THE USE OF ORTHOGONAL GRIDS IN TURBINE CFD COMPUTATIONS

Mark G. Turner, Te Liang, Phillip P. Beauchamp, Ian K. Jennions
GE Aircraft Engines
Cincinnati, OH 45215

ABSTRACT
For structured grid codes, the H type grid has become a standard, with deviations being used for exceptional circumstances. Among these circumstances are boundary layer or wake capturing, leading and trailing edge resolution and high grid shear. The latter of these is especially troublesome as some codes may converge with highly sheared grids to solutions which are inaccurate. In the case explored here, the trailing edge shock system of a high pressure nozzle guide vane is smeared over many grid cells using a conventional H grid, although better resolution can be obtained by careful attention to the grid. The paper demonstrates this on a very simple case and proposes the use of a grid that is nearly orthogonal to the flow to overcome the problem. The grid generation and treatment of periodic boundary conditions including a novel flux balance boundary condition across the mis-matched grids are discussed. Both two-dimensional and three-dimensional results are presented against test data to show the efficacy of this technique.

INTRODUCTION
Designers of first stage high pressure (HP) turbine vanes face many challenges. The vane accepts extremely hot air from the combustor and deflects it to a high exit angle for the rotor. The injection of a large amount of cooling air into the passage, to protect the vanes, further complicates the design. Lastly, the design must be completed while maximizing overall efficiency, not an easy task considering the efficiency levels already being achieved in today's engines.

In particular, single stage HP turbines reduce overall engine length and weight but require especially careful design. Inherent in the single stage design is a requirement for a high turning vane in order that sufficient power can be extracted from the single rotor, leading to transonic flow conditions and shocks from the vane trailing edges. These shocks have to be carefully controlled so that they do not seriously affect the downstream rotor and hence overall performance. Similar requirements have lead designers of space shuttle turbines to choose impulse type blading for compact designs with low weight (Griffin, 1992). Although not driven to the same extreme, primarily through efficiency considerations, aircraft engine HP turbine designers have the same sort of challenges. Exit angles from HP vanes are typically around 70°, with modern designs considering angles as high as 82°.

At these high exit angles, conventional turbomachinery H type grids such as those used by Dawes (1990) and Adamczyk et al. (1989) exhibit large amounts of shear which can affect flow solvers to the point at which the solutions are highly inaccurate. The exact extent of this problem will depend on the formulation of the solver and the skill of the user. New strategies have to be devised to reduce the shear and enable accurate solutions for this type of flow. The adaptive grid approach looks promising (Dawes, 1992 and Connell et al., 1993) but its application to three-dimensional (3D) Navier-Stokes solutions in an industrial design system is still some way off. Instead, new grid structures can be devised (Arnone and Stecco, 1991 and Hah, 1989) to allow the structured grid codes to continue to work for these problems.

The present paper demonstrates the problems encountered using an H grid on a simple 2D case, showing the effect of grid shear and aspect ratio on shock generation and propagation. A nearly orthogonal grid (here called an I grid) is then proposed, which overcomes the inadequacies of the H grid. The way in which these grids are generated...
and the boundary condition implementation are then discussed including a flux balance boundary condition across the mis-matched grids. A 2D case is then examined with I and H grids against test data to show that conclusions from the simple case do carry over to real geometries. Finally the use of I grids on two 3D geometries, for which limited test data exists, is reported.

**COMPUTATIONAL BACKGROUND**

The code used in this study is an extension of that reported by Turner and Jennions (1992), with its application to transonic fans being reported in an accompanying paper (Jennions and Turner, 1992). It represents an on-going concerted effort at General Electric Aircraft Engines (GEAE) to develop a comprehensive 3D Navier-Stokes code for turbomachinery. As the detailed numerics have been discussed in the first of these two papers, a brief description emphasizing the main attributes of the solver is presented in order that the extension of the method to I grid topologies may be understood.

