial x_j} \right\}^2,$$

$$f_1 = 1.0, \quad f_2 = 1 - 0.3 \exp(-Re_t^2),$$

$$f_\mu = \exp \left[\frac{-3.4}{(1 + Re_t/50)^2} \right]$$

where Re_t is the turbulent Reynolds number;

$$Re_t = \frac{1}{\mu_t} \rho \frac{k^2}{\epsilon} \quad (12)$$

Other coefficients are unchanged from their high- Re values.

1.4 LES flow solver

The LES version of DELTA solves the spatially-filtered Navier-Stokes equations for incompressible flow, by volume-averaging over the computational cells; the transport equations are now written (an overbar now indicates a resolved (filtered) variable):

$$\begin{aligned} \frac{\partial(\rho \bar{u}_i)}{\partial x_i} &= 0 \\ \frac{\partial(\rho \bar{u}_i)}{\partial t} + \frac{\partial(\rho \bar{u}_i \bar{u}_j)}{\partial x_j} &= -\frac{\partial P}{\partial x_i} + \frac{\partial \bar{\tau}_{ij}}{\partial x_j} - \frac{\partial \tau_{ij}^r}{\partial x_j} \end{aligned} \quad (13)$$

where P is given by:

$$P \equiv \bar{P} + \frac{1}{3} \tau_{kk} \quad (15)$$

and τ_{ij}^r is the deviatoric part of the SGS stress tensor, given by:

$$\tau_{ij}^r = \tau_{ij} - \frac{1}{3} \tau_{kk} \delta_{ij} \quad (16)$$

where

$$\tau_{ij} = \rho \bar{u}_i \bar{u}_j - \rho \bar{u}_i \bar{u}_j$$

The Smagorinsky^[15] SGS model, written in terms of a Boussinesq-type eddy viscosity assumption, is introduced to calculate τ_{ij}^r . Whilst many SGS models have been proposed at various levels of complexity, it is argued here that, as long as the mesh resolution is such that about 80% of the fluctuating energy are captured (resolved) by the grid, then the adoption of a simple SGS model such as Smagorinsky is sufficient. This is an algebraic model, which takes the form:

$$\mu^{SGS} = \rho l^2 \bar{S} = \rho (C_s \Delta)^2 \bar{S} \quad (17)$$

and the deviatoric part of the SGS stress tensor is then:

$$\tau_{ij}^r = -2\mu^{SGS} \bar{S}_{ij} \quad (18)$$

where $\bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)$ is the filtered strain rate tensor, $\Delta = (\Delta x \Delta y \Delta z)^{1/3}$ is the filter length, $\bar{S} = (2\bar{S}_{ij} \bar{S}_{ij})^{1/2}$ is the magnitude of $\bar{S}_{ij}$ and C_s denotes the Smagorinsky constant; Δx ,

Δy and Δz represent the local cell size in the Cartesian directions.

In order to account for the reduced growth of small scales near walls, the value of the eddy viscosity μ^{SGS} has to be reduced. Thus, the Smagorinsky model is modified according to Van Driest^[16] via:

$$\begin{aligned} \mu^{SGS} &= \rho (C_s \Delta)^2 \bar{S} = \\ &\rho [C_{so} (1 - \exp(-y^+ / A^+)) \Delta]^2 \bar{S} \end{aligned} \quad (19)$$

Where $C_{so} = 0.1$, $y^+ = \frac{\rho y u_\tau}{\mu_t}$ and $A^+ = 25$; u_τ is the skin friction velocity, y the wall normal distance, and μ_t the molecular viscosity.

1.5 Numerical details

The RANS version of the DELTA code uses structured multi-block grids, with all solution variables stored at cell centres. Spatial fluxes are computed at cell faces using a generalised formulation of a monotonic upstream-centered scheme for conservation laws (MUSCL)-type scheme for convection. A parameter allows selection of a range of discretisation schemes (1st order, central differencing, 2nd and 3rd order upwind), and alternative limiter schemes are also available.

Numerical modifications, as reported in Refs. [9, 11], have been introduced to increase the accuracy of DELTA in LES mode. Second-order convection discretisation using a total variation diminishing (TVD) principle is implemented as an explicit deferred correlation to the basic first order upwind scheme; all diffusive terms are discretised using central differencing. Temporal advancement is effected via an explicit, low-storage Runge-Kutta 3rd order accurate method. Finally, modified scaling of the standard Rhie and Chow^[17] smoothing terms was also introduced to take account of the very small time steps needed in LES calculations, combined with the significant variations in cell volume size across the solution domain. Previous calculations of internal flows^[18] and free shear flows (jet flow)^[19] with this second order approach have demonstrated that this numerical methodol-

ogy provides adequate LES performance. The code has been parallelised to allow use of a personal computer (PC) cluster; the multi-block structure allows a straightforward domain-decomposition approach to parallelisation.

