Abstract
In a gas turbine engine secondary flows have a detrimental effect on efficiency. The current numerical study is aimed at determining which turbulence model in a commercially available CFD code is best suited to predicting the secondary flows. Experimental validation is used to determine the appropriateness of the model. The numerical study was performed using Numeca’s FINE™/Turbo and all of the appropriate turbulence models were tested. It was found that the Baldwin-Lomax, Spalart-Allmaras and $k$-ε predicted the magnitude of the velocity well, but did not capture the velocity magnitude profile well. The $k$-ω and the SST $k$-ω captured the profile better, but did not predict the average value as well as the other models tested. It is for this reason that the SST $k$-ω turbulence model was chosen as the most suitable for analysis of secondary flows, as the flow features are more accurately predicted, thus aiding in understanding secondary flow.

Nomenclature
AGS  Abu-Ghannam and Shaw transition model
$k$  Turbulent kinetic energy (m$^2$.s$^{-2}$)
$\varepsilon$  Turbulent energy dissipation rate (m$^2$.s$^{-3}$)
$\omega$  Specific dissipation rate (s$^{-2}$)
SST  Shear Stress Transport

Introduction
A great deal of research over the years has gone into reducing secondary flows in gas turbine engines, to understand them better. There is yet to be a prediction method that accurately predicts secondary flows [1]. Most of the studies that have been performed have only dealt with experimental investigations of planar or annular cascades, with some studies concentrating on fundamental studies using cylinders [2].

The reason for the popularity of linear cascade studies is due to the reduced complexity. Detailed measurements between the blades are also possible. In an annular, rotating environment it can be prohibitively expensive to perform detailed measurements. For this reason it has become more and more popular to perform numerical studies in place of experimental studies.

Computational Fluid Dynamics (CFD) is currently in common use to analyse turbomachinery due to the significant amount of information that can be obtained. One of the topics of study is the reduction of secondary flows. Reducing the secondary flows reduces the losses and is thus of great interest.

The secondary flow vortex system was first described by Hawthorne [3]. As shown in Figure 1, the passage vortex is the dominant secondary flow feature which has been reported by numerous researchers namely: Marchal and Sieverding[5], Moore and Adhye [6], Moustapha et al. [7], Hodson and Dominy [8], and Harrison [9], to name a few. Quoting Gregory-Smith [10], Ingram [11] stated that the passage vortex is formed when a sheared flow is turned. The slower fluid follows a tighter radius of curvature, which leads to the tangential flow across the passage. In order to conserve continuity a vortical flow is generated. The cross passage pressure gradients act to strengthen the passage vortex [12].

Horseshoe vortices are another dominant flow structure. They are caused by the impingement of the boundary layer on the leading edge of the blade, a phenomenon that can also be seen with cylinders [2]. In Figure 2 it can be seen that the horseshoe vortex gets entrained in the passage vortex. Figure 2 was reproduced from Eymann et al. [4].

In a rotating environment the inlet flow is skewed as it enters the rotor. As the boundary layer flow enters the rotor, the boundary layer gets skewed due to the change of reference frame caused by the rotating hub. The passage vortex gets further strengthened by this flow [12,14,15,16]. Thus in a rotating environment, like a gas turbine engine, the passage vortex is more pronounced than in a cascade.

Since the secondary flow has a detrimental effect on the performance of the turbine engine, much research has gone into reducing the secondary flows. Some of the methods used include, but are not limited to:

- Blade stacking variations such as the approach of Watanabe and Harada [17].
- Leading edge bulbs and fillets such as the work by Lethander et al. [18], and Zess and Thole [19].
- Axisymmetric contouring has been investigated by many authors, for example Boyle et al. [20].
- Non-axisymmetric endwall contouring, Gregory-Smith [21], Brennan [22] and Harvey et al. [23].

In order to analyse how these reduction mechanisms work, it is necessary to understand how the flow is affected by addition of the mechanism. Numerical analyses are ideal for investigating the flow since all the flow properties are available everywhere in the domain. A suitable turbulence model is required however to capture the flow effects accurately. The current
Investigation is aimed at determining which of the turbulence models available in a commercial turbomachinery CFD code; FINE\textsuperscript{TM}/Turbo will produce the best results. Once the best turbulence model is determined, it would prove useful in understanding how the previously listed loss reduction mechanisms work.

**Experimental setup**

The experimental rig used to validate the CFD was described in Snedden et al. [24]. The instrumentation can be seen in Table 1. The geometry of the blades used was based on the design used in the Durham cascade [25]. The blades were however redesigned such that they would be more appropriate in an annular rotating environment, instead of a linear cascade as originally tested. Thus the blade profile at the hub is the same as in the Durham cascade, but the profile at the tip differs to account for rotation. More about the Durham cascade setup and geometry can be found in Harvey et al. [25], Hartland et al. [26], Ingram [11] and Snedden et al. [24].

Since the measurement is done with a 5 hole probe, profiles of the absolute velocity magnitude will be used for comparison. The measurements downstream of the rotor are averaged because of the rotation and the slow response time inherent in 5 hole probes. Three passes were made downstream of the rotor at different tangential locations so that an average could be taken to minimise the effects of clocking.

**Investigation of turbulence models**

NUMECA’s FINE\textsuperscript{TM}/Turbo 7.4.1 CFD package was used for the numerical portion of the current investigation.

![Figure 1: Hawthorne’s classical secondary flow model [3]](image1)

![Figure 2: Schematic showing the various vortical structures in a turbine reproduced from Eymann et al. [4] who reproduced it from Vogt and Zippel [13].](image2)

**Table 1: Experimental instrumentation used by Snedden et al. [24]**

<table>
<thead>
<tr>
<th>Primary Instrumentation</th>
<th>Instrument</th>
<th>Uncertainty</th>
</tr>
</thead>
<tbody>
<tr>
<td>Torque</td>
<td>Himmelstein MCRT 28002T(5-2)CN-TGA-G + Model 721</td>
<td>±0.03N.m</td>
</tr>
<tr>
<td>Speed</td>
<td>Mechanical Power Instrument</td>
<td>2RPM</td>
</tr>
<tr>
<td>Barometric Pressure</td>
<td>Siemens Sitrans P 7MF4233-1FA10-1AB6-Z A02+B11</td>
<td>0.05% of full scale</td>
</tr>
<tr>
<td>Differential Pressure</td>
<td>5 x Siemens Sitrans P 7MF4433-1CA02-1AB6-Z A02+B11</td>
<td>0.05% of full scale</td>
</tr>
<tr>
<td>Temperature</td>
<td>PT1000 RTD’s</td>
<td>±0.05°C</td>
</tr>
</tbody>
</table>

**Secondary Instrumentation**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Instrument</th>
<th>Uncertainty</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steady Flow mapping</td>
<td>Aeroprobe CPC5-C159-305-015.3-16</td>
<td>0.8% in Velocity magnitude, 0.4° in flow angles</td>
</tr>
<tr>
<td>Turbulence</td>
<td>TSI 1211-20 single component film</td>
<td>±0.77% mean velocity*</td>
</tr>
<tr>
<td>Tangential Traverse</td>
<td>Custom cable system rotating the outer casing</td>
<td>Better than 0.01°</td>
</tr>
<tr>
<td>Radial and Yaw traverse</td>
<td>Rotodata Mini actuator</td>
<td>0.01mm</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.1°</td>
</tr>
</tbody>
</table>

*Stamatios (2002)

**Numerical boundary conditions**

The current steady state investigation is a precursor to future unsteady state research. The unsteady method decided upon was the Domain Scaling method, which is a moving mesh method. Thus the grid used is chosen such that it can be used for steady state as well as unsteady state simulations. To meet the interchangeable criterion the grid had to be created such that the rotor and stator have a matching periodicity. FINE\textsuperscript{TM}/Turbo sets this requirement for the Domain Scaling method such that the area of the face upstream of the rotor/stator interface was equal to the face downstream of the interface.

The profiles used to generate the turbine blades can be seen in Figure 3a. The 1½ stage blade assembly used for the CFD analysis can be seen in Figure 3b.
Since the current investigation is interested in the endwall effects, it was considered necessary to have as low a \( y^+ \) value as possible. The grid was thus generated with \( y^+ < 2.5 \) for the entire endwall and blade surfaces. In order to capture the flow features adequately a fine mesh was used, containing over 5 million cells. This approaches the maximum possible for the available computer, which had 8 gigabytes of RAM.

**Turbulence models**

The turbulence models investigated were those available in FINE\textsuperscript{TM}/Turbo V7.4, thus no user coded, or modified turbulence models were investigated. The choice of turbulence models investigated was based on the list of recommended, low Reynolds number models found in the user manual [28] for the flow conditions to be tested. The turbulence models investigated are as follows:

- Baldwin-Lomax
- Spalart-Allmaras
- Spalart-Allmaras with the Abu-Ghannam and Shaw (AGS) transition model
- The low Reynolds number Yang-Shih \( k-\epsilon \) model
- The non-linear low Reynolds number \( k-\omega \) model
- The Wilcox \( k-\omega \)
- The Shear-Stress Transport (SST) \( k-\omega \)

**The Baldwin-Lomax** is ideal for design cycle analysis where a robust and fast model is required. It is a two layer algebraic (zero equation) model. The turbulent viscosity is calculated differently in the inner and outer layers. Prantl’s mixing length model is used in the inner layer, whereas the mean flow and the length scale are used for the outer layer [28].

**The Spalart-Allmaras** turbulence model is a one equation model. Unlike the Baldwin-Lomax model, the Spalart-Allmaras model solves the transport equation for eddy viscosity, thus it is always continuous. Due to the robustness and ability to treat complex flows the Spalart-Allmaras has become very popular. Compared to the \( k-\epsilon \) turbulence model it is more robust and less computationally expensive [28].

**The Spalart-Allmaras with AGS** is the standard Spalart-Allmaras, with a transition model incorporated in order to determine the transition of the boundary layer from laminar to turbulent. For this investigation only the Abu-Ghannam and Shaw model was investigated, even though other transition techniques are available. The AGS model is based on the correlations obtained by Abu-Ghannam and Shaw [29] obtained using experimental data of transition on a flat plate with pressure gradients [28].

**The low Reynolds number Yang-Shih \( k-\epsilon \) model** is a form of the standard \( k-\epsilon \) turbulence model that does not have a wall function. It infers the boundary layer profile from the input values at the wall [28].

**The non-linear low Reynolds number \( k-\epsilon \) model** does not use the first order closure model approximation according to the Boussinesq hypothesis. It is time consuming and is recommended for research only [28].

**The Wilcox \( k-\omega \) model** [30] predicts free shear flow spreading rates that show good agreement with experiments for far wakes, mixing layers and planar, round and radial jets [28].

**The SST \( k-\omega \) model** was developed to blend the robustness and accurate formulation of the \( k-\omega \) and the free stream independence of the \( k-\epsilon \). The \( k-\omega \) and the \( k-\epsilon \) are both multiplied by a blending function and added together [31]. The blending function is created such that in the near wall regions the \( k-\omega \) is used, and in the free stream regions, far from the walls, the \( k-\epsilon \) is used [28,32].

![Figure 3: Blade geometry used for the current investigation](image-url)
Boundary Conditions

The type of boundary conditions that were chosen coincided with the values that were directly measured in the experimental setup of Snedden et al. [24], as shown in Table 2. The inlet was specified by velocity components as velocity was the quantity that was controlled in the experiment. The outlet was specified as having a radial pressure equilibrium at midspan ($R=0.1725$). The working fluid was set as air with perfect gas attributes. The characteristic values listed in Table 2 are used by FINE$\textsuperscript{TM}$/Turbo to calculate certain values [28].

