ABSTRACT

A Navier-Stokes solver is applied to investigate the 3D viscous flow in a low speed linear compressor cascade with tip clearance at design and off-design conditions with two different meshes. The algebraic turbulence model of Baldwin-Lomax is used for closure. Relative motion between the blades and wall is simulated for one flow coefficient. Comparisons with experimental data, including flow structure, static and total pressures, velocity profiles, secondary flows and vorticity, are presented for the stationary wall case. It is shown that the code predicts well the flow structure observed in experiments and shows the details of the tip leakage flow and the leading edge horseshoe vortex.

NOMENCLATURE

c = blade chord
C_p = static pressure coefficient
C_p = total pressure coefficient
I, J, K = indices in axial, pitchwise and spanwise
LE = leading edge
Lz = coordinate from tip vortex core to suction side
PS = pressure side
s = pitch
SS = suction side
TE = trailing edge
u, v, w = velocity components
u_a, v_a, w_a = averaged velocity
u_t = endwall moving speed
V_1 = inlet velocity

1. INTRODUCTION

Wind tunnel experiments are particularly important in discovering new phenomena or structures of the three-dimensional viscous flow in turbomachines. However, the detailed analysis of the highly 3D flow structures in the flow passages and tip clearances can benefit strongly from the support of reliable Navier-Stokes solvers. Hah (1986) and Perrin et al. (1992), by solving the Reynolds-averaged Navier-Stokes equations, predicted the overall effects of tip clearance in axial compressor rotors. Pouagare and Delaney (1986) using a SIMPLE-based algorithm reinforced some assumptions on compressor endwall flow phenomena, such as leading edge separation, leakage flow, separation line of tip leakage vortex, etc. The effects of Reynolds number on tip leakage flow in turbines were investigated by Moore et al. (1989) through laminar flow calculations for an idealized two-dimensional tip gap geometry. Choi (1992), by taking a much finer mesh inside the tip gap, captured some fine-scale leakage flow structures. Basson and Lakshimayan (1993) captured accurately the tip clearance flow in a turbine nozzle through optimization of grid size and artificial dissipation. Adamczyk et al. (1993) and Copenhaver et al (1993), with their numerical experiment, simulated well the interaction of the tip leakage vortex and the in-passage shock. Yet, many aspects, such as the nature of the 3D flow and energy loss inside the tip clearance, remain unclear.

The present authors have presented a series of experimental investigations on the 3D viscous flow in a low speed, large scale, linear compressor cascade with and without tip clearance, as referenced in Kang and Hirsch (1991, 1993a, b, c). In these investigations, some new flow phenomena were observed, such as a concentrated shed vortex in the wake and a multiple vortex structure around the blade tip in the cases with tip clearance. To confirm the experimental observations and to gain further insight into the complex three-dimensional flow phenomena, a 3D
Navier-Stokes solver, recently developed by Hirsch et al. (1991), has been applied to the compressor cascade under the experimental conditions with two different meshes. In this paper, the predicted data and their comparisons with the experimental results are reported for the case with tip clearance and stationary wall. The influence of relative motion between blades and the endwall is also studied through the numerical simulation.

2. NUMERICAL CALCULATION METHOD

Brief Description of the Code The Navier-Stokes code, developed by Hirsch et al. (1991), solves the time-dependent Reynolds averaged Navier-Stokes equations, with the algebraic turbulence model of Baldwin-Lomax or the two-equation k-ε turbulence model for closure. It is based on a multiblock/multigrid approach and incorporates various numerical schemes, based on either central or upwind discretization.

For the present calculation, the algebraic turbulence model of Baldwin-Lomax is selected. A first calculation was done on a coarse mesh, with a second-order upwind symmetric TVD scheme and Minmod limiter. However, further calculations were performed with a central scheme with second order dissipation coefficient set to 0, and the fourth order to 0.1. All the calculations were performed with a V-cycle multigrid technique. The numerical procedure applied a four-stage Runge-Kutta scheme with standard coefficients (1/4, 1/3, 1/2, 1) for the upwind scheme and a five-stage Runge-Kutta scheme with coefficients of (1/4, 1/6, 3/8, 1/2, and 1) for the central scheme. Incorporating implicit residual smoothing, the fine mesh calculations were performed at a CFL number of 7, requiring close to 300 interactions for a residual reduction of three order of magnitude. It has been confirmed in the cases without tip clearance that the accuracy of the converged solutions, such as pressure distribution computed with the upwind and central schemes, is well identical under the same mesh size.