2 Test case geometry, grid, boundary conditions

2.1 Test case geometry and grid

A test case with comprehensive experimental data has been chosen in this paper. Full details of the test rig geometry are reported in Barker et al^[6]; the layout of the annular passage occupied by the OGV row and pre-diffuser is shown in Fig. 1, which thus constitutes the solution domain geometry for the present study. The mean radius of the OGV blade row is 375 mm with a passage height h of 36.6 mm. The OGV row consisted of 160 vanes each of 39 mm chord and with a thickness/chord ratio of 6 percent. The OGV row was preceded by a single stage rotor and an inlet guide vane (IGV) row with a long annular passage upstream; there were 160 OGVs and 80 IGVs, so the azimuthally repeating sector for this geometry is 4.5° corresponding to 1 IGV and 2 OGVs. At OGV exit the blade row is connected to an annular diffuser with an area ratio and axial length typical of current aero-engine designs. In this test case the OGV blades have a two-dimensional (2-D) cross section with no blade sweep or lean.

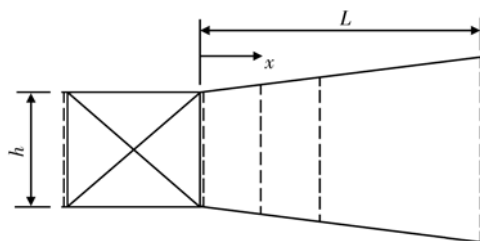


Fig. 1 OGV/diffuser geometry, annular passage view

The computational domain for the numerical predictions (Fig. 2) extended from just upstream of the OGV leading edge to diffuser exit, and covered a 2 OGV sector. Experimentally measured conditions^[6] were available at the inlet plane of the chosen domain over this 2 OGV sec-

tor; for the present study, this data was pitch-averaged circumferentially to produce radially varying profiles only, so strictly speaking only 1 OGV passage could have been considered. A future study will attempt to capture IGV wake effects and so the full solution domain has been retained for this work for simplicity. Mesh generation was carried out using the Rolls-Royce parametric blade meshing software PADRAM^[20]. This provides a structured multi-block grid designed specifically for meshing blades within annular or linear cascades. The mesh for high- Re predictions consisted of approximately 8 million cells. A near wall refined mesh was also generated for low- Re RANS—approximately 10 million cells with 32 nodes in the first 1 mm normal wall distance, with $y^+ \leq 0.3$ for the first node. As a first step this low- Re grid has also been used for LES.

2.2 Boundary conditions

For the RANS simulation, the inlet profiles corresponded to pitch-averaged measured radial profiles of all three components of velocity; these were specified to be uniform azimuthally over the whole 2 OGVs inlet plane. Ref. [6] had also carried out hot-wire measurements that allowed estimation of the inlet profiles for k and ϵ . A zero gradient outflow condition was employed at diffuser exit. Periodic boundary conditions were applied in the circumferential direction on the azimuthal side planes. For the high- Re model, a standard wall function method based on the logarithmic wall-law was adopted. For the low- Re model, integration down to the wall allowed simple no-slip conditions for velocity and for turbulence variables, fixed conditions: $k = 0$ and $\epsilon = 0$.

In the LES predictions, for this preliminary study, the same radially only varying profiles of all three components of velocity (as employed in RANS) were used at domain inlet with no superimposed fluctuations. This is recognized as a gross simplification and is currently being improved. At the diffuser outlet a convective outflow condition was set, and at the side azimuthal

boundaries, the same periodic boundary treatment as used in RANS.