The Reynolds averaged form of the full 3D Navier-Stokes equations including the energy equation written in Cartesian coordinates are solved. The flow is assumed to be compressible with adiabatic walls; supersonic and transonic solutions are allowed. The equations are cast in terms of absolute velocity but are solved in a relative non-Newtonian reference frame rotating with the blade. All that is needed to close this system of equations are models for the laminar and turbulent viscosity.

The laminar viscosity is modeled by Sutherland’s law and a choice of turbulence models is provided. The Baldwin-Lomax model (Baldwin and Lomax, 1978) implemented as a nearest wall model for three-dimensional flows and two forms of the $k - \varepsilon$ turbulence model, originally by Launder and Spalding (1974), are available. In order to achieve engineering solutions in acceptable times both models use optional wall functions which yield the shear stress and shear work terms at wall boundaries. All runs presented in this paper, except the simple case which is inviscid, use the $k - \varepsilon$ model with wall functions.

The equations of motion are integrated to produce a cell centered finite volume flux balance which, with the addition of the usual second and fourth order smoothing terms, are solved using the explicit Runge-Kutta scheme of Jameson et al. (1981). Various stage schemes are available in the code with the default being the 5 stage scheme described by Cedar and Holmes (1989) which has good dissipative properties for use with multigrid. This scheme worked well for transonic fans but for the high turning turbine cases addressed here, a 2 stage scheme with two evaluations of dissipation was used. This scheme is more robust than the 5 stage scheme, a property that seems to be needed to cope with the round trailing edges and high turning of these geometries. Velocity and temperature gradients are computed using auxiliary control volumes, an approach similar to that adopted by Kallinderis and Baron (1987). Local time stepping and multigrid are used to accelerate convergence. At present up to five levels of multigrid may be used, with a V or W cycle and optional sub-iteration. Residual averaging with constant coefficients (usually 1.0) is used to stabilize solutions rather than accelerate convergence. This was found to be necessary to converge fine grid cases and, as it adds very little computer time, is currently used as default. For a Runge-Kutta scheme with an odd number of partial steps the averaging is applied on the finest grid level and the odd steps only.

The $k - \varepsilon$ turbulence model is discretized about the same flux balance control volumes used by the explicit flow solver, with $k$ and $\varepsilon$ stored at cell centers. The resulting equations are solved implicitly using an ADI scheme on a cross flow plane. Alternatively, the algebraic equations used in Baldwin-Lomax are solved on a cross flow plane with reference to the nearest wall. In practice the viscosity is updated every time step of the flow solver, although the number of explicit time steps per turbulent viscosity update can be varied.

Except for the new periodic boundary condition treatment which is explained later, the boundary conditions are the same as those already described in previous papers. There are several downstream boundary condition options available. The option used for the solutions presented in this paper extrapolates the cross-stream gradients from the next to last plane to the exit plane and adjusts the level to match the circumferential average static pressure specified at a given spanwise location. Although this boundary condition is not totally non-reflective, it does allow the shock waves to exit the solution domain without causing spurious reflections.

Quality solutions depend on the control of numerical viscosity, i.e. an accurate differencing scheme and a good grid. Because the Navier-Stokes solver was extended from a well validated Euler solver, the amount of numerical viscosity produced by the code for inviscid applications has been monitored extensively for H grid cases. The extension to quality viscous solutions was therefore made knowing the underlying errors associated with a purely inviscid solution. The current extension to I grids is made with this background confidence in the numerical scheme.

**WHY DON’T H GRIDS WORK EFFICIENTLY?**

Figure 1a shows a portion of the grid for the flow in a duct with a small ramp (3°) on the lower wall. The actual grid (240x322 cells) extends many times the height of the duct upstream and downstream of the ramp. If we now introduce a supersonic ($M = 1.2$) inviscid flow into the duct, a shock, with subsequent expansion and reflection patterns, is produced as shown in Figure 1d. The pressure distribution along the lower surface of the duct is as shown in Figure 1g (0°) and is used with the Mach num-
ber contours to assess the accuracy of subsequent calculations. The predicted pressure rise is very close to the analytical result. However, the absolute accuracy of the solution does not matter for this demonstration, only how the subsequent changes in grid structure affect the solution from this base case.

