ABSTRACT
This paper presents the results of a combined experimental and CFD investigation of a first stage low pressure (LP) turbine nozzle guide vane (NGV). The configuration is characterized by a small number of low aspect ratio vanes in a strongly diverging annulus. The experimental part of the study has been conducted in a two-stage rotating rig environment with detailed area traverse measurements upstream and downstream of the NGV. The inlet whirl to the NGV has been varied by a row of pre-swirl vanes (PSV) located in the turbine rig inlet. The effect of inlet whirl angle on the development of the passage vortices and on the NGV pressure loss is assessed by using measurements and steady state 3D CFD predictions. The area traverse results, which neglect any mixing loss downstream of the traverse plane, show the traditional airfoil loss-loop characteristic with minimum loss at design incidence and increased loss at both positive and negative incidence. By considering the turbine overall performance, however, it is observed that the isentropic turbine efficiency increases for negative incidence and decreases for positive incidence to an extent that cannot be explained by the measured NGV total pressure loss only. This leads us to believe that the non-uniform flow field at the exit of the NGV generated by the wake and secondary flow impacts significantly on the loss generation further downstream in the turbine. Unsteady 3D CFD predictions of the NGV and the downstream rotor passage aerodynamics have been conducted in order to study the convection of the NGV wake and passage vortices through the rotor passage and the resulting unsteady loading on the rotor. The strongest interaction takes place in the hub region. The unsteady loading in the tip region is characterized by incidence variations which typically only affect the rotor unsteady pressure distribution in the leading edge region up to 40% axial chord. In the hub, however, the strong three-dimensional character of the flow forces large amplitude fluctuations to occur in the rotor unsteady pressure distribution at the throat region. These fluctuations affect the character and the level of diffusion on the suction surface and are therefore likely to have significant impact on the rotor loss. In striving to reduce manufacturing cost and engine weight by exploiting high outer annulus wall slope and smaller gap-to-chord ratios these types of effects are bound to become increasingly important. The application of advanced prediction tools, such as used in this paper, will inevitably play an important role in this development.

1. NOTATION
\( c \) \\
\( c_x \) \\
\( c_s \) \\
\( f_s \) \\
\( f_r \) \\
\( f_T \) \\
\( L_P \) \\
\( NGV \) \\
\( P_r \) \\
\( P_s \) \\
\( PSV \) \\
\( T \) \\
\( T_s \) \\
\( T_r \) \\
\( u \) \\
\( \Delta T \) \\
\( \Delta H_{stag}/u^2 \) \\
\( \gamma \) \\
\( \Pi \) \\
\( \eta \) \\
\( \delta P \) \\
\( \delta \eta \)

2. INTRODUCTION
LP turbines used in modern high by-pass ratio civil aero-engines typically have a polytropic efficiency in the range of 91% to 92%. It appears as if the techniques currently applied in the aerodynamic design of LP turbines have reached a level of maturity that makes it difficult to significantly improve on the turbine efficiency in the near future. The challenge for the future instead lies with reducing the manufacturing cost and power plant weight at retained, or improved, performance.
This results in an aspect ratio for the first stage NGV which is considerably lower than for the rest of the blading.

This paper presents detailed aerodynamic measurements and CFD calculations for a low aspect ratio LP turbine NGV with steep outer annulus wall slope. Measurements have been conducted in a two-stage LP turbine rig with varying inlet whirl capability. The strength of the secondary flow in the guide vane passage varies with the inlet whirl. At positive incidence the increased cross-passage static pressure gradient in the vane passage will generate a stronger passage vortex which will migrate further into the free-stream. The aerodynamic loss of a blade row is commonly divided into three sources: 1) inlet loss that occurs upstream of the leading edge of the vane, 2) profile and trailing edge loss associated with the boundary layers on the airfoil surface and 3) endwall loss. In this study the total aerodynamic loss is derived by area traverse measurements upstream and downstream of the vane and secondary flow development in the NGV passage is predicted using a steady state viscous CFD prediction method. 3D unsteady calculations have been carried out for the NGV and the downstream rotor blade to gain a better understanding of the convection of the NGV wake and secondary flow through the downstream rotor. Additional loss mechanisms associated with the low aspect ratio vane arise from 1) the mixing of the NGV wake and passage vortex in the downstream blade rows and 2) additional loss in the downstream blade rows due to high amplitudes of unsteady flow. It is shown that these two loss contributions can be significant and that they need to be understood, quantified and managed to further increase the annulus wall slope from current levels.