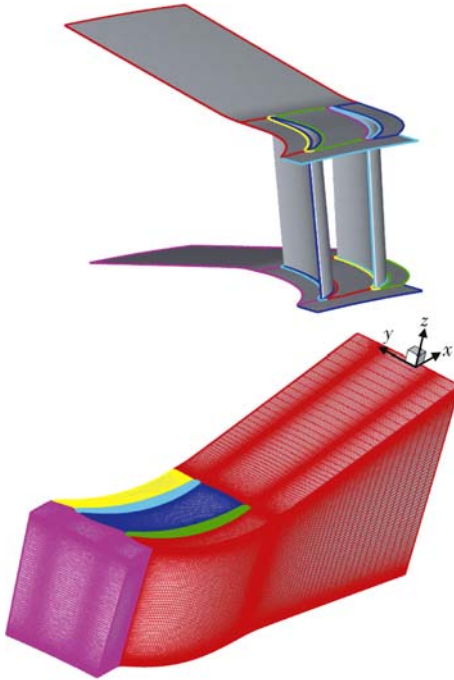


Fig. 2 CFD geometry (solid model) and multi-block topology

3 Results

3.1 RANS results

Fig. 3 presents axial velocity contours close to the vane (suction) surface (less than 1 mm normal distance) for both high- Re and low- Re models. The flow over the leading edge is seen to be 2-D over much of the span, but the interaction of the end-wall boundary layers and the acceleration over the leading edge creates three-dimensional (3-D) secondary flows that give rise to two regions of low momentum near the vane hub and tip endwalls. These features can be identified with two peaks in the inlet swirl velocity profile associated with the rotor characteristics.

Both models predict rather similar flow development, with the low- Re model producing slightly greater acceleration around the leading edge, and hence stronger secondary flows. The near hub low momentum region appears earlier, but the velocities are smaller in the upper region, which is close to separation near the trailing edge.

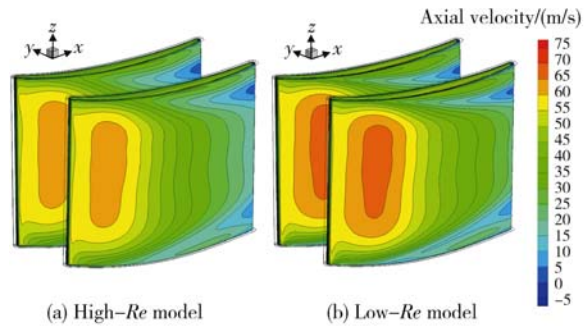


Fig. 3 Axial velocity contours on vane suction surface

Predicted contours of axial, radial and circumferential velocity components at OGV exit from both high- Re and low- Re $k-\epsilon$ model solutions are illustrated in Fig. 4. The two low momentum regions identified above can be seen to form two zones of thicker wake in the axial velocity contours in Fig. 4. The secondary flows generated by the upstream blade row can be seen best in the radial velocity contours. A small a-