Computational Grid The cascade blades are NACA 65-1810 with a chord of 20cm. Pitch and span of the cascade are respectively 18cm and 20cm, and the stagger angle is 10°. See Kang and Hirsch (1991) for the details of the cascade geometry.

Body-fitted grids with two calculation domains, shown in Fig.1, were made by the Interactive 3D Geometry Modeling and Grid Generation System (IGG) developed by Dener and Hirsch (1992), to represent the cascade geometry. The first domain with a H-H mesh is limited in the region with one pitch in tangential direction and half span in height, since the existence of the tip gap produces little influence on the midspan flow field based on the experimental observations in Kang and Hirsch (1993b). In the axial direction, the mesh extends from 1.5 chord upstream to 1.5 chord downstream. The second domain occupies the space in the tip gap with a O-H mesh.

Two different grids were calculated. One consists of 49x21x37, in the axial (I), blade to blade (J) and spanwise (K) directions, in domain 1 and 49x13x17 in domain 2, as shown in Figs.1 to 3. This mesh has 48,902 points and is called 'coarse mesh' hereafter. For increasing the grid points under the limitation of the computer (HP workstation), a fine mesh of 97x41x45 in domain 1 and 97x13x13 in domain 2 was created. This mesh has 195,358 points and is called 'fine mesh' hereafter. In both meshes, there are 13 O-lines leading to 25 points across the blade profile in the gap. Over the gap, however, there are 17 points for the coarse mesh and 13 points for the fine mesh. Hence, it was expected that the fine mesh would produce high accuracy inside the passage, but the coarse one would show more details of the gap flow.

Boundary Conditions In the inlet plane, three velocity components and static pressure are given. At this location
Fig. 4 Schematic of the flow pattern on the wall for 2.0% clearance, from Kang and Hirsch (1993b).

Fig. 5 Schematic of the vortex structure around the tip, from Kang and Hirsch (1993b).

The profiles of the pitchwise and axialwise velocity components employed in the calculation were calculated from the measured velocity (Kang and Hirsch, 1993b) at 40% chord upstream of the leading edge and from the flow angles around midspan. The pressure in the inlet plane is considered uniform and also taken from the measured data at midspan. Riemann boundary treatment is employed for the inlet boundary. Hence the real imposed parameters in the inlet are entropy (first characteristic variable), axial velocity (second characteristic variable), and pitchwise and spanwise velocity components.

No-slip boundary conditions are used on the blade surfaces and the stationary endwalls. Periodic conditions are enforced along the boundaries upstream and downstream of the passage, and also in the gap. A symmetric (or mirror) boundary condition is used at the midspan. In the outlet plane, uniform static pressure is imposed with the averaged midspan value, measured at 25% chord downstream.

3. CALCULATION CONDITIONS

In the calculation, only one clearance level with 4mm, or 2.0% chord, is run for two experimental inlet flow angles, β1=29.3° (design condition) and 36.5° (off-design condition). Other inlet flow conditions are the same as the experiments. The inlet flow velocity V1=23.5m/s, the Reynolds number, based on the velocity and blade chord, is about 3.0 x 10^5; The inlet boundary layer is turbulent with a shape factor of about 1.22 and a displacement thickness of 0.014 chord. The inlet Mach number was increased in the computations by 4 times for improving the convergence.

Although the blade surface boundary layer may have laminar and transition states, as observed in the experiments (Kang and Hirsch, 1991), the calculations will be undertaken with the supposition that all the boundary layers of the blade surfaces and endwall are in a turbulent state.

4. REVIEW OF THE FLOW STRUCTURE

Before discussing the calculated data, a brief review on the experimental observations of the 3D flow may be necessary. Figures 4 and 5 show the endwall flow pattern and the multiple vortex structure around the tip.