3. EXPERIMENTAL SET-UP

The experimental study was carried out in a two-stage uncooled full scale engine parts rotating rig at the altitude test facility in Stuttgart, Germany. The facility operates with pressurized air supply at the turbine inlet and with an exhaust extraction system to simulate the full range of flight conditions. A row of PSVs, located in the inlet parallel annulus, about 2 NGV chords upstream of the NGV leading edge (Fig. 2), was used to vary the first stage NGV inlet whirl. Turbine performance measurements were taken in the performance planes P1 and P4. The temperature drop derived from the torque meter and corrected for windage, bearing and heat loss compares well with the rake measured temperature drop. Static pressure measurements were taken on both annulus walls immediately upstream of the NGV leading edge and at the downstream traverse measurement plane. Pressure and suction surface static pressure measurements were also taken on one airfoil at 10%, 50% and 90% height. The facility typically delivers a turbulence level of 2% at the turbine rig inlet. The supplied pressure level was adjusted to simulate typical cruise Reynolds numbers (5 x 10^6 based on NGV true chord and exit Mach number).

Full area traverses at 22 radial and 26 circumferential locations were conducted simultaneously at stations P1 and P2. At plane P1 a sector corresponding to 1.3 PSV pitch was traversed using a three-hole cobra probe. At plane P2 a five-hole cobra probe was used in order to resolve the radial com-
ponent of the flow velocity. The circumferential extent of the area traverse in this plane was 1.3 times the pitch of the NGV. The traverse planes were staggered circumferentially such that the wake disturbance of the upstream probe did not influence the measurements at plane P2. Both traverse probes were used in fixed angle calibration mode and have been calibrated over a range of ±20° yaw angle. The five-hole probe has also been calibrated in the range of 10° to 40° pitch angle. The approximation of the calibration coefficients leads to a measurement uncertainty of ±0.4° and 0.5% of the Mach number. The measurement points closest to the walls are taken at approximately 3.5 mm distance from the inner and outer annulus walls, which corresponds to one probe head diameter. The measurements at these points have not been corrected for the influence of the probe on the flow field. Previous investigations have shown that the type of probe head used is insensitive to variations in Reynolds number and no separate calibration for Reynolds number has been done. The accuracy of the position of the probe is ±0.1 mm in the radial direction and ±0.1 deg in the circumferential direction. The accuracy of the pressure scanner is 0.1% of the inlet total pressure. The PSVs are 2D symmetric airfoils which can be adjusted to obtain ±10° of inlet whirl angle to the turbine. The traverse measurement in plane P1 (located about 3.6 chords downstream of the PSV trailing edge) shows that the wake of the PSV is well mixed out at this location. Surface flow patterns were obtained by injecting a mixture of silicon oil and titanium oxide particles from locations on both endwalls immediately upstream of the NGV.