If the channel is rotated by \( \alpha \), preserving the geometric dimensions of the duct, the grid shown in Figure 1b results. Here the cross-stream distance measured normal to each grid line running in the flow-wise direction has not changed from the grid in Figure 1a. Similarly, the distance between points along the lower boundary has not changed. What has changed is the amount of shear in the grid \( \alpha \) and the cell aspect ratio as measured by the ratio of the cell edge lengths in the cross-stream direction to those in the flow-wise direction, as shown in the Figures 1d-f schematics. It is trivial to show that the ratio of these lengths is the secant of the shear angle, \( \alpha \).

As the grid shear is increased the shock pattern resolution worsens, resulting in the dramatic difference between \( \alpha = 70^\circ \) and \( \alpha = 80^\circ \) shown in Figures 1e and 1f (rotated for ease of comparison). This is further shown by the pressure plotted along the duct lower boundary in Figure 1g where the \( \alpha = 80^\circ \) solution (although fully converged) exhibits very little resemblance to the base case. Figure 1h summarizes these experiences by plotting the peak static pressure predicted for the first shock for shear angles ranging from 0° to 80°. Depending on the desired accuracy, it would be easy to pick an angle past which solutions are unacceptable.

As defined above, the cell aspect ratio has changed at the same time as the grid shear has been introduced. If the aspect ratio is decreased by doubling the number of cross-stream grid points (the 240x64 grid solution), some of the shock pattern returns (Figure 1i). Further refinement could be performed, but it is apparent that the amount of grid needed to regain the base solution would be prohibitive, especially considering the large grid alternative proposed below. Finally, if the number of points had been doubled in the flow-wise direction (the 480x32 grid solution), a not altogether unreasonable choice, Figure 1i shows that no improvement would have been noted, the problem is clearly aspect ratio dependent.

A solution to the above problem is to make the grid as orthogonal to the flow as possible (Figure 1c). In this case the results from the I grid are identical to those from the unsheared grid for any value of \( \alpha \).

In summary, this simple case has shown how H grid solutions can deteriorate significantly as grid shear is increased. The effect can, to some extent, be ameliorated by increasing the number of grid points used, but a much better alternative exists. I grid solutions use less grid points, converge faster (e.g. twice as fast for the \( \alpha = 70^\circ \) case) and hence require less CPU time, a critical consideration when running 3D design cases.

### GRID GENERATION

The grid generation algorithm used is a modified form of the 2D NASA GRAPE program, developed by Sorensen (1980). The alterations maintain the basic core solver integrity while changing the general input and boundary setup routines to new specific routines which build a volume grid for turbomachinery applications using an 'onion peel' approach. In addition, the program has an optional output format which allows a Quasi-3D grid to be developed. In this case a volume grid consisting of a single cell in the spanwise direction, whose height is adjusted to the lamina thickness predicted by a throughflow analysis, is generated. Although it lacks generality, the program requires few user inputs, making it more appropriate in an industrial design setting.

The standard grid developed by the algorithm is a single block H grid, in which the grid boundaries describe the passage between two adjacent blades and have upstream and downstream periodic extensions. Using the 'onion peel' approach a series of blade-to-blade grids, each developed on a surface of revolution defined by an arbitrary meridional cut, are stacked up to form the full 3D grid. The process starts with a hub conforming surface and proceeds upward through each spanwise surface of revolution to the casing conforming surface. The spanwise distribution of surfaces is controlled by inputs to a pre-processing algorithm which produces grids like those shown in Figures 5c and 5d. For each blade-to-blade surface the inlet and exit boundaries are restricted to either a constant axial value for axial flow boundaries or a constant radius for radial flow boundaries. This ensures that the inlet and exit boundaries of the volume grid are also surfaces of revolution. In establishing the blade-to-blade H grid boundaries, the designer has global control of the number and distribution of points along the blade surface and periodic extensions. Additionally, the designer may control the presentation angle (the angle on the surface of revolution between the blade-to-blade and throughflow lines) at both inlet and exit boundaries.

