STUDY OF RADIAL TIP CLEARANCE EFFECTS IN A LOW-SPEED AXIAL COMPRESSOR ROTOR

I.G. Nikolaou, K.C. Giannakoglou, K.D. Papailiou
Laboratory of Thermal Turbomachines
National Technical University of Athens
Athens, GREECE

ABSTRACT
A three-dimensional space marching code is used for the numerical modelling of the flow in an isolated axial flow compressor rotor. The rotor is analyzed at four operating points, up to near stall conditions. Numerical results are first validated versus available experimental data and then further exploited in order to illuminate flow patterns in the inter-blade region. The tip leakage impact on the main passage flow and losses level as well as the effect of blade loading on the hub corner stall extent and the radial displacement of the flow are fully detailed. In order to account for the rotor geometry, the modifications performed in an existing software are mainly concerned with the accurate modelling of the clearance which is formed above the curved blade tip; for this purpose, a local H-type mesh is embedded to the main passage grid.

INTRODUCTION
There are quite a few papers in the literature which report numerical or experimental procedures for the investigation of the tip leakage flows in compressor or turbine bladings (Pouagare and Delaunay, 1986, Dring et al, 1995, Foley and Ivey, 1994, Kang and Hirsch, 1994, Foley et al, 1993, Storer and Cumpsty, 1990, to name a few). Nevertheless, as long as the designers' efforts are directed towards the minimization of the induced losses, modelling attempts will be continuing. Efficient numerical methods along with the capabilities of modern supercomputers allow the numerical analysis of tip clearance related phenomena, with satisfactory accuracy in a reasonable computing time. They can be used, on a routine basis, at the design or analysis of modern turbomachines.

Three-dimensional Navier-Stokes codes are really capable to elucidate complex flow phenomena occurring in stationary or rotating turbomachinery rows, with finite tip clearance. The way tip gap is modelled and the selection of an appropriate turbulence model are important parameters which may affect the reliability of the obtained results. The spatial discretization

Presented at the International Gas Turbine and Aeroengine Congress & Exhibition
Birmingham, UK — June 10-13, 1996
within the tip region is usually carried out using embedded grids and it is discussed thoroughly in a following section.

In this work, a pressure correction code, incorporating the k-ε turbulence model and the wall function technique, is used for the detailed analysis of a low-speed isolated rotor. Despite its flaws, the wall function technique is used to accomplish the flow prediction in an affordable computational time, with the available computing means.

Experimental work on this rotor was carried out in the United Research Center's Large Scale Rotating Rig and measured data are available in the literature (Wagner et al., 1983). The rotor is analyzed by imposing very thick hub and tip inlet boundary layers. A parametric investigation of the role of the effect of changing the flow coefficient, on the flow patterns in the interblade region and the rotor exit, is carried out. The tip leakage flow, hub corner stall, the blade loading, the radial flow redistribution and the blockage of the flow are investigated. Experimental evidence is available for some of them while the present analysis aims at elucidating some others.

Lastly, the effect of the curved blade tip is numerically investigated, by re-examining the case using a new grid adapted to a blade of constant height and by comparing these results with those corresponding to the real blade.

THE NUMERICAL METHOD

The computational tool used in the present analysis is a 3-D finite-difference code for the solution of the Reynolds averaged Navier-Stokes equations. The physical model used is that of incompressible flows, without solving for the energy equation. Turbulence is modelled through the standard k-ε model and the wall function technique. Mean flow equations will be omitted, for the sake of brevity and only the transport equations solved for k and ε (Jones and Launder, 1972) will be given, namely

\[
\frac{\partial (\rho k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{\mu_s}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + \frac{\varepsilon}{\sigma_k} \rho
\]

and

\[
\frac{\partial (\rho \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{\mu_s}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + \sigma_\varepsilon (C_\varepsilon - 1) \frac{\varepsilon}{k}
\]

The constants used are all given in the Nomenclature. The k-ε model is not corrected to account for streamline curvature and Coriolis effects. Calculations performed using an existing model which modifies appropriately the turbulence generation term (as in Lapworth and Elder, 1988) provided almost identical results. All equations are written in the cartesian coordinate system, transformed to the computational one in the usual way and integrated over control volumes.