4. INLET WHIRL IMPACT ON NGV AERODYNAMICS

The detailed flow field in the NGV passage has been predicted for a range of inlet whirl angles using the 3D MEFP code, which has been supplied by Rolls-Royce plc. The definition of incidence that is used in this paper is shown in Fig. 3. This computation method is based on the pressure correction method (Moore and Moore, 1985) and has previously been extensively validated and found to accurately predict both static pressure distributions and loss levels (Lakshminarayana, 1991). The program employs an algebraic mixing length model (Moore and Moore, 1985) and transition needs to be specified at a discrete point by the user. The mesh consists of 93 (stream-wise) x 41 (circumferential) x 33 (span-wise) computational cells. This gives about 7 grid points in the endwall boundary layer and approximately the same number of grid points in the boundary layer on the airfoil surface. The inlet conditions are specified at plane P1 (Fig. 2). A constant inlet total temperature is specified and the radial distributions of total pressure and whirl angle are derived from the area traverse measurements at the same plane. The exit condition is specified at the leading edge plane of the downstream rotor. Here, the radial distribution of static pressure is based on a throughflow prediction matched to the rig measured static pressures on the annulus walls. Computed 3D surface Mach number distributions at hub, mid and tip sections are com-
Fig. 4. NGV surface Mach number distributions at design incidence.

Fig. 5. Radial distributions of circumferentially mass averaged total pressure upstream (plane P1) and downstream (plane P2) of the NGV.
compared with S1-S2 predictions and measured distributions (Fig. 4.a-c). Due to the high outer annulus wall slope and, in particular, the high outer annulus wall curvature immediately upstream of the NGV leading edge, the velocity ratio is low in this region. The strong passage vortex on the outer annulus wall unloads the tip region. This effect can be clearly seen from the loading diagram and is very well predicted by the 3D CFD method (Fig. 4.c). The radial distribution of circumferentially mass averaged total pressure at planes P1 and P2 are shown in Fig. 5.a-c both for measurements and 3D CFD predictions at three different incidence angles (0°, -10° and +10°). The inlet boundary layers on the inner and outer annulus walls are convected through the passage and radially redistributed at the exit of the NGV. The exit total pressure distribution is very well predicted by the 3D CFD method. The radial migration of the tip loss core is significantly greater than the migration of the hub loss core. Fig. 5.a shows the tip loss core at 60% height whereas the hub loss core is so close to the hub that it is difficult to detect. The large radial migration of the tip loss core is due to the radial pressure gradient forcing the low total pressure fluid radially inwards. This effect is strengthened by the relatively thick outer annulus wall inlet boundary layer. The radial migration both on the hub and the casing decreases with negative incidence (Fig. 5.b) and increases with positive incidence (Fig. 5.c). At positive incidence the cross-passage static pressure gradient is stronger in the leading edge region leading to an earlier formation of the passage vortex and hence an increased radial migration at the exit.
Predicted particle trajectories on the airfoil suction surface have been compared with streak lines on the suction surface obtained from oil flow visualization (Fig. 6.a-b). Major features are very well predicted. The location of the suction surface separation line of the passage vortices are very well predicted both in the hub and tip regions. Between the two separation lines on the suction surface (in the region 20% to 50% height) the suction surface boundary layer is unaffected by the secondary flow. The oil flow visualization on airfoil 2 (Fig. 6.a) indicates that the suction surface boundary layer remains laminar, or possibly transitional, a significant distance downstream of the throat. In this region, where the shear stress is low, the suction surface boundary layer shows streak lines with a strong inward radial component. When transition occurs the shear stress increases and the streak lines are again aligned in the stream-wise direction. The situation for airfoil 1 (Fig. 6.a) is different. A similar radial inflow cannot be detected. This indicates that an earlier and shorter transition occurs here. A close look at the streak lines on several airfoils around the annulus revealed that the vanes which have been instrumented with leading edge probes consistently did not show the radial migration of the suction surface boundary layer whereas the uninstrumented vanes consistently show it. Therefore, it can be concluded that the instrumentation affects the suction surface boundary layer and triggers transition earlier than it would otherwise take place. The CFD prediction (Fig. 6.b) does not show the radial deflection of the streak lines in the region of 20% to 50% height and more closely compares with the flow visualization on the instrumented vane (airfoil 1). For the CFD predictions transition to turbulence has been specified at a discrete point at the location of peak Mach number on the suction surface. This is a reasonable assumption for Reynolds numbers around $5 \times 10^5$. As a result, the flow in the transitional region, e.g. the radial component of the streak lines, is not predicted. The complicated flow associated with transition within a laminar separation is not well modeled by the current transition criterion. Nevertheless, despite this shortcoming the computations accurately predict the radial distribution of loss. Measured and predicted contours of constant total pressure (non-dimensionalized by inlet total pressure) at the NGV exit (plane P2) are shown in Fig. 7.a-b at design point. The radial migration of low total pressure fluid is evident. The hub loss core appears very close to the hub annulus whereas the tip loss core appears at about 60% height. Outboard of about 60% height the NGV wake is tangentially inclined and has reached...
a relatively high degree of mixing due to the interaction between the NGV wake and the tip passage vortex. In the hub, however, both the area traverse measurements and the predictions indicate a significantly increased wake depth compared with the wake depth in the tip region.