The alternative grid developed by the algorithm is a single block I grid. The mechanics of building a sequence of blade-to-blade I grids into a volume grid are identical to that described for H grids. The basic difference lies in how the boundaries of the grid are established. Figure 1c illustrates one type of I grid. The blade surfaces form part of the domain boundary which complicates indexing and the inlet and exit are not at a constant axial location which complicates the boundary condition treatment. The alternative I grid is shown in Figure 3a and places the blade in the center of the passage, something that is dealt with using triple grid lines (Cedar and Holmes, 1989) in the current flow solver. Because this feature already existed, this alternative form of I grid was chosen in the current work. The only flow solver coding specific to I grids was the implementation of interpolation along periodic bound-
The blade-to-blade grids for the I grid problem are created by assembling two surface of revolution sub-grids into a single grid which extends from mid-passage to mid-passage. For each surface of revolution a sparse H grid problem is iterated a few times and the mid-passage line is extracted and saved as the I grid periodic boundary. The desired grid is then solved in two parts using sub-grids extending from the pressure surface to the periodic line and from the suction surface to the periodic line. Along the periodic line the point distribution for each sub-grid is independently allowed to develop in such a way that the blade-to-blade lines are orthogonal to the periodic line. This is accomplished by assuming the grid is locally orthogonal and subject to the Cauchy-Riemann conditions.

The discretization uses central differencing of the resulting Laplace equation for the axial point locations using a 5 point stencil centered on the periodic boundary. The fictitious point in this stencil is obtained by central differencing the first of the Cauchy-Riemann conditions. A single pass Gauss-Seidel like method is performed along the boundary to obtain the new axial distribution while the corresponding blade-to-blade values are determined by interpolating the new axial values against the pre-determined periodic line.

In the early iterations this scheme can result in a non-monotonic point distribution. If this occurs the ill-conditioned points are adjusted to force a monotonic distribution. The scheme is applied to each sub-grid independently with no restrictions that points on the periodic line be aligned. A close examination of the resulting grid along the mis-matched periodic boundary would show that in areas where multiple cells on one side abut a single cell on the other side there exists small voids in the grid (Figure 2a). Although the voids cannot be completely removed, the problem can be minimized by forcing the edge points on the multi-cell side to exactly intersect the straight line segments forming the cell face on the single grid side as shown in Figure 2b. Once the two sub-grids are solved and adjusted they are re-assembled into a single block grid extending from mid-passage to mid-passage with the blade treated using the triple line concept.

The inlet and exit boundaries at a given spanwise location are at a constant z (or r for a radial inflow or outflow). Otherwise the inlet and exit boundary conditions, especially in 3D, would be very difficult to apply. An orthogonality condition is being applied along the mid passage line, which is consistent with the inlet and exit boundary only if the grid angle is turned to axial. If this is not done, the grid mis-match becomes excessive. However the turning to axial must be done gradually or the high curvatures pull on the grid too much. Also this means that the inlet and exit boundary may have to be placed further upstream or downstream than otherwise desired.

**PERIODIC BOUNDARY CONDITION TREATMENT**

The major modification to the code to allow for I grids has been the implementation of the periodic boundary condition for mis-matched grids. Figure 2c shows a schematic of the mis-matched grid along the periodic boundary of the I grid. A very simple linear treatment is applied such that \( \psi_r \), a dependent variable at the “phantom” boundary cell, is linearly interpolated from \( u_1 \) and \( u_2 \) along the first line of interior boundary cells. The interpolation is based on the axial coordinate \( (z) \), but could be a meridional or radial coordinate if centrifugal or radial geometries are to be solved using an I grid. Also this implementation assumes no spanwise interpolation is required (i.e. that each grid surface in the blade-to-blade direction is a surface of revolution). The interpolation is applied to all the principal flow variables and at each multigrid level.