Around the tip gap, the flow in the pressure side corner will in general diverge along the line Lr in Fig.4. One part of the flow turns towards the suction side and joins with the passage vortex motion. The other part, however, passes through the tip gap towards the suction side corner in the adjacent passage and forms the tip leakage flow. Correspondingly, a separation line will be formed in the endwall flow pattern, as the line Lv in Fig.4. Due to the tip leakage flow, a multiple vortex structure, with three vortices, will appear around the tip. One is the tip leakage vortex (TLV) the other two small size vortices are the tip separation vortex (TSV) and the tip secondary vortex (SV) respectively, as shown in Fig.5.

With the rotation of the tip leakage vortex, the blade
sections close to the tip are strongly reloaded. The vortex core is quasi-circular after roll-up downstream of midchord, as observed from the contours of total pressure in the traverse sections and secondary velocity vector lines. Around the core the coefficient of total pressure loss reaches its higher value. The axial velocity profile along a line passing through the core shows a wake-like shape. Generation of the tip separation vortex makes the flow inside the tip gap very complex. The limiting streamlines on the blade tip surface normally exhibit a divergence pattern. The vortex increases in size and strength along the chord. Due to the contracted area of the tip gap, the fluid moving towards the gap is accelerated. This acceleration continues a short distance inside the gap, affected by the vortex motion. After the vortex, the wall pressure tends to recover with the leakage flow mixing. In the present blade geometry and flow conditions, the mixing process is not fully completed at the gap exit, especially behind the midchord, as the gap exit velocity shows a wake-like profile close to the tip. Near the blade leading edge, the wall static pressure inside the gap is normally uniform because the leakage flow here is dominated by the streamwise pressure gradient, and the gap exit flow is fully developed.

5. DISCUSSIONS ON COMPUTATIONAL DATA

Surface Flow Visualization At first, the predicted surface flow pattern, or limiting streamline patterns, are shown in Figs.6 and 7 at the off-design condition, compared with the experimental observations. Streamlines passing through the mesh points close to a solid wall are considered limiting streamlines in this study. It is seen from Fig.6b, calculated
with the coarse mesh, that the code predicts the endwall flow phenomena observed in the visualizations, such as the reattachment line in the pressure side and the separation line of the tip leakage vortex in the suction side. However, the predicted separation line $L_1$ in Fig.6b is closer to the midpassage downstream of the midchord than in the experiments. At the trailing edge, for example, the separation line in Fig.6a is located at about 26% pitch from the suction side, compared to 36% pitch in Fig.6b. In addition, the calculated streamlines under the blade are almost straight over the whole chord, there are no turns as in Fig.6a before midchord. No apparent difference was found from the endwall flow patterns between the fine and coarse mesh calculations.

Figure 7 presents the predicted limiting streamline patterns on the tip surface at the design and off-design condition, compared with the ink-trace visualization picture of the blade tip. It is seen that the main features of the flow, observed in the visualization, are reproduced in Fig.7b,c and d with both meshes. The reattachment line of the tip separation vortex coincides well with the experiments. It is located near the mean line of the blade profile and disappears at about 70% downstream of the leading edge in the coarse mesh case (more points over the tip gap). With the fine mesh (less points over the gap), however, the reattachment line is closer to the pressure side edge and disappears near the trailing edge.

With a decreasing inlet flow angle, this line (Fig.7d) tends to shift towards the pressure side, which implies a reduction of the tip separation vortex core. It is also observed from the calculations that the separation line of the tip separation vortex is not along the pressure side edge but a short distance inside the gap entrance.

Figure 8 shows the predicted flow structure around the tip and near the trailing edge with the coarse mesh. It includes the tip separation vortex, the tip leakage vortex; and the trailing edge separation vortex. The streamlines, indicated in the figure as tip leakage flow, pass through the midgap of the pressure side. However, the tip secondary

Fig.9 Comparisons of blade static pressures at midspan and close to the tip at off-design condition. $\Delta$: Exp., midspan; $\circ$: Exp.,1.5%span; $\cdot$- - : fine mesh cal., midspan; $-$: fine mesh cal., 1.5%span; $- - -$: coarse mesh cal., 1.5%span

Fig.10 Contours of static pressures on the endwall, calculated with a) coarse and b) fine meshes
vortex in Fig. 5 is not confirmed in this calculation. It will be seen later from the secondary flow plots that this vortex is well reproduced in the fine mesh calculation.

**Static Pressure Distributions.** Figure 9 shows the comparisons of the measured and calculated blade static pressure at midspan and close to the tip (1.5% span) at off-design conditions. It can be seen that at midspan, the calculation data are very comparable to the experimental data, but a large difference occurs close to the tip on the suction surface, on which strong reloading was observed around the midchord in the experimental data but not in the calculations. The difference may be related to the limitation of the simple algebraic turbulence model—Baldwin-Lomax model. This model, developed for 2D calculation, would underpredict the flow approaching separation (Stock and Haase, 1987) and the strength of the tip leakage vortex. The underprediction of the tip leakage vortex will in turn effect the reloading around the midchord, as it is tightly correlated with the vortex motion.

Comparing between the fine and coarse mesh computations, there is almost no difference in midspan pressures. But difference do exists close to the tip. The fine mesh predicts better the unloading in the leading edge region, even though the downstream reloading is still underpredicted. The difference of endwall static pressures between the fine and coarse mesh computation can also be found from Fig.10, which shows the static pressure isolines at the off-design condition. Due to the tip leakage vortex motion, the contours of wall static pressure just under the vortex core should exhibit a pressure trough, as shown by Inoue et al. (1991) and Moyle et al. (1992). It is seen clearly that the pressure trough is predicted with the fine mesh calculation but not with the coarse mesh. Under the blade, however, the pressure trough related to the tip separation vortex is well predicted in both meshes.

**Secondary Flows and Total Pressure Losses.** Figure 11 shows a comparison of the vector lines of secondary flow velocity in selected traverse (S3) planes, in which S3 plane No.2 is at the leading edge and S3 plane No.11 is located at 1% chord before the trailing edge. The secondary flow vectors are obtained by projecting the velocities on the plane normal to the flow direction at midspan at the same pitch coordinate. For presentation, the secondary flow vector is turned around the y-axis until it lies in the measuring plane, Kang and Hirsch (1991). The spiral nodes occurring in the planes denote respectively the horseshoe vortex (HV), passage vortex (PV), and tip leakage vortex (LTV). Figure 12a shows the contours of total pressure loss in the measured traverse planes from the leading edge to downstream of the exit for off-design conditions, compared to calculated total pressure losses, with the fine mesh, in the corresponding planes in Fig.12b.

Since the clearance allows a certain amount of fluid to pass through the gap, a leading edge horseshoe vortex may be weakened or even not occur, especially when the clearance is great. At present condition, i.e., 2.0% clearance, one can see a node of the suction side horseshoe vortex (HVs) from the experimental results (Fig.11a) in the suction

![Fig. 11 Comparison of secondary flow vector lines, a) experimental b) computational with coarse mesh, and c) computational with fine mesh at off-design condition](image-url)
Fig. 12 Comparison of total pressure loss contours, a) experimental and b) computational with fine mesh at off-design condition immediately inside the passage. The suction side leg, however, dissipates with its stretching into the passage due to both the fluid viscosity and the traverse pressure gradient, and completely disappears at about 11% chord (S3 plane No.3 or 1=21 in Fig.11b) downstream of the leading edge. The apparent rotation of the tip leakage vortex starts from about S3 plane No.5 (32% chord), which compares well with the experimental results. In the coarse mesh calculations, the tip leakage vortex (TLV) and the passage vortex (PV) are diffusing towards the cascade exit. However, the fine mesh calculations predict well the TLV and PV vortices downstream of the cascade. The secondary vortex (SV) around the suction side tip (Fig.5) deduced from experiments is well reproduced in the fine mesh calculation, as seen from Fig.11c and Fig.13 for details in the suction side corner of section I=51 (around midchord) and I=69 (near the trailing edge). This small size vortex starts before S3 No.5 and evolves towards downstream along the tip suction side. Near the trailing edge, it wraps round the tip leakage vortex (TLV). The interaction between the tip leakage vortex and the secondary vortex (SV) may cause some instability, since the convergence history of residuals indicates the appearance of some unsteadiness. The maximum and minimum residuals occur in the suction side corner before 30% c, where the TLV and SV start.