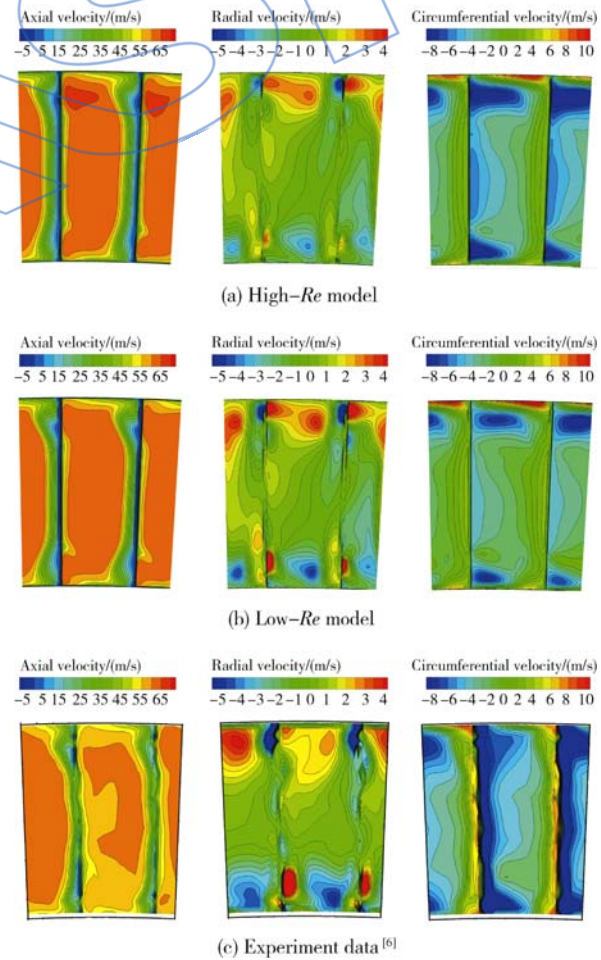


Fig. 4 Predicted OGV exit velocity contours

mount of residual swirl remains, predominantly associated with the low momentum regions identified in the axial velocity field. It can be seen that the predicted wake thickness from the low- Re model is thinner (particularly over the central portion of the vane span), and the regions where the vane has not taken all the swirl velocity out of the flow are smaller. In general comparison with the measured velocity components shown in the bottom part of Fig. 4 shows very good agreement in terms of the pattern of velocity distribution in all components, and good agreement in terms of magnitude. The thinner wake and more concentrated secondary flow regions predicted by the low- Re model are closer to the measured behaviour, although the underturning predicted by the high- Re model in the central part of the vane is also seen in the measured data. The development of the OGV-created velocity field as it passes through the annular pre-diffuser (axial component only) is given in Fig. 5. This figure indicates how the wakes generated by the upstream blade row develop as influenced by the superimposed effects of turbulent mixing due to shear stresses and the adverse pressure gradient generated by the diffuser area ratio. The OGV wakes grow in thickness in the central part due to mixing. The secondary velocity field leads to curved distortions of the axial velocity contours (principally in the hub/tip regions); the contour levels generally decrease in magnitude as flow passes down the diffuser, associated with adverse pressure gradient effects. The secondary flows also lead to the low momentum fluid in the two thickened wake regions combining with the end-wall boundary layers to form two distinct low velocity regions next to the walls. These regions are where separation would take place first, but for the diffuser length/area ratio considered no negative velocities are predicted. Both high- Re and low- Re models show essentially the same characteristics of the wake development. The stronger secondary flows noted above for the low- Re model produce more curved contours. The two near-wall low velocity regions have

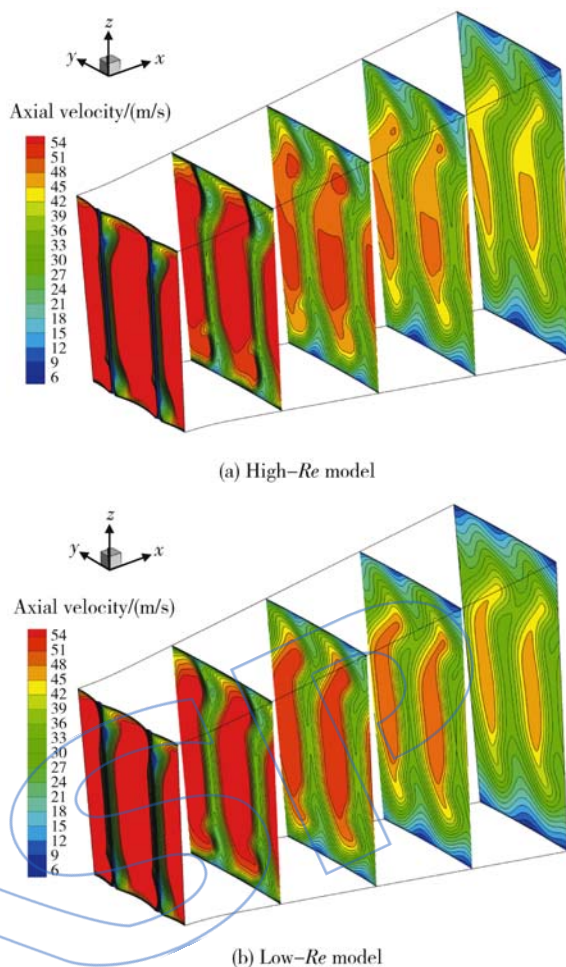


Fig. 5 Predicted axial velocity contours through the diffuser

merged by diffuser exit in the high- Re -predictions, but these remain distinct in the low- Re solution. In the experiments of Ref. [6] the velocity field was measured at pre-diffuser exit and this is compared to RANS predictions in Fig. 6. The agreement is good. Although the flow has remained attached to both casings over the whole

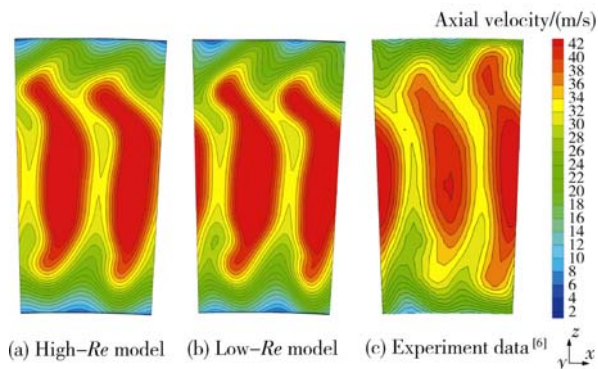


Fig. 6 Axial velocity contours at diffuser exit

sector, circumferential variations in the flowfield caused by OGV wakes are still clearly evident. The 3-D velocity and turbulence conditions provided by the upstream rotor and OGV row have enabled the low-energy flow adjacent to each casing to survive the pressure rise in the pre-diffuser of area ratio 1.6. The near-end wall flow is predicted to be in a worse state than observed in the measurements by both models. Note also that the different nature of the two high velocity regions in the measured data is due to the fact that one OGV has an IGV in front of it and one does not. At present the pitch-averaging approach adopted for the inlet profiles in the current RANS predictions do not allow capture of this effect. Solutions are underway using full 2-D inlet conditions. However, there is still some evidence that the secondary flows seem stronger in the experimental data than the current predictions.