The numerical discretisation is based on a non-staggered, vertex centered scheme, by applying central differencing for the viscous terms and a quadratic upstream interpolation (QUICK scheme, Leonard, 1979) for the convection part in the mean flow equations. The convection terms in the turbulence equations are discretized using the hybrid scheme (Patankar, 1980). The continuity constraint is enforced through the pressure correction equation (SIMPLE, Patankar and Spalding, 1972) in the context of a space-marching algorithm. Detailed description of the method is given in Giannakoglou et al (1992) and Lymberopoulos et al (1993).

The space-marching character of the algorithm relies upon the assumption of a primary flow direction, from the inlet to the outlet of the field. Successive sweeps are performed along this direction, through a multi-pass algorithm. When sweeping over successive cross-stream grid surfaces, pseudo three-dimensional equations are solved for the mean-flow and turbulence equations. Upstream and downstream neighboring nodes are contributing to the equations solved through their more recent values and are explicitly handled (Tourlidakis, 1992). On each cross-stream plane, the Alternating Direction Implicit procedure (ADI) is used. Source terms split into two parts, one which is treated explicitly and the other which enhances the diagonal entries. Figure 1 illustrates the two types of the cross-stream planes, in the computational space, swept by the ADI scheme and lying upstream or downstream and between the leading and trailing edges; their shape is explained in the section dealing with the multi-block grid system. The implementation of the ADI scheme over these planes is very suitable, for it can be easily adapted to their form. The use of cartesian coordinates...
implies a particular treatment of periodicity conditions, not as straightforward as in a cylindrical reference system. Apart from the axial velocity component, the two others are coupled over the periodic nodes, in order to express the periodicity in terms of the radial and the pitchwise velocities.

In order to satisfy the mass conservation equation, a pressure correction Poisson equation is written on each cross plane. This equation contains extra entries resulting from the velocity-pressure coupling scheme (Rhie and Chow, 1983) aiming at suppressing pressure oscillations, common in collocated grids. A one-dimensional mass correction (Lapworth and Elder, 1988) is used to accelerate convergence.

DESCRIPTION OF THE ROTOR
The isolated axial flow compressor rotor has casing diameter equal to 1.524 m and hub-to-tip diameter ratio equal to 0.80. Experimental data are reported in Wagner et al., 1983, where the test rig is described. The rotor consists of 28 blades; the blade section airfoils have the thickness distribution of the NACA65 series and a circular arc camber line. The blade chord remains constant in the spanwise direction and equal to $C=0.1528$ m.

The tip clearance height varies with the axial distance, from the leading to the trailing edge. This is due to the variation of the blade height, while the casing has a cylindrical shape. The tip gap height at the leading edge is equal to 2.3%, it reduces to 1% at midchord, while at the trailing edge it has its maximum value equal to 3 percent span. The effect of the curved blade tip will be investigated separately, by comparing results obtained using the exact tip geometry with predictions corresponding to the straight tip case.

Our goal is to numerically investigate a class of flow problems occurring in the same rotor geometry, under different inlet conditions. With constant rotation speed $N=510$ RPM, four cases corresponding to flow coefficients $\Phi=0.65$, 0.75, 0.85 and 0.95 are analyzed. The tip height was kept the same in all these cases and, of course, the same grid was used. The computational results are compared with measurement data, when available and provide extra information on the flow patterns within the blading, where the measurements are insufficient. The $\Phi=0.85$ is the nominal design flow coefficient, while the $\Phi=0.65$ is at near-stall conditions. In all cases, the inlet velocity profiles include thick boundary layers at both hub and tip.

The inlet velocity profiles are axial as described in Wagner et al., 1983. These profiles, non-dimensionalized by their maximum value correspond to a unique curve, plotted in figure 2. The mean velocity is equal to the 84% of its maximum value. At the inlet, the displacement thicknesses are 7.8 (hub) and 7.9 (shroud) percent span and the momentum thicknesses are 5.3% span (constant at the hub and the shroud). The inlet turbulence intensity is equal to 0.02% and this determines the level of the turbulent kinetic energy. The inlet turbulent energy dissipation $e$ is uniform and corresponds to an eddy viscosity coefficient set equal to twenty times the molecular viscosity, in the absence of any experimental data. The axial flow at the inlet allows a constant static pressure to be imposed.

GRID GENERATION
For compressor rows with non-zero tip clearance, the use of pinched blade tip, has been widely used so far. This is numerically convenient (Tourlidakis and Elder, 1993), but leads to a poor analysis of the flow field for other than very thin blades (Kunz et al, 1992). Embedded H-type grids or combination of patched or overlapping O- and H-type grids, are more precise modelling techniques.

In the present analysis, the modelling of the (straight or curved) blade tip is achieved by combining two H-type grids. The clearance gap is filled using a local H-type grid which is blended with the main passage H-type grid. In spite of its
simplicity, the proposed meshing technique is well suited for compressor blades, due to their reduced thickness; in the compressor cases, complicated mesh systems, including C-, H- and O-type grids, like those used in turbine calculations with tip clearance (see for example Heider et al, 1993) are not necessary. The two domains used herein are illustrated in Figure 3. The tip gap domain forms a notional protrusion of the main passage one, entering the tip clearance region.

The task of grid generation is split into two parts. Firstly, the main grid is generated by stacking 2-D grids in the spanwise direction, with appropriate clustering near the hub, the shroud and the blade tip. The clearance grid is generated by linear interpolation between the surface grids corresponding to the notional upper extensions of the pressure and suction side of the blade. The main and clearance grids have the same grid density in the spanwise and through-flow directions. In the blade-to-blade direction, the clearance grid stretching can be specified independently of the main grid.

The H-type grid for the main passage of the rotor consists of 85x41x49 nodes and the H-type grid filling the tip gap region has 39x11x11 in the streamwise, pitchwise and spanwise directions, respectively. This gives a total number of nodes equal to approximately 175,000. A perspective view of some indicative grid surfaces is given in Figure 4.

ANALYSIS OF RESULTS

Exit Flow Distributions

As an overall measure of the predicted flow field, the circumferentially averaged spanwise distributions of the axial velocity component, the absolute pitchwise one and the relative flow angle β (denoted by "beta" in the figures), at the rotor exit, are presented in Figure 5. The grid surface located approximately at 30 percent chord downstream of the trailing edge, is christened "exit plane". The predicted distributions are compared with available measurement data; the accuracy of the measured flow angle was ± 1 deg; the measured pitchwise velocity distribution is not available, but this curve can be deduced by post-processing the measured quantities. For the examined flow coefficients, the agreement is very satisfactory, since predicted and measured distributions remain identical along the major part of the span.

The agreement can be seen to be very good close to midspan. For the lower flow coefficient, however, small discrepancies exist even in the core region. This can be related to the predicted pitchwise velocity values, since, in the same region, the axial velocity is in very good agreement with measurements. In all other cases, small discrepancies appear close to the endwalls. Close to the casing, the interaction of the tip clearance flow with the main passage one is still active, at this plane. This can be justified by plotting the same distributions slightly upstream or downstream of the location traced in Figure 5 (not shown here). Discrepancies are more pronounced close to the hub, which, in the case of maximum loading, are propagated up to the mid-span. Differences in the predicted flow angles at hub are visible, but the same differences appear in the circumferentially averaged one. Thus, the deficit in the blockage factor K (that is the 1-K quantity) is a measure of the deviation of the flow from axisymmetry. This deviation is usually attributed to different parameters, such as the tip leakage, the blade wakes and the secondary flows. In the present case, the effect of the hub corner stall is mainly reflected on K. The blockage factor was calculated either as suggested by Wagner et al, 1983, or through a direct integration of the calculated velocity profiles. In both cases, the K distributions were the same, within engineering accuracy.

For all flow coefficients, the radial distributions of the blockage factor deficit (1-K), at the same "exit" plane, are plotted in Figure 6 and compared with available measured data, Wagner et al, 1985b. A very low (1-K) value is observed far from the endwalls. There, K is too close to unit (K>0.98) and this is an indication of an almost axisymmetric flow in this region. The K-constant region becomes narrower and its location moves radially, as loading increases; this is in full agreement with previous remarks concerning the radial redistribution of the flow. In the Φ=0.65 case, the K-constant region is confined between 60% and 75% of span, while in the
The radial extent of the blockage in the hub region increases considerably as the flow coefficient decreases and, in this respect, predictions are in full agreement with measurements. On the other hand, in the measured distributions, the maximum (1-K) value in the hub region increases gradually, as long as $\Phi$ decreases. In the predicted distributions, the maximum K value at hub is approximately 90% and this remains almost constant, irrespective of the rotor loading.

In the tip region, differences in the extent of blockage are not so much influenced by the increased loading. The tip blockage varies from 15% (at $\Phi=0.95$) to 25% (at $\Phi=0.65$), in both prediction and experiment. Nevertheless, our prediction overestimates the blockage, as loading increases; such differences are not so clear in the experiment, even if K still decreases as $\Phi$ decreases. The tip leakage flow seems to be confined in a narrower region close to the casing, where the blockage is higher.

**Airfoil Pressure Distributions**

In figure 7, the static pressure distributions at five spanwise locations along the blade are shown, for all flow coefficients. Static pressure is normalized using the constant inlet static pressure and the dynamic head corresponding to the relative velocity at the inlet, at the same spanwise distance. The predicted pressure distributions are in good agreement with measurements, at least along the upper half of the blade. The agreement is also very satisfactory along the pressure side of the airfoil, at spanwise locations close to the hub. Along the last part of the suction side, that is for the last 40 percent chord approximately, differences between predicted and measured pressure occur. Predicted pressure distributions do not reproduce exactly the plateau in pressure, as in the experiments. This may be attributed to differences in the amount of backflow related to the hub corner stall. This will be more clear in the next paragraph, where the flow patterns close to the suction side of the blade are illustrated. At the two higher coefficients, a flow overspeed close to the leading edge, on the pressure surface, appears. This is in agreement with measurements and shows that, even if the thick inlet boundary layers do unload the blade midspan, off-optimum flow conditions occur at the design flow coefficient.

As the tip leakage flow is more pronounced in the $\Phi=0.65$ case, it is interesting to note that, even there, the agreement between predictions and measurements at the 95 percent span section is perfect. The excess pressure difference between the two sides of the blade, occurring along the first 25 percent of the chord, is a qualitative indication of the strength (and the location) of the leakage flow and is captured accurately. For the higher flow coefficients, pressure differences between pressure and suction side become milder and move downstream. As expected, in the $\Phi=0.65$ case, the leakage flow is aligned with the normal direction to the mean camber line.

In the nominal operating point $\Phi=0.85$, the midspan region is less loaded than the upper and the lower part of the blade. The same differences can be seen in lower flow coefficients, as well. At the $\Phi=0.65$ case, this unloading at midspan is the reason for not extending the hub corner stall in the full span.
region, close to the trailing edge.