An interesting aspect of the vane is the local airfoil section stack in the hub region. The structural requirements of the radial struts severely limit the possibility of optimizing the stack to control the secondary flow. The local stack in the hub region (Fig. 8) results in the airfoil surface not being perpendicular to the endwall. The pressure surface normal points radially outwards and the suction surface normal points radially inwards. This local airfoil lean in the hub region produces a radial force on the flow which has an undesirable effect on the flow in the corner between the suction surface and the annulus wall. Fig. 8.a shows secondary flow vectors in the hub suction surface corner region. A corner vortex, with the opposite sign to the passage vortex is apparent. The formation of this corner vortex leads to increased endwall blockage and increased endwall loss. Fig. 8.b shows the predicted results for an improved airfoil stack. It can be seen that the size of the corner vortex is significantly reduced. Fig. 9 shows that the predicted pressure loss is reduced by an improved airfoil stack in this region and suggests that the designer can exercise some control over endwall loss.

5. IMPACT OF INLET WHIRL ON TURBINE EFFICIENCY
One of the objectives of the turbine rig test was to establish the impact of varying turbine inlet whirl on turbine performance. The turbine isentropic efficiency was measured for three different PSV settings (-10°, 0° and +10°) using both torque meter derived temperature drop and temperature drop from rake measurements. The isentropic efficiency \( \eta \) is defined in Eq. (1) for an uncooled adiabatic turbine assuming constant specific work \( c_p \).

\[
\eta = \frac{\Delta T}{T(1 - \Pi^{(\gamma - 1) / \gamma})} \quad \text{Eq. (1)}
\]

Eq. 1 can be differentiated to express the change in pressure loss (\( \delta \Pi / \Pi \)) as a function of a change in isentropic efficiency (\( \delta \eta / \eta \)) at constant specific work \( c_p \Delta T / T \).

\[
\frac{\delta \Pi}{\Pi} = \frac{\gamma - 1}{\gamma - 1} \frac{\Delta T}{T} \frac{\delta \eta}{\eta} \quad \text{Eq. (2)}
\]

Hence, the exchange rate between total pressure loss and isentropic efficiency depends on \( \Delta T / T, \eta \) and the ratio of specific heats \( \gamma \). For this particular application the exchange rate is given in Eq. (3).

\[
\frac{\delta \Pi}{\Pi} = -1.38 \frac{\delta \eta}{\eta} \quad \text{Eq. (3)}
\]