It is noticed that the spiral nodes, as appearing in Fig.11, can be constructed as a combination of a source and a free-vortex or a sink and a free-vortex. In the former, streamlines move outward from the nodes; in the latter, streamlines move inward towards the nodes. A more detailed discussion on this behavior goes beyond the range of the present paper.

In the total pressure contours (Fig.12), the generation and development of the tip leakage vortex can be seen from the high loss values in the suction side corner. It is seen that the isolines in the corner, predicted with the fine mesh (Fig.12b), compare well to the experimental ones (Fig.12a) and are very consistent with the rotation center of the tip vortex shown in Fig.11a and 11c. The predicted loss values in the tip vortex core are also consistent with the experiments, as in S3 plane No.16, where the maximum loss for calculation and experiments is respectively 0.92 and 0.85. In the wake, however, the predicted loss is much larger than the experimental data since the predicted boundary layers of the blade surface are higher than the experiments. These discrepancies might again require calculations with finer meshes or different turbulence models and/or suitable laminar-turbulent transition models.

Figure 14 shows the cross section vorticity plots of the fine mesh computation, compared with the experimental vorticities in the S3 plane No.15 (25% c behind the trailing edge). It is observed that the vorticity field is dominated by the tip leakage vortex (TLV).

Velocity Profiles Comparisons of the measured and calculated blade to blade profiles of axial velocity are made at each S3 plane near the blade tip (at y/l=0.03) and at midspan for 2.0% clearance at the design condition. As a selection, Figure 15 shows the profiles in three axial locations, i.e., S3 plane No.3 (11% c, 7 (55% c) and 11 (98% c). Figure 16 presents the spanwise velocity profiles at 3% span in S3 plane No.7 (55% c).

A good agreement can be found everywhere except for the region close to the suction surface where the tip leakage
Fig. 14 Comparison of vorticity vectors in a section of 25%c behind the trailing edge at off-design condition. a) experimental and b) computational with coarse mesh vortex core is present. In the suction side (Fig. 15), the profiles at 3% span close to the leading edge (S3 plane No.3) show similar distributions as midspan, since in this axial location the suction side horseshoe vortex has disappeared and the tip leakage vortex has not been generated yet (Fig 11). Downstream of S3 plane No.3, however, the profiles exhibit the tip leakage vortex core clearly, around which the axial velocity takes lower values (or wake-like profile) and the total pressure loss is high. Comparing the axial velocity profile around midchord (Fig. 15b), it is found that the tip leakage vortex, predicted with the coarse mesh, diffuses much faster than the experimental one and the fine mesh computational one. Approaching the trailing edge (Fig. 15c), the predicted core is closer to the suction surface than the experiments, especially for the calculation with the coarse mesh. Moreover, the vortex predicted with the coarse mesh is weaker than the experimental and the fine mesh one, as the amplitude of the spanwise velocity profile (Fig. 16) around the vortex core is smaller in the coarse mesh case. The spanwise velocity near the pressure side is always directed to the blade tip over the whole chord, i.e., it is negative in the present coordinate system. Approaching the blade pressure surface, the spanwise velocity increases rapidly and shows a very thin boundary layer on the pressure surface.