Conservation of mass or other fluxes is not guaranteed by this linear interpolation boundary condition. An option also exists to correct for the flux imbalance by linearly interpolating the cumulative flux on each boundary and applying the average of the boundary fluxes to each face. This is actually accomplished by calculating a correction which is half the interpolated flux on the opposite side minus half the calculated flux and applying this to the residual. This is applied along each boundary for the mass flux, the three momentum fluxes, the energy flux, the five smoothing fluxes, and the four viscous fluxes. This correction is only applied at the fine grid.

The flux balance treatment has been a recent option added to the code. For some time, the linear interpolation option has been used by aero designers at GEAE for analyzing 2D and 3D designs, and even though it is simple, it had worked quite well. This is demonstrated by the 2D results presented in the next section. Both options have been used to analyze the 3D results presented later in the paper. Although the results presented are from the flux balance option, there is very little difference in the solutions for these cases. Even though the differences are small for these cases, other cases show major differences and the mass imbalance and entropy production along the periodic boundary are improved considerably when the flux balance option is used.

Other geometries might also benefit from the use of the flux balance option. The transonic fan solutions presented by Jennions and Turner (1992) were run using an H grid. At the tip, these blades have high stagger angles and the H grids have a lot of skew. The added orthogonality of the I grid should produce more accurate results, and the flux balance would ensure a fully conservative solution.

**VKI-LS82, 2D TURBINE NOZZLE**

A Von Karman Institute (VKI) gas turbine nozzle guide vane (VKI-LS82) presented at a VKI Workshop (1982) was chosen to demonstrate the relative capabilities of H versus
I grid on a real 2D geometry. This vane has a leaving angle around 80° and test data, including surface static pressure measurements, are available for comparison.

The entire 2D I grid, with 48 cells in the blade-to-blade direction and 224 cells in the flow-wise direction, is shown in Figure 3a (where only every fourth line is shown for clarity) with an enlargement of the throat region (for the full grid) in Figure 3b. The grid is reasonably orthogonal, given that a good number of points are required around the trailing edge and the grid must be pulled into this region by the elliptic solver in order to maintain good definition. The blade is positioned in the center of the grid with the periodic lines, across which there is a grid mis-match, running between two adjacent blades. The grid shape (Figure 3a) mimics the axial inlet of the flow and the turning of the flow through the vane, while being gradually turned back to axial from the trailing edge.

The resulting isentropic Mach numbers from both test and computation are shown in Figure 3d for a variety of exit isentropic Mach numbers, ranging from subsonic to supersonic. The two subsonic conditions are well predicted, including a slight overspeed on the suction surface that develops at \( x/c = 0.27 \) for the \( M_{2,\text{sen}} = 0.70 \) case and \( x/c = 0.29 \) for the \( M_{2,\text{sen}} = 0.85 \) case. As the exit flow becomes supersonic the effect of the pressure surface trailing edge flow is felt on the vane suction surface. Figure 3c shows computed interferometry pictures for these three cases and Figure 3e shows an enlargement of the trailing edge region for the \( M_{2,\text{sen}} = 1.43 \) case. Interferograms are contours of constant density using an alternating grey scale which clearly show shock and wake action. They are used in experimental visualization and make the assimilation of complex phenomena easier. On the pressure surface, the flow rapidly accelerates as the effect of the round trailing edge is felt. It then encounters the flow from the suction surface and is deflected, producing the pressure surface leg of the classic fish-tailed shock system characteristic of this type of flow. The suction surface feels the effects of this pressure surface trailing edge flow by first accelerating in response to the expansion and then rapidly decelerating in response to the pressure surface shock. These features can readily be seen in both the experimental and computed profiles shown in Figure 3d. It appears that the \( M_{2,\text{sen}} = 1.12 \) data point at \( x/c = 0.4 \) may well be in error as it does not track the trend being set by the rest of the data.