For varying PSV setting but constant turbine reduced speed \( (N / \sqrt{T}) \) and specific work \( \Delta H / T \) one could assume that any
change in isentropic efficiency is only due to a change in the LP 1 NGV pressure loss since the flow conditions to the other blade rows, at least in a radial and timewise averaged sense, are unchanged. It is important to note that the absolute level of LP 1 NGV pressure loss cannot be obtained from the turbine efficiency measurements. The discussion will instead focus on the change in total pressure loss with incidence angle. For this purpose the ratio of pressure loss to pressure loss at design condition (ΔP/ΔP_{dp}) is used. Fig. 10 displays this quantity as a function of PSV setting both derived from the area traverse measurements and using the turbine efficiency measurements. Derived NGV pressure loss in the area traverse plane P2 shows the traditional airfoil loss loop characteristic with minimum loss at design incidence and increasing loss both at positive and negative incidence. The pressure loss derived from the turbine efficiency measurements, however, indicates a much different trend. The lowest loss is found at negative incidence and increases monotonically with increasing incidence (Fig. 10). The total pressure loss derived from shaft torque measurements and that derived from temperature rake based efficiency both show the same trend. The NGV inlet swirl angle affects the turbine performance in a manner which cannot be explained by only considering the LP 1 NGV loss. There are additional loss mechanisms downstream of the NGV which are strongly dependent on the LP 1 NGV turning. The additional loss mechanisms are due to: 1) mixing loss of the NGV wakes and passage vortices in the downstream blade rows and 2) additional boundary layer loss in the downstream blade rows due to large amplitudes of unsteady flow. The impact of these mechanisms was assessed with the aid of an unsteady 3D calculation of the LP 1 NGV/rotor stage.

6. STATOR-ROTOR PREDICTION

Unsteady 3D computations have been carried out using the VISIUN-2FR 3D viscous unsteady code which has been extensively applied and validated for a large range of different flow conditions (Schlechtriem and Lötzerich, 1997). The actual ratio of NGV-to-rotor airfoils of 21/100 has been approximated by a 1/5 ratio. Fig. 11 shows a snapshot of the two-frame-of-reference computation at mid-span. A total of 327624 grid points were used to discretize the six airfoil passages (one NGV passage and five rotor passages). The NGV discretization consists of 113256 points (66 stream-wise x 33 span-wise and 52 circumferential). The number of points in one rotor passage is 56 x 33 x 23. The mesh used is too coarse in the tip region. Consequently, predicted rotor loss is not accurate. The focus of this study is instead on the propagation of the NGV wakes and secondary flow through the rotor and their effect on the unsteady rotor pressure distribution. This is mainly an inviscid phenomenon within the rotor passage, and the mesh size has been selected to accurately resolve the non-uniformities associated with the NGV wake and secondary flow. During the iteration process, rotor airfoils move relative to the stator airfoils. Fig. 11 shows the accurate treatment of the wake-blade interaction. The algorithm at the sliding boundary shows an excellent conservation of the inherent flow structure. Moreover, the phase lag of the pressure field and the associated lift distribution is calculated properly.

Entropy contours have been chosen to highlight the effects of the wake-blade interaction (Fig. 12). Due to the NGV exit flow angle distribution and the secondary flow/wake interaction the NGV wake is not a straight radial line. The wake in the tip region precedes the wake at lower heights. As a result, an oblique interaction of the NGV wake with the rotor airfoil takes place. This blade vortex interaction induces additional vortices in the rotor flow field. In this context a discussion of the unsteady effects is inevitable. Due to the pitch ratio two blade passing frequencies are involved:

\[ T_s = \frac{P}{u} = \frac{i}{i_s} \quad \text{Eq. (4)} \]

\[ T_r = \frac{x}{u} = \frac{i}{i_r} \quad \text{Eq. (5)} \]

where \( i \) is the angular frequency, \( u \) is the blade pitch and \( w \) is the rotation speed, respectively. A useful quantity in unsteady aerodynamics is the reduced frequency \( k \):

\[ k = \frac{c_i}{c_x} \quad \text{Eq. (6)} \]

where \( c_x \) is the axial chord and \( c_i \) is the axial velocity. It turns out that \( k = 0.5 \) for the rotor and \( k = 7 \) for the NGV. For reduced frequencies in the range 0.1 to 2 it could be expected that the amplitudes of the pressure fluctuations are large and that the phase shift between the forcing (NGV wake) and the unsteady lift distribution is large. In this reduced frequency range both inertial effects (e.g. local time derivatives of momentum) and vortex shedding are significant. The loading diagrams (Fig. 14.a-c) clearly show that once the wake blade interaction has taken place the flow does not have time to recover to an undisturbed state before the next interaction occurs.

In Fig. 10 the entropy contours at an axial plane at 33% blade chord are shown and in Figs. 14.a-c the loading diagrams at 10%, 50% and 90% height are shown for the same instant in time. In the entropy plot the positions of the rotor airfoils are numbered from 1 to 5 and the numbering of the loading distributions refer to the same rotor airfoils. Due to the time periodicity the loading distributions for the five different rotor airfoils at an instant in time can also be viewed as loading distributions for one of the rotors taken at time intervals of T/5.

The dominant aerodynamic feature of the NGV exit flow field is the strong passage vortex in the tip region. Therefore, the most significant unsteady pressure loading on the rotor would be expected to occur in the tip region as the NGV tip passage vortex convects through. This is, however, not the case. Figs. 14.a-c clearly show that the largest unsteady fluctuations are present in the hub region. There are several reasons for this: 1) The radial pressure gradient forces the low total pressure fluid within the wake towards the hub increasing the wake depth with decreasing distance from the hub. 2) Due to the larger exit whirl at the NGV tip than at the hub the stream-wise distance from the NGV trailing edge to the rotor leading edge is larger at the tip than at the hub. As a result, the wake...
Fig. 11 Entropy contours for coupled NGV-rotor calculation at mid height. The flow field is displayed at time $t_0$. 
reaches a higher degree of mixing in the tip region than in the hub region. 3) The axial Mach number level is higher in the hub region than in the tip region and small changes in flow or blockage will have a larger impact on the pressure fluctuation. The character of the unsteady pressure fluctuations are very different at the hub and at the tip. In the tip region the fluctuations are limited to the region located upstream of 40% axial chord. The character of the fluctuations resemble incidence fluctuations caused by flow angle variations at the NGV wake boundaries. Based on the secondary velocity vectors in Fig. 13 the three-dimensional effects in the tip region appear to be small. In the hub region, however, the pressure fluctuations are more pronounced in the region of 50% to 70% axial chord affecting the throat Mach number and back surface diffusion level. This is a region which can have a major impact on the profile loss. The velocity vectors in Fig. 13 show strong radial flows outwards on the pressure surface of airfoil 2 and inwards on the suction surface of airfoil 2. This leads to an over acceleration on the suction surface at 50% chord of airfoil 2 (Fig. 14. b) leading to a lower surface pressure at 50% chord significantly below the static pressure at the throat and an increased levels of back surface diffusion. It is not clear from these calculations what the impact of these large scale amplitude pressure fluctuations is on the rotor boundary layer loss. Since, typically 60% of the profile loss is generated by the suction surface boundary layer downstream of the throat, it is reasonable to assume that the pressure fluctuations occurring in this region can be a major contributor to rotor loss significantly higher than would be the case under steady state conditions.