Figure 17 shows comparisons of the pitchwise distributions of axial velocity at 2%, 8.5%, 21% and 50% span in S3 plane No.14 (12.5%c behind the trailing edge) for design condition. In this figure, the coordinate z increases from the

Fig. 15 Comparison of blade to blade profiles of axial velocity near tip (3% span) and at midspan at three chordwise locations: a) 11%c, b) 55%c and c) 98%c for design condition: △: Exp., midspan; ○: Exp., 3% span; - - - - : fine mesh cal.; - - - : coarse mesh cal.
Fig. 16 Comparison of blade to blade profile of spanwise velocity near tip (3% span) at 55% c at design condition. o: exp., 3% span; —: fine mesh cal.; — — —: coarse mesh cal. suction side to the pressure side. It is seen that the prediction is reasonable over the pitch at all the spanwise locations. The wake and the core of the tip leakage vortex, both with high loss and low axial velocity values, are predicted at 8.5% and 21% height. However, the predicted vortex core is apparently shifted towards the suction side, as observed from Fig. 15 near the trailing edge. Near the wall at 2% span (Fig. 17a), the experimental data show the wake and the vortex, but in the prediction, these two regions have been mixed. The predicted wake around midspan is apparently sharper and thicker than the experiment, especially in the fine mesh calculation. This could result from the assumption of mirror symmetry at midspan in the computations.

Figure 18 presents the spanwise distributions of calculated axial velocity profiles, close to the centers of passage and tip leakage vortices, in S3 plane No. 14 (12.5% c behind the trailing edge) together with the section streamlines, calculated with the fine mesh, to indicate the location of the profiles. The experimental data closest to the locations 1 and 2 are also presented in Fig. 18. The wake-like shape in the profiles at section 1 around the tip leakage vortex center is observed in the calculation. The calculated position of the center, however, is closer to the endwall than

Fig. 17 Comparison of pitchwise distributions of axial velocity at a) 2%, b) 8.5%, c) 21% and d) 50% span in a section of 12.5% c behind the trailing edge at design condition. Δ: Exp; —: fine mesh cal.; — — —: coarse mesh cal.
Fig. 18 Comparison of axial velocity profiles along lines 1 and 2, close to the centers of tip leakage vortex (TLV) and passage vortex (PV), in a section at 12.5%c behind the trailing edge at design condition. o: Exp., section 1; Δ: Exp., section 2; ---: fine mesh cal.; --: coarse mesh cal.

The experimental data, which explains part of the discrepancies between the experimental and calculated profiles at section 1. The profiles close to the passage vortex (PV) do not show any wake-like shape neither but appear as normal boundary layers. Good agreement between the experimental and predicted data close to the passage vortex, or around the midpassage, can also be seen.

Tip Leakage Flow The tip leakage flow is mainly characterized by the gap inlet and exit velocities, and the static pressure distributions along the tip surface and the endwall under the blade. It is highly three-dimensional, as seen from the surface flow visualizations in Figs. 6 and 7.

Comparisons of the wall pressure distributions between the calculations and the experiments are presented in Fig. 19 for off-design flow conditions. In this figure, the z-axis extends from the pressure to suction side, with the zero point at the suction side. It is seen that, over most of the chord, the calculated data are reasonable. A large difference between the experimental and the calculated results occurs behind the midchord and under the blade profile, due to an underprediction of the mass flow through the tip gap, as seen late from Fig. 20. This may imply that the strength of the predicted tip separation vortices is weaker than the experimental, in connection with the underestimated blade loading near the tip (Fig. 9).
Comparisons of the calculated gap exit velocity with the experimental data are presented in Fig. 20 for the off-design condition, in which \( y=0.2m \) corresponds to the blade tip and \( y=0.204m \) to the casing wall. It is seen that the calculated data at all the sections are reasonable. It is to be recalled here that the 'coarse' mesh has 17 points over the gap, while the 'fine' mesh has 13 points over the gap. Although at 22\%c, Fig. 20a, the 'coarse mesh calculation is clearly more accurate, the remaining sections are less conclusive. In both calculations, the predicted boundary layers are much thicker than the experimental.