A detailed comparison of predicted and measured wake capture is best examined from circumferential profiles at fixed radial height, and this is shown for the mid-annulus height in Fig. 7 at OGV exit. It can be seen that compared with high- Re model predictions the low- Re model shows a thinner wake, which is closer to the measured data at the OGV exit.

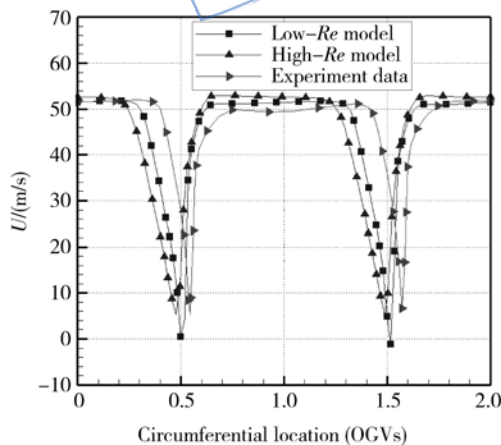


Fig. 7 Wake axial velocity profiles at OGV exit

Fig. 8 shows circumferentially-averaged axial velocity profiles plotted against radial height at OGV exit and at diffuser exit. The agreement

between both RANS models and experimental data at OGV exit is very good. Due to the adverse pressure gradient, strong deceleration and boundary layer growth occurs throughout the diffuser, although, as noted above, the flow does remain attached on both endwalls. Compared with the measured diffuser exit profile, RANS predictions from both models predict the endwall boundary layers to be in worse condition than measurements, particularly on the outer wall.

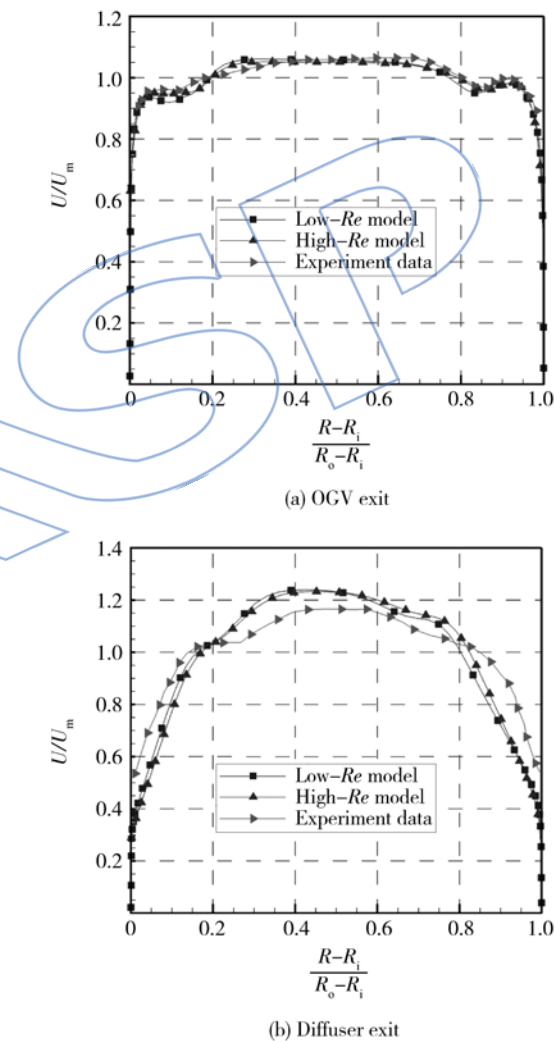


Fig. 8 Circumferentially averaged axial velocity

Fig. 9 presents an overall picture of flow development through the diffuser as represented by static pressure and turbulence energy characteristics. Only the high- Re predictions are shown as the low- Re solution is similar. The stagnation region at the vane leading edge is evident. The boundary layer growth and the pressure/suction