Detailed Flow Patterns
Figure 9 illustrates constant relative velocity contours on three transverse planes, for the $\Phi=0.85$ and $\Phi=0.65$ cases. The three transverse grid planes are located at distances 35%, 65% and 103 percent axial chord from the leading edge; thus, the first two sections are within the blade passage, while the third one is slightly downstream of the trailing edge. The very thick endwall boundary layers at the inlet, previously shown in figure 2, cause strong secondary effects; the secondary flow interacts with the tip clearance one in regions which can be depicted as low dynamic energy regions in figure 9. As expected, the tip flow occupies a greater part of the span in the $\Phi=0.65$ case, influencing the pressure side of the adjacent blade, at the last part of the blade.

For all flow coefficients, the contours of the relative velocity, nondimensionalized the blade speed at midspan, are plotted in figure 10, at the exit plane lying at 30% axial chord downstream of the trailing edge. The corresponding experimental patterns (Wagner et al, 1993) are included in the same figure.

The predicted flow patterns appear to accurately match the measured ones. Two regions with small values, one near the hub at the suction side and another at the upper part of the span, between the two wakes are present. The first is mainly affected by the corner stall while the second is related to the tip leakage flow. Their size increases gradually as the flow coefficient decreases.

Figure 11 presents the velocity patterns along a grid surface at mid gap. The surface plotted is axisymmetric. Velocity vectors are formed by retaining the axial and the peripheral velocity component. In the $\Phi=0.85$ case, the effect of the tip leakage is confined within the suction surface of the blade. In the $\Phi=0.65$ case, the tip leakage flow is stronger, interacts with the secondary flow and creates a stagnation zone in between the pressure and the suction side.

In order to justify the comments made in the section dealing with the airfoil pressure distributions, the distribution of the tip leakage flow, crossing the notional extension of the suction side of the blade, is provided in figures 12 and 13. Figure 12 shows the nondimensionalized normal velocity crossing the aforementioned surface. Results for the case of uniform tip height is also shown, but this will be discussed later. The peak in normal velocity is moving downstream as the flow coefficient decreases. This is in full agreement with the location of the maximum pressure difference between suction and pressure surface. Figure 13 shows the accumulated mass flow through the gap. In the $\Phi=0.65$ case, the tip mass flux, normalized by the inlet flow rate, is (as expected) higher.

Pressure Losses
An overall estimation of the predicted losses can be obtained by plotting the spanwise distribution of the pressure loss coefficient, for the four examined flow coefficients. The pressure loss coefficient PLC, defined as

\[ PLC = \frac{\text{predicted loss}}{\text{inlet mass flux}} \]

is plotted in figure 14. Subscripts 1 and 2 indicate rotor inlet.

Flow Patterns along the Suction Side
In order to examine qualitatively the flow patterns at the suction side of the blade, which is likely to justify several previous remarks, velocity vectors over the first grid surface, adjacent to the suction side, are traced in figure 8. Without taking into account the exact curved shape of the analyzed grid surface, the velocity fields plotted in figures 8a to 8d consist of the calculated axial and radial velocity components. A qualitative comparison with flow visualization figures presented by Wagner et al, 1983, can be made. The visualization figures are omitted in the interest of space.

A first remark concerns the extent of the hub corner stall, as the flow coefficient changes. The radial extent of the stall region is in full agreement with the blockage distribution, shown in figure 6, already compared with experimental data. The radial flow motion which carries high loss fluid from the hub corner towards the casing, starts at 70 percent chord in the nominal conditions and much more upstream, namely at 30 percent, in the near stall case. In the $\Phi=0.65$ case, the radial migration of fluid stops approximately at midspan, where the blade is more unloaded.

In the grid surface which is analyzed, the strong back flow reported by Wagner, 1983, is not so clear. Also, unlike experiments, no important radial outflow is found at the tip.

The same unloading also influences the radial velocity distribution along the suction side. All these remarks are in full agreement with the available flow visualization.