Table 2: CFD Boundary conditions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet velocity [m/s]</td>
<td>21.38</td>
</tr>
<tr>
<td>Inlet temperature [K]</td>
<td>293</td>
</tr>
<tr>
<td>Number of rotor blades</td>
<td>20</td>
</tr>
<tr>
<td>Number of stator blades</td>
<td>30</td>
</tr>
<tr>
<td>Inlet turbulence [%]</td>
<td>&lt;1</td>
</tr>
<tr>
<td>Outlet pressure [kPa]</td>
<td>82.9</td>
</tr>
<tr>
<td>Rotational speed [RPM]</td>
<td>2300</td>
</tr>
<tr>
<td>Characteristic length [m]</td>
<td>0.06</td>
</tr>
<tr>
<td>Characteristic velocity [m/s]</td>
<td>25</td>
</tr>
<tr>
<td>Characteristic density [kg/m$^3$]</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Method of comparison

In order to ensure that the turbulence model is appropriate the CFD results will be compared with experimental data. The experimental data used for the comparison was the 5-hole probe steady state measurements. The CFD is therefore run in steady state, using the experimental conditions in order to make an appropriate comparison.

Comparison with pressure plots was not performed because the measurement technique only allowed an approximated static pressure. The 5-hole probe was connected using a modified connection method of Kaiser [33]. The calibration was performed using the method of Ingram [34]. The static pressure is calculated using the pitch ports only, since the yaw ports are only measured relative to each other, allowing a more accurate yaw angle but less accurate static pressure.

Results and Discussion

As can be seen from the list of turbulence models there are some variants. For instance the Spalart-Allmaras model is run with transition modelling and two versions of the $k$-$\omega$ model were tested.

In order to simplify the graphs the variants are compared with each other first. The model that fits the trend best is then compared with all the other turbulence models. By splitting up the comparison in this manner, the number of data series present on the graphs is reduced, making the graphs less cluttered, and easier to interpret. Downstream of the stator there is very little variation in the velocity profile when comparing all the different turbulence models and their variants. Thus none of the variants are plotted in Figure 5a.

It was thought that due to the low Reynolds number the flow might be transitional in regions. Thus the inlet turbulence boundary conditions were altered to investigate possible effects of the inlet conditions. For the Spalart-Allmaras three values of turbulence viscosities were used, namely $\mu_i=0.0001$, $\mu_i=0.0005$ and $\mu_i=0.0009$. The default value in FINE$\textsuperscript{TM}$/Turbo is $\mu_i=0.0001$ and is typical for external flows [28]. For turbo-machinery flows the recommended value of turbulence viscosity is between $\mu_i=1$ and $\mu_i=5$, thus the extreme values as well as a middle value is chosen (these values are based on the viscosity of air as being $\mu=1.8e^{-3}$Pa.s).

Comparison of variants

Looking at Figure 4a and Figure 4b it is evident that there is negligible difference between the different variants, even when changing the inlet turbulence level. Changing the turbulent viscosity level any more increases it beyond the recommended levels [28], thus ratios beyond this limit were not tested. Figure 4a shows that the predicted values were at the appropriate level.

The experimental average velocity is $v=24.92$ m/s compared to the numerical average velocity of $v=24.79$ m/s. The trend is approximately correct, but it does not capture all features in the profile. Since all the variants of the Spalart-Allmaras produce approximately the same velocity profile, only the standard Spalart-Allmaras will be compared with the other turbulence models, due to its reduced computational expense.

The biggest differences between the experimental results and the numerical results are found at the hub and casing. Since the resolution in the experimental measurements at the hub and casing are not as fine as the numerical results, the differences in turbulence models in this region are not relevant.