To complement the simple example of grid shear presented previously, a study of the effects of H versus I grid on this geometry has been performed. An H grid, conforming to rules on aspect ratio found from the duct case, and using the same number of cells as the I grid was generated by using the elliptic solver previously described. Figure 4a shows the same portion of this grid as Figure 3b did for the I grid. The H grid shown has one quarter the number of points in each direction for clarity, the difference in shear being self evident. The solution obtained using this grid is shown in Figure 4b, and rivals the I grid result. The suction surface peak Mach number is not quite as high as that from the I grid, but the pressure surface trailing edge expansion and shock are being correctly generated and transmitted by the H grid. For this case, to obtain a solution with the H grid the CFL number had to be dropped by 30% in order to obtain a stable solution. Even with this, the convergence history was not as smooth as the I grid, demonstrating that I grids are easier to run and converge than the highly sheared H grids.

In order to demonstrate grid dependence, the number of blade-to-blade points for both the I grid and H grid was halved. The isentropic surface Mach number is cross-plotted on Figure 4b and contours of absolute Mach number for both calculations are shown in Figure 4c and 4d. The H grid is now failing to pick up and transmit the pressure surface shock system while the I grid results are remarkably unaffected by the halving of the grid. It seems for this case that the initially chosen I grid was far denser than it needed to have been to capture the relevant physics. These results indicate that the cost of running an I grid calculation would be significantly cheaper than its H grid counterpart.

LAR and ATNA 3D Turbine Vanes

A major problem with finding test data for 3D HP blading is that normal industry tests include cooling air, while the tests run in universities tend to be two-dimensional. One common way of testing the individual effects of cooling air in industry is to test in a ‘pizza pie’ configuration. Here a number of different cooling configurations are tested simultaneously, the annulus being split into a number of sectors with each sector having a cooling arrangement which is different from its neighbor. In this way, the effects of band cooling (cooling from the hub and casing surfaces) can be separated from blade surface cooling and they both can be separated from the pure aerodynamics by completely blocking the holes in one sector. As with all compromises, this type of testing does not supply all the data that would be required to validate a Navier-Stokes code, e.g. the mass flow rate cannot be measured in individual sectors and therefore is not known. With this in mind simulations of two vanes will now be presented and compared to the available test data.

The two vanes have leaving angles approaching 80°. The first is the LAR (Low Aspect Ratio) vane, which is representative of vanes in production use. The second is the ATNA (Advanced Turbine Nozzle Aerodynamics) vane representative of an advanced 3D vane. LAR was designed and funded by GEAE, while ATNA was a joint program between the Navy and GEAE. The nozzles were both run in the same facility, a schematic of which is shown in Figure 5a. The schematic shows the LAR configuration with the downstream probe retracted and assembly of the vane. The ATNA configuration, complete with contoured endwalls, is simply bolted into the same test hardware.
The vanes and flowpath, along with the corresponding grids used are shown in Figures 5b-d. A grid using 36 cells in the spanwise direction, 42 cells in the blade-to-blade direction and 128 cells in the flow direction (195,536 total cells) was used for both vanes, an I grid of quality similar to that shown for the 2D cases being used in the blade-to-blade direction. The axial grid is coarser than that used in the 2D cases but is typical of that used by designers on a day-to-day basis, when capturing shocks while achieving low run times is important. As cooling air is not modeled in the computation, a uniform distribution of total temperature was assumed at the inlet. Both the dilution of total temperature by the cooling air and the heat transfer effect of the cool vane in the relatively hot stream are not accounted for in the computation.

The test and computed results in terms of contours of total pressure at the traverse plane are shown in Figure 6. The view is aft looking forward and two complete passages are shown. In both cases the test results presented are with the band cooling holes blocked, leaving the blade surface holes to flow. Several observations are striking from the test data. The LAB vane is not truly radial and the endwall boundary layers in both cases, which have to be extrapolated close to the endwalls, are remarkably thick. The cause of such behavior has yet to be found.