surface boundary layer merging to form the OGV wake can be seen best in the turbulence energy field. The double-peaked nature of the contours in the wake is evidence of two boundary layers. Within the downstream diffuser, the steep velocity gradients in the wake generate large turbulent shear stresses, which reduce as the wake mixes out; the pressure forces associated with the increasing area duct then start to dominate flow development. The pressure rise occurring in the downstream diffuser is enhanced by the mixing out of blade wakes, and it is notable in Fig. 9 how the increasing static pressure field quickly becomes spanwise uniform. Finally, high- Re k - ϵ model predictions of the overall performance of the OGV/diffuser system are assessed against the mass-averaged static pressure rise coefficient (C_p) and total pressure loss coefficient (λ) defined and measured in the experimental study^[6]. Table 1 shows that the high- Re model provides good agreement, underpredicting overall loss but only by about 2%. This provides good justification for the adoption of a high- Re model in the optimisation methodology as described in Ref. [7]; on the basis of the evidence in this study, it seems a low- Re model is not required.

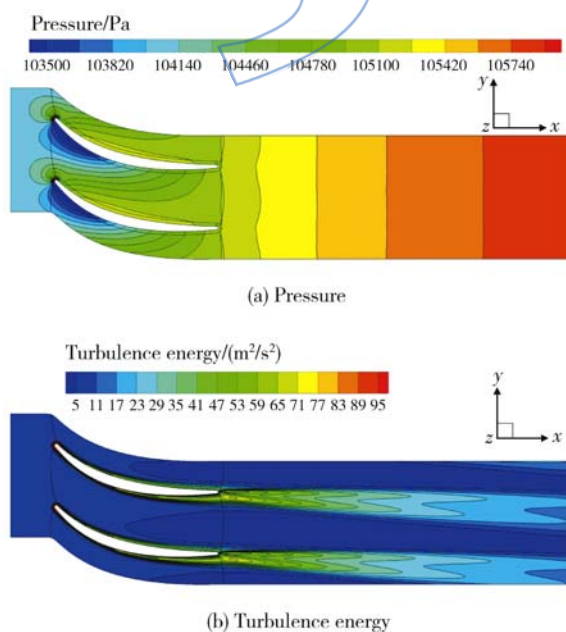


Fig. 9 Predicted flow contours at annulus mid-height (high- Re model)

Table 1 Static pressure rise coefficient and total pressure loss coefficient

	Experiment data ^[6]	High- Re model
$C_{p, \text{vane}}$	0.419	0.371
λ_{vane}	0.111	0.102
$C_{p, \text{diffuser}}$	0.550	0.541
$\lambda_{\text{diffuser}}$	0.055	0.070
$C_{p, \text{overall}}$	0.676	0.657
λ_{overall}	0.141	0.139

3.2 LES results

The DELTA flow solver was run in LES mode for the OGV/diffuser problem. In this preliminary study the same mesh was used as in the low- Re RANS predictions; in addition the inlet boundary conditions were set as the same as used in the RANS predictions. It is recognised that both these assumptions will need improvement before a direct comparison between LES and RANS predictions is possible; thus here only illustrative indications of the LES solutions are provided. The LES calculations were run with a time step of 4×10^{-7} s and 900 000 time steps were performed to obtain statistically stationary mean values. Contours of instantaneous axial and radial velocity components through the entire OGV/diffuser flowfield on a mid-annulus plane are presented in Fig. 10. The unsteadiness in the wake can be clearly seen, and there is evidence of large scale eddies also towards the vane trailing edge on the suction surface.

The origin of the trailing edge large scale unsteady motions can be seen in Fig. 11, which shows time-averaged axial velocity contours on two planes close to the vane suction surface, one at just 0.25 mm normal distance and the second at 0.9 mm. A separated flow on the suction side of the blade is produced; this is extremely thin (see Fig. 11), but, as all separation zones, is extremely unsteady. The consequence is that large turbulent shear stresses are produced associated with this flow region. These are illustrated in Fig. 12. It can be seen that the high shear

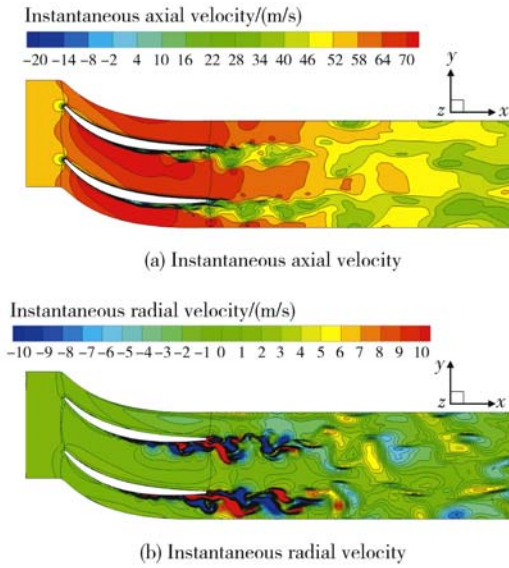


Fig. 10 LES predicted contours of instantaneous axial velocity and radial velocity

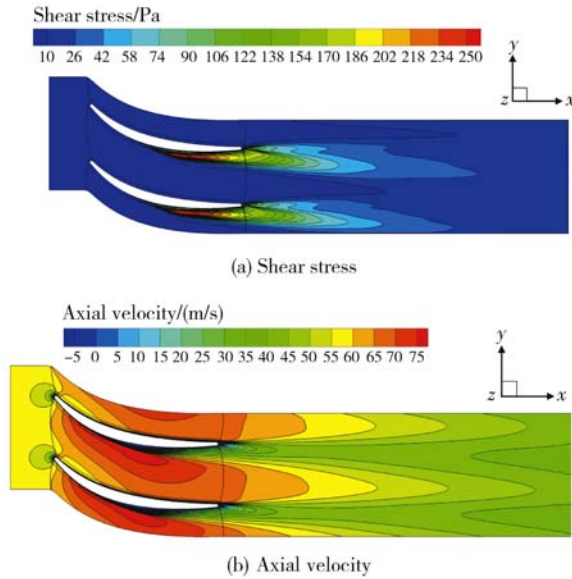


Fig. 12 Contours of statistical shear stress and axial velocity

There is high probability that this separation is caused by either (i) the use of steady inlet conditions, or (ii) an inadequate (for LES) mesh resolution on the vane surface. Work is underway to remedy both these issues and only then can a proper comparison of LES with RANS performance be made.

4 Summary/conclusions

A comparative study has been made of two RANS turbulence models (standard high- Re and Launder-Sharma low- Re $k-\epsilon$ models) applied to the prediction of the flow interaction between the wakes shed by a compressor OGV row and an annular pre-diffuser of area ratio 1.6 and non-dimensional length 2.23. A test case was chosen^[6] where inlet profiles were readily available for CFD inlet boundary conditions, as well as flow development profiles and overall pressure rise/loss coefficients for validation purposes. The results of the predictions showed that, although differences were observed between the two models (for example the low- Re model produced thinner OGV wakes in better agreement with expt. data), the high- Re model provided good agreement with flow development and overall performance data. The choice of a high- Re model for use in CFD-based optimisation of such

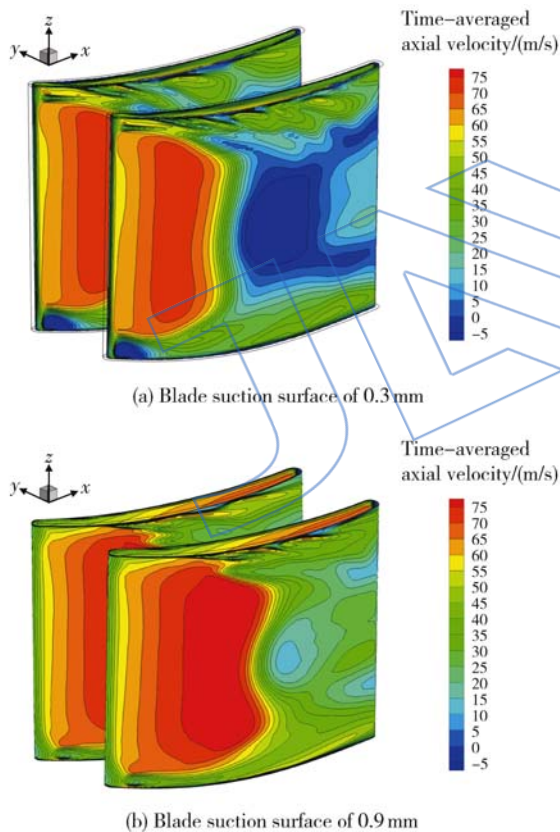


Fig. 11 LES predicted contours of time-averaged axial velocity on blade suction surface

stresses lead to a small separation in the trailing edge wake and also rapid mixing of the wake in the downstream diffuser. These features are in contrast to the non-separated behaviour observed in the experiments.

flows was thus justified by the present investigation, with no need for the more advanced low-Re model, which requires much finer mesh resolution. Preliminary calculations were also performed using the LES approach for the same OGV/diffuser flow problem. Whilst these predictions showed promising capture of large scale unsteady wake motions, the accuracy of the LES is currently hampered by the adoption of simplistic (steady) inlet conditions and also possibly the need for better mesh resolution of the OGV boundary layers, which lead to unrealistic flow separation on the OGV suction surface. Future work is underway to apply more realistic inlet conditions, refine the LES mesh, and also study an alternative LES SGS model.