7. CONCLUSION
The detailed flow field in a first stage, low aspect ratio, LP turbine nozzle guide vane with strongly diverging annulus walls has been investigated in a two-stage rotating turbine rig operating with varying degrees of inlet whirl angle. Area traverse measurements upstream and downstream of the first stage NGV have been conducted in order to assess the impact of inlet whirl angle on the NGV total pressure loss and exit flow field. The large outer annulus wall slope gives rise to a strong tip passage vortex which affects the exit flow field between 50% and 100% height. The measurements are supported by 3D CFD predictions and show a strengthening of the passage vortices at hub and tip at positive incidence and a weakening at negative incidence. The pressure loss across the NGV derived from the area traverse measurements has been compared with an equivalent pressure loss derived from the isentropic turbine efficiency measurement. The NGV pressure loss exhibits an airfoil loss loop characteristic with minimum loss at design incidence and increasing loss at both positive and negative incidence. The turbine efficiency measurements, however, show a much different trend. The highest efficiency occurs at negative NGV incidence and decreases monotonically with increasing incidence. The efficiency measurements indicate a 75% increase in NGV loss at 10° positive incidence compared with the design point whereas only a 25% increase can be measured in the plane downstream of the NGV. This has lead us to believe that there is additional loss generation downstream of the low aspect ratio NGV due to the non-uniform flow field at the exit of the NGV. This additional
loss mechanism is caused by two factors; 1) mixing loss of the NGV wake and secondary flow in the downstream blade rows and 2) additional boundary layer loss in the downstream rotor due to unsteady pressure fluctuations. In order to study these effects further, 3D unsteady CFD calculations were conducted for the NGV and the downstream rotor. These calculations show the strongest interaction in the hub region of the rotor passage. There are several reasons for this: 1) The radial pressure gradient forces the low total pressure fluid within the wake towards the hub such that the wake depth increases towards the hub. 2) The NGV has larger exit swirl in the tip than in the hub which means that the stream-wise distance from the NGV trailing edge to the rotor leading edge is larger in the tip than in the hub, hence the wake reaches a higher degree of mixing in the tip region than in the hub region. 3) The axial Mach number level is higher in the hub region than in the tip and a small change in flow or blockage will have a larger impact on the pressure fluctuation. The pressure fluctuations in the hub region affect the back surface diffusion process and could therefore have a significant impact on the rotor loss. In striving for high annulus wall slope and smaller gap-to-chord ratio for the benefit of turbine cost and weight these types of effects are bound to become increasingly important. Parameters that could be used to optimize the configuration are for example; 1) the radial distribution of gap-to-chord ratio and 2) the rotor loading distribution that has to be designed such that the rotor loss characteristic is insensitive to the type of fluctuations that it will experience. The optimization process will inevitably become increasingly dependent on the availability advanced CFD prediction methods, such as presented in this paper, and the ability to routinely perform these predictions iteratively within the design cycle.

8. REFERENCES
Bridgeman, M., J., Cherry, D., G. and Pedersen, J., 1983, NASA/GE Energy efficient engine low pressure turbine scaled test vehicle performance report, NASA CR-168290.
Moore J., Moore J. G., 1985, Performance evaluation of linear turbine cascades using three-dimensional viscous flow calculations, J. Turbomachinery, Vol. 109, pp 258-267.
Lakshminarayana B., 1991, An assessment of computational fluid dynamics techniques in the analysis and design of turbomachinery. 990 Freeman Scholar Lecture, ASME J. Fluids Engg., Vol. 113, pp 315-352.
Schlachtriem S. and Lötzerich, M., 1997, Breakdown of tip leakage vortices in compressors at flow conditions close to stall, to be presented at the IGTI/ASME Turbo EXPO 97 in Orlando FL, USA.
Schulte, V. and Hodson, H. P., 1996, Unsteady wake-induced boundary layer transition in high lift LP turbines, ASME 96-GT-486.
Schulte V. and Hodson H., 1997, Prediction of the becalmed region for LP turbine profile design, to be presented at the IGTI/ASME Turbo EXPO 97 in Orlando FL, USA.
Cobley K., Siden G. and Amsted N., 1997, Design of a new three stage low pressure turbine for the BMW Rolls-Royce BR715 turbofan engine, to be presented at the IGTI/ASME Turbo EXPO 97 in Orlando FL, USA.
Lindner E., 1995, Numerical and experimental analysis of secondary flow in modern state-of-the-art low pressure guide vane rows, ASME paper 95-GT-169.

9. ACKNOWLEDGEMENT
The authors wish to thank BMW Rolls-Royce GmbH for permission to publish this paper. The authors also wish to acknowledge the constructive input from D. Joslyn of BMW Rolls-Royce GmbH for this paper.