Figure 21 shows the chordwise distribution of the area-averaged gap inlet (solid line) and exit (dashed line) velocities and static pressures, a) resultant, b) pitchwise and c) spanwise velocities, and d) static pressure coefficient, calculated with coarse mesh at off-design condition. o: Exp., Kang and Hirsch (1993a)

Fig. 22 a) Area-averaged total pressure coefficient difference between gap inlet and exit, and b) area-averaged static pressure coefficient difference between gap exit and the maximum at the pressure side endwall, calculated with coarse mesh at off-design condition. o: Exp., Kang and Hirsch (1993a)
It is seen from Fig.21 that the resultant and pitchwise velocities achieve their maximum values and the static pressures achieve their minimum values around 20%c, where the leakage flow reads its maximum value, as can also be seen from Fig.22b. Compared to the inlet (pressure side) resultant velocity, the exit value (the suction side) is larger near the leading edge but is smaller over the other portion of chord. The spanwise velocities (Fig.21c) at both the inlet and exit are almost uniform except close to the leading and trailing edges. The large negative values in the inlet are due to the convergence of the passage flow towards the gap. At the exit, the nearly zero spanwise velocity indicates that the flow leaving the gap is similar to a jet. The averaged static pressure of the gap exit is similar to that of the blade suction surface near the tip (Fig.9) in both distribution profile and value. The pressure of the gap inlet, however, has been significantly reduced due to the flow acceleration. It is found that the difference of the averaged static pressures between the inlet and exit is quite small with maximum difference of about 10%. The difference of the inlet and the exit total pressure coefficients, or internal loss, is presented in Fig.22a. In figure 22b, the difference of the maximum static pressure in the pressure side endwall and the exit static pressures is presented. This maximum value should be normally close to the reattachment line $L_T$ in Fig.4. The difference of the exit and the maximum static pressure of the pressure side endwall are significant. Hence, it is concluded that, to balance the pressure at the suction side corner, or the gap exit, most of the pressure drop occurs in front of the gap inlet.

Influence of Relative Motion In real machines, one has to take into account the influence of the relative motion between wall and blade on the secondary flow field. A numerical simulation, which corresponds to a flow coefficient of 0.5 has been performed on the fine mesh. The flow condition imposed to the inlet boundary is the same as the design condition. Figure 23a and b show the secondary flow plots at the trailing edge for moving and stationary walls, respectively. It is seen that the moving wall pulls the tip leakage vortex significantly from the suction side corner to the pressure side corner and pushes the passage vortex far away from the wall towards the suction side.

To answer the questions raised by N. Cumpsty and E. Greitzer in the discussion of our previous paper, Kang and Hirsch (1993c), the present authors proposed a plot of tip vortex locus $L_T/s$ at rotor exit, against flow coefficient, to indicate how the relative motion effects the tip leakage vortex trajectory. The experimental data were from Inoue et al. (1991) and Moyle (1991). This plot is again shown here in Fig.24, where the points from the present calculations have been added. The present computations again show the strong influence of relative motion on the tip leakage vortex trajectory, and the present simulation appears to show the correct trend.

6. CONCLUSIONS

Comparisons of the results predicted with a 3D Navier-Stokes solver, with a Baldwin-Lomax turbulence model on two different mesh levels, with the experimental data are presented. It is seen that the code predicts well the very complex 3D flow in a linear compressor cascade with tip clearance. The tip separation vortex inside the gap deduced from ink-trace visualization on the tip surface in the previous experimental study is reproduced. Increasing grid
points greatly improves the predicted results. However, the tip leakage flow is still underestimated, which may be attributed to insufficient mesh resolution and/or to the simple turbulence model. Significant influence of relative motion on the tip leakage vortex is confirmed through a simulation under the same flow condition.

The present validations show that
- the overall 3D flow structure can be globally reproduced with a coarse mesh of close to 50,000 points, including the tip clearance gap.
- in order to capture the tip separation vortex, a minimum of 13 points over the clearance height appears necessary; earlier computation with 7 points did not allow to capture this vortex.
- more detailed comparisons with the experimental data, especially the viscous and turbulent dominated effects, requires finer meshes.
- the present fine mesh calculations, with close to 200,000 points, may still not be fine enough to ensure mesh independence and to reproduce quantitatively all the details of the flow.
- in addition to the above, the influence of turbulence modeling and/or transition effects should also be assessed. However, this can only be done, without ambiguity, on fine enough meshes.

REFERENCES