As with the Spalart-Allmaras, the $k$-$\varepsilon$ variants are very similar. To reduce complexity the Yang-Shih $k$-$\varepsilon$ turbulence model is thus treated as the best, since the added complexity of the variants do not improve the accuracy.

The $k$-$\omega$ variants are significantly different, and are thus compared directly with the other turbulence models.

Comparison of turbulence models

With the differences in the turbulence model variants known it is now possible to find which turbulence model produces the best correlation. Looking at Figure 5a it is evident that none of the turbulence models show an ideal correlation. The turbulence models either have a good area averaged velocity, or a good reproduction of the velocity profile. A good reproduction of the velocity profile is sought, since the current investigation is more interested with flow features and not mean values.

Figure 5 shows that all the turbulence models produce very similar results. At the casing, however the correlation is not as good as at the hub. The low value of
the point \( R = 0.196 \) \( m \) in the experimental velocity profile is thought to be due to the tip leakage flow. It is thought that the over-prediction of the velocity at the casing is due to manufacturing anomalies in the experimental setup. The tip gap could be slightly larger then predicted, either due to a blade being slightly shorter or due to the casing not being perfectly circular at the measurement locations. Some of the discrepancy could also be attributed to the hole through which the probe is inserted into the flow.

With reference to Figure 5b it is clear that all the turbulence models fail in some aspect or another. The Spalart-Allmaras, Baldwin-Lomax and the \( k-\epsilon \) follow the values at an appropriate level, but do not capture the profile very well. The \( k-\omega \) and the SST \( k-\omega \) however follow the profile reasonably well, but fail to capture the correct values.
In the current investigation the trend is of more importance then the actual values, since the values can generally be captured experimentally. The trend is a result of the flow features, these features and the manner and location in which they are initiated are of importance, since this aids in understanding of the flow features. Thus the \( k-\omega \) models are the most appropriate.

The Wilcox \( k-\omega \) [30] captures the trend very well at the casing, with the tip leakage flows peaking in the same locations. Conversely at the hub it does not quite capture the position of the peaks. The trend shows the correct type of features, but in the wrong locations. The SST \( k-\omega \) however captures the hub features very well, but captures the flow features poorly at the casing. Therefore the SST \( k-\omega \) was chosen, since the area of interest is the flow near the end wall.

**Conclusions and Recommendations**

Looking at the velocity profiles it was evident that none of the current turbulence models were sufficient to model the secondary flows present in a rotating environment. Thus the most appropriate turbulence model was determined from a set of available turbulence models in a commercial CFD code, Numeca’s FINE\textsuperscript{TM}/Turbo. It was found that the Spalart-Allmaras and the Baldwin-Lomax models are adequate for the cases where the average velocity, and thus flow rates are of importance. The Spalart-Allmaras and the Baldwin-Lomax models also have a lower computational expense then the other models.

The \( k-\epsilon \) models gave a fair approximation, but did not improve the accuracy of the simulation when compared to the added computational expense. The SST \( k-\omega \) model performed the best out of the two \( k-\omega \) models tested. It did not capture the velocity values very accurately, but it captured the velocity magnitude profile the best of all the turbulence models tested.

With reference to the data it is evident that Baldwin-Lomax is still one of the best all purpose models for turbomachinery. The SST \( k-\omega \) shows some promise as it predicted the velocity magnitude reasonably well, but the Baldwin-Lomax predicted the radial and tangential velocities better.

Due to the complex nature of secondary flow it may still be sometime before computational hardware and the numerical models are such that the complexities can be appropriately modelled. Until such time it is important that new models be investigated and validated against experimental data. It is also recommended that the computational expense be carefully weighed against the required accuracy. FINE\textsuperscript{TM}/Turbo does offer higher order turbulence models, but they are non-linear and not recommended for use in design cycle analysis [28] due to the added computational expense.

**Acknowledgments**
The authors would like to gratefully acknowledge Project VITAL and the CSIR for financial support, without which none of the current research would have been possible.

**References**