In the predictions the LAB wake is radial, in agreement with the trailing edge geometry of the vane. There is also an overshoot in total pressure on the pressure surface side of the LAB vane. This type of overshoot has been related to three separate mechanisms: a) the presence of a high turbulent viscosity gradient, b) dispersion errors as the thin pressure surface boundary layer sees the suction surface boundary layer and c) fourth order smoothing effects. The first of these three mechanisms seems to dominate in this case and while crude fixes of the turbulence model in the wake have considerably improved the solution, a more elegant general formulation of the turbulence model for the wake region is the subject of further work. The computed ATNA wake follows the data, thickening significantly towards 40% span. The thick experimental endwall boundary layers are again present in this case, indicating that it may be rig related rather than the blade geometry.

The endwall boundary layer thickness is also apparent in Figure 7 which presents the circumferentially averaged total pressure profiles for both vanes. The design intent with ATNA was to demonstrate that high total pressure fluid could be moved to the endwall region, an intent that was aided by the inviscid 3D code (Holmes and Tong, 1985) which was the building block for the current Navier-Stokes code. It is clear that this objective has been achieved, as both the test and the computed results show high total pressure fluid in the endwall region. This goal has been achieved, however, at the expense of the mid-span total pressure, a 'hole' appearing around 15%-40% span, corresponding to the wake thickening seen in the contour plots.

The LAR vane is relatively unexciting and is predicted to be so. Total pressure is almost constant across most of the span with the profiles dropping off at the endwalls. There is predicted to be a small area of separation on the suction surface near the hub (Figure 8) which can be seen to influencing the total pressure contours in Figure 6b.

Mach number contours on both the cross-stream traverse plane and the mid-span blade-to-blade plane are shown in Figure 9. The shocks and wakes for both vanes with the affects of separation for LAB are evident. There is virtually no pressure surface shock for LAB and a weak expansion/shock system exists for ATNA. The clear distinction between supersonic and subsonic fluid across the shock in the case of the LAB result is distorted in the case of the ATNA result. The ATNA shock appears weaker and the sonic line is located further up the span. Again the effect of the curved wake is noted in the ATNA results, as good (high total pressure) fluid is trapped between the curving wake and the endwall surfaces. Overall the effect of lean, which is very different between the two geometries, is being predicted by the code.

CONCLUDING REMARKS

This paper has demonstrated the extension of a 3D Navier-Stokes structured grid solver, used on a daily basis in industry, from H grids to a more orthogonal I grid arrangement. A novel flux balance boundary condition has been implemented across the mis-matched periodic boundary. Simple cases to demonstrate the problems with H grid have been shown, along with 2D and 3D cases against experimental data to prove the efficacy of this technique.

ACKNOWLEDGEMENTS

The authors would like to thank Yen-Ming Chen for his running of the simple duct case and the Navy for permission to publish the ATNA results. We would also like to thank GE Aircraft Engines for permission to publish this paper.

REFERENCES


Figure 1: SIMPLE DUCT EXAMPLE
i) Grid refinement effects

Pinlet

h) First pressure rise versus shear angle

Figure 1: SIMPLE DUCT EXAMPLE (CONT.)
Figure 2: GRID PERIODIC BOUNDARY
Figure 3: VKI-LS82 I GRID RESULTS

a) I grid (every fourth line shown)

b) Enlargement of throat region (full grid)

c) Computed interferograms (density contours using an alternating grey scale)

d) Loading comparison

e) Mach number contours for $M_{2,isen}=1.43$ (Contour increment is 0.05)
Figure 4: DEREFINEMENT STUDY ON VKI-LS82 AT $M_{\text{inlet}}=1.43$

Mach number contours for the coarse grids (24 blade-to-blade cells)
Contour interval is 0.05
a) Rig schematic

b) Vane Hardware

c) LAR grid and flowpath

d) ATNA grid and flowpath

Figure 5: LAR AND ATNA GEOMETRIES
Figure 6: CROSS-STREAM TOTAL PRESSURE CONTOURS AT THE TRAVERSE PLANE – CONTOUR INTERVALS ARE 0.5 psi
Figure 7: RADIAL TOTAL PRESSURE PROFILES

Figure 8: LAR SUCTION SURFACE SEPARATION NEAR HUB
Figure 9: COMPUTED MACH NUMBER CONTOURS – CONTOUR INTERVALS ARE 0.05