FIG.(6): SPANWISE BLOCKAGE FACTOR DISTRIBUTIONS, AT THE "EXIT" PLANE. CONTINUOUS LINE=PREDICTION, SYMBOLS=EXPERIMENT.
FIG.(7): SPANWISE STATIC PRESSURE DISTRIBUTIONS.
FIG.(8): FLOW PATTERNS ALONG THE GRID SURFACE CLOSE TO THE SUCTION SIDE.

\[ PLC = \frac{P_{cr1} - P_{cr2}}{P_{cr1} - P_{ei}} \]

and outlet, respectively. In the \( \Phi = 0.85 \) case, the flow core which is characterized by lower losses is extended from 40% up to 80% span, while in the near stall region the same area is restricted between 65% and 75% span. In the tip region, changes in losses extent and level are in consistency with the variation in loading. In the hub region, the radial mass removal affects the spanwise distribution of losses. In the \( \Phi = 0.65 \) case, the impact of the extended hub corner stall on the distribution of losses is confined between 15 and 65 percent span.

Investigation of the Role of the Curved Blade Tip

As described in the section "Description of the rotor", the blade tip is curved, with its minimum value at midchord. In
FIG. (9): RELATIVE VELOCITY CONTOURS ON THREE TRANSVERSE PLANES, FOR $\Phi = 0.65$ (UPPER, INCREMENT = 1.75 m/s, $V = 28.2$ m/s) AND $\Phi = 0.85$ (LOWER, INCREMENT = 2.1 m/s, $V = 35.7$ m/s).

has influenced the axial velocity distribution in the full span. Only the part of the profile which is too close to the hub remains unchanged. It is worth noting that in the constant tip case, the pronounced tip flow phenomena result to a much weaker removal of the flow in the radial direction. The differences in the peripheral velocity are restricted in the area from 70% to 95% span. It is the same part in the b angle profile, illustrated in figure 15b, where greater differences appear. The changes in the distribution of the mass crossing the notional extension of the suction side of the blade, in the tip gap, by changing the blade height, have already been shown in figures 12 and 13. The total mass carried out by the tip clearance flow is about 30% greater than the one corresponding to the real shape of the blade. The normal velocity is, in this case, much more uniform.

CONCLUSIONS

A low-speed axial compressor rotor was analyzed using a pressure correction code and the k-ε turbulence model. An existing Navier-Stokes solver was modified in order to account for the exact shape of the flow domain, including a tip clearance of non-uniform height and a blade tip of finite thickness. By means of the modified calculation tool, a parametric investigation of the effect of the variation of the flow coefficient on the flow patterns in the interblade region and the rotor exit, was carried out. As a general remark, the Navier-Stokes code used proved capable to provide insight to the complex flow phenomena occurring in turbomachinery rows, with a finite tip-clearance. The tip clearance flow can be accurately captured, provided that the tip shape is accurately modelled. By modifying the blade tip shape and consequently the tip clearance height, results have been changed considerably. Important physical aspects of the examined case are captured. The hub corner stall exists in all examined flow coefficients. The stall as well as the radial migration of the flow along the suction side are much stronger as loading increases. In agreement with the experiments, the thick inlet boundary layers are the main reason for the blade unloading at midspan which inhibits the full-span flow separation. The peripheral blockage of the flow is the reason for the radial redistribution of the flow.

ACKNOWLEDGMENT

This work was funded by the European Community AC3A project AERO-CT92-0039, entitled "Advanced Civil Core Compressors Aerodynamics". The authors would like to thank all partners for their permission to publish this work. They are also indebted to Dr. A. Tourlidakis and Prof. R. Elder, Cranfield University, U.K., for their contribution during the early phases of the code development.

REFERENCES

FIG.(11): VELOCITY PATTERNS AT MID GAP (UPPER: \( \Phi = .65 \), LOWER: \( \Phi = .85 \)).


**FIG.(14): SPANWISE PRESSURE LOSS COEFFICIENT DISTRIBUTIONS.**

**FIG.(15): SPANWISE DISTRIBUTIONS OF THE CIRCUMFERENTIALLY AVERAGED FLOW QUANTITIES AT THE ROTOR EXIT.**