Acknowledgments

The authors would like to acknowledge helpful discussions and collaborations with other members of the Loughborough University Technology Centre in Combustion Aerodynamics. The author is grateful for receipt of an Engineering and Physical Sciences Research Council (EPSRC) Dorothy Hodgkin Postgraduate Award.

References:

- [1] Klein A. Characteristics of combustor diffusers[J]. *Progress in Aerospace Sciences*, 1995, 31(3): 171-271.
- [2] Barker A G, Carrotte J F. Influence of compressor exit conditions on combustor annular diffusers Part I: diffuser performance[J]. *AIAA Journal of Propulsion and Power*, 2001, 17(3): 678-686.
- [3] Barker A G, Carrotte J F. Influence of compressor exit conditions on combustor annular diffusers Part II: flow redistribution within the diffuser[J]. *AIAA Journal of Propulsion and Power*, 2001, 17(3): 687-694.
- [4] Barker A G, Carrotte J F. Compressor exit conditions and their impact on flametube injector flows[J]. *ASME Journal of Engineering for Gas Turbines and Power*, 2002, 124(1): 10-19.
- [5] Walker A D, Carrotte J F, McGuirk J J. Enhanced external aerodynamic performance of a generic combustor using an integrated OGV/diffuser design technique [J]. *ASME Journal of Engineering for Gas Turbines and Power*, 2007, 129(1): 80-87.
- [6] Barker A G, Carrotte J F, Luff J, et al. Design of an integrated OGV/diffuser system [R]. Loughborough: Final Report of LOPOCOTEP Project, Report No. TT03R01, Department of Aeronautical and Automotive Engineering, Loughborough University, 2003.
- [7] Wang J, Priddin C H, McGuirk J J. CFD-based automatic optimisation of flow interactions in combustor OGV/pre-diffuser systems [R]. Loughborough: Internal Report No. TT11R16, Loughborough University, 2011.
- [8] Page G J, Zhao H, McGuirk J J. A parallel multi-block RANS method for propulsion installation applications [C]// *Proceedings of 12th International Symposium on Air Breathing Engines*. Melbourne, Australia: AIAA, 2001, 1: 864-876.
- [9] Page G J, Li Q, McGuirk J J. LES of impinging jet flows relevant to vertical landing aircraft[R]. AIAA-2005-5226, 2005.
- [10] Wu X, Page G J, McGuirk J J. An approach to improve high-frequency noise prediction in LES of jets[R]. AIAA-2006-2442, 2006.
- [11] Veloudis I, Yang Z, McGuirk J J, et al. Novel implementation and assessment of a digital filter based approach for the generation of large eddy simulation inlet conditions [J]. *Flow Turbulence and Combustion*, 2007, 79(1): 1-24.
- [12] Menter F R. Two-equation eddy viscosity turbulence models for engineering applications[J]. *AIAA Journal*, 1994, 32(8): 1598-1605.
- [13] Launder B E, Spalding D B. The numerical computation of turbulent flows[J]. *Computer Methods in Applied Mechanics and Engineering*, 1974, 3(2): 269-289.
- [14] Launder B E, Sharma B I. Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc[J]. *Letters in Heat and Mass Transfer*, 1974, 1(11-12): 131-140.
- [15] Smagorinsky J. General circulation experiments with the primitive equations I: the basic experiment[J]. *Monthly Weather Review*, 1963, 91(3): 99-164.
- [16] Van Driest E R. On turbulent flow near a wall[J]. *Journal of Aeronautical Sciences*, 1956, 23(11): 1007-1011.
- [17] Rhie C M, Chow W L. Numerical study of the turbulent flow past an aerofoil with trailing edge separation[J]. *AIAA Journal*, 1985, 21(11): 1525-1535.
- [18] Dunham D, Spencer A, McGuirk J J, et al. Comparison of unsteady Reynolds averaged Navier-Stokes and large eddy simulation computational fluid dynamics methodologies for air swirl fuel injectors[J]. *Journal of Engineering for Gas Turbines and Power*, 2009, 131(1): 011502(8 pages).
- [19] Karabasov S A, Afsar M Z, Hynes T P, et al. Using large eddy simulation within an acoustic analogy approach for jet noise modelling[R]. AIAA-2008-2985, 2008.
- [20] Shahpar S, Lapworth L. Parametric design and rapid meshing system for turbomachinery optimisation[R]. *ASME GT2003-38698*, 2003.