Numerical Simulation of Unsteady Viscous Compressible Flows Applied to Blade Flutter Analysis

L. D. GUNNAR SIDÉN
Aero/Thermodynamics
Volvo Flygmotor AB
Trollhättan, Sweden

ABSTRACT

A numerical method for simulating quasi three-dimensional unsteady viscous compressible flow is developed and applied to blade flutter analysis. The Reynolds-averaged Navier-Stokes equations are solved in a time accurate manner on a continuously deforming computational mesh. Turbulence is accounted for by the inclusion of a two-layer algebraic turbulence model. This method is compared with measurements and a full velocity potential solver for two different subsonic compressor cascades and a range of flow conditions. Both methods are found to predict unsteady attached flows reasonably well. However, the Navier-Stokes solver picks up the pressure fluctuations associated with the unsteady leading edge separation bubble. These types of fluctuations have large amplitudes and tend to dominate the cascade's aerodynamic damping behavior.

INTRODUCTION

Flutter in turbomachines

The trend of turbojet aeroengine fans and compressors is towards larger mass flows and more slender blade profiles. This development increases the demand for accurate flutter prediction methods since the occurrence of flutter is traditionally assumed to be more likely at high speed flows, at off-design incidences and for thin blades. Compressor flutter is commonly classified into four groups (Carta, 1986). Stall flutter (subsonic or supersonic), which may occur at operation near the surge line, is probably one of the most difficult aerelastic phenomena to predict. For this type of flutter viscous effects are of great importance due to the strong adverse pressure gradients through the blade passage that lead to thick turbulent boundary layers, often combined with laminar separation bubbles at the leading edge and turbulent separations near the trailing edge. The other two risk zones are at low pressure ratio (choke flutter) and at high mass flow (supersonic uninstalled flutter).

The work in this paper is based on three assumptions: a two-dimensional (or quasi three-dimensional) blade to blade geometry is considered; the analysis is isolated to one blade row; the mechanical system is assumed to be very stiff. The reason for the first two assumptions is mainly due to the computational cost. The methodology is not, in principle, limited to two-dimensional flow, but it may be expanded to three dimensions including effects due to rotation, tip clearance and shrouds. The second assumption considers the issue of upstream and downstream boundary conditions, which here are assumed to be steady and known. As a consequence of the last assumption the flutter frequencies are identical to the resonance frequencies of the structural system without any unsteady aerodynamic forces acting on it. By making these three assumptions the problem reduces to determining the unsteady pressure force and moment acting on a blade, or a set of blades, whose motion is known. The non-dimensionalized net work input is obtained by integrating the product of the pressure force and the displacement of the blade during one flutter period.

\[ \delta_{\text{aero}} = \frac{1}{A_0^2} \int_0^T \int \sigma_p (v \cdot n) ds dt \] (1)

Here, \( A_0 \), \( \sigma_p \), \( v \) and \( n \) are the blade amplitude, surface pressure coefficient, local blade velocity and the unit normal on the blade surface directed outwards. The surface integral is evaluated around the blade and the time integral is evaluated for one flutter period. This work is called the aerodynamic damping and may be negative as well as positive. If the sum of the mechanical damping and the aerodynamic damping is negative then flutter will occur.

Previous work

Several techniques have previously been employed for the computation of the aerodynamic forces on a fluttering blade cascade. Whitehead (1982) and Verdon (1982) assume the flow to be adiabatic, reversible and irrotational, so the equations are those for a velocity potential. The potential is continuous except for a jump across the wake. The unsteady solution is obtained by assuming a small perturbation around the steady mean flow such that the equations may locally be linearized. This technique is especially attractive since a harmonic response can be assumed and, hence, no time marching is necessary. A complex potential, that carries the information about the velocity potential's magnitude and phase, is solved for at each node point in the computational mesh. This method has been further refined by Cedar and Stow (1985) to account for boundary layer blockage and the quasi three-dimensional effect of a varying streamtube thickness. The assumption of locally linear equations, however, does not make the method suitable for flows involving strong shocks and large blade amplitudes. Other authors (Joubert, 1984; Böles et al., 1987; Gerolymos, 1988 and He, 1989) solve the Euler equations by time marching methods instead. This completely removes the constraint of linearity but requires at least an order of magnitude more computer time and computer.
storage. In the first part of the analysis the steady state solution is sought by supressing the blade motion and marching forward in 'pseudo time' until the residual is small enough to be neglected (the time step is locally optimized to converge as fast as possible). The steady state solution is then used as an initial condition for the unsteady analysis, in which the Euler equations are solved in a time accurate manner. The unsteady forcing is due to the moving solid blade boundaries. In order to simulate the blade motion the computational mesh has to be deformed such that the computational blade boundary at all points in time coincides with the actual physical blade boundary. This is done by deforming the mesh in the vicinity of the blade. To account for this mesh motion extra terms arise in the governing equations. The simulation of non-zero interblade phase angles is handled by storing the computed flow variables at the periodic boundary nodes and applying the stored quantities at a later time on the opposite periodic boundary. In this way the computational domain only has to include one blade passage. The inviscid methods neglect viscous losses, although the effect of boundary layer blockage can be accounted for by performing a coupled integral boundary layer calculation.

**Present work**

In this paper the Reynolds averaged Navier-Stokes equations are solved and turbulence is accounted for by a two-layer algebraic eddy-viscosity model. This approach is especially suited for compressor simulations, which are commonly characterized by strong interactions between the viscous and inviscid regions of the flow. The geometric flexibility of the Taylor-Galerkin finite element method (Sidén et al., 1990; Morgan and Peraire, 1987) is exploited on an outer unstructured mesh and the computational benefits of the factorized implicit scheme by Beam and Warming (1977) are utilized on an inner boundary fitted mesh of C-type. The interface boundary between the two domains is handled by overlapping the two boundaries. This coupled technique has been shown useful, especially for time accurate simulations which require a uniform time step to be used. Fully implicit methods are often unnecessarily expensive since a large portion of the domain (far from the blade) consists of large elements and, hence, does not need to be treated implicitly. On the other hand, when using an entirely explicit scheme the smallest element in the mesh, probably located in the boundary layer region, limits the maximum allowed time step in accordance with a stability criterion of the following form:

$$CFL(CFL + a/Re_x) \leq \beta \quad (2)$$

Above, $CFL$ is the CFL number, $Re_x$ is the cell Reynolds number and $a$ and $\beta$ are constants approximately equal to unity, whose exact values depend on the particular scheme used. Equation (2) reduces to the CFL stability criterion for inviscid flow.

Three steady state runs are presented for the case of a double circular arc (DCA) compressor cascade. Results in terms of isentropic Mach number distribution, boundary layer displacement thickness and momentum thickness are compared with measurements by Hoheisel and Seyb (1986), Navier-Stokes calculations by Ho (1990) and full potential computations using a coupled integral boundary layer method (Walker, 1987). At design conditions all solvers are in good agreement with the measurements. However, for the off-design case the Navier-Stokes solvers predict a more accurate boundary layer growth. The second example represents a subsonic compressor configuration at an inlet Mach number of 0.5, which has been tested at the Office National d'Études et de Recherches Aérospatiales (ONERA) in France (Szeczenyi, 1980). Simulations have been performed for three different freestream Mach numbers ranging from attached flow to partly separated flow. Steady and unsteady surface pressures are compared with measurements and a linearized full velocity potential method (Whitehead, 1982). For the low incidence case both computational methods show similar results, but for the high incidence cases the results differ. The Navier-Stokes solver shows the correct trend from a maximum unsteady surface pressure at rearward the location of the moving boundary layer reattachment point, but the amplitude of the unsteady surface pressure distribution differs substantially from the measurements. These differences between calculations and measurements are not completely understood at the present time but could possibly be attributed to the fact that in the experiments one blade was oscillating and the other blades were fixed, whereas in the computations all blades were oscillating with an interblade phase angle of 180 degrees. Another possible reason for the differences is the simplified modeling of turbulence in the separated region at the leading edge. However, more work is needed to clarify the importance of both these issues.

**GOVERNING EQUATIONS**

Since a part of the computational mesh is deforming, it is convenient to express the governing equations in total time derivatives rather than partial time derivatives. The total time derivative expresses the rate of change of a quantity following a fixed point in a general coordinate system that is free to move and deform.

$$dU/dt = \partial U/\partial t + u_i\partial U/\partial x_i \quad (3)$$

Above, $u_i$ is the local velocity of the moving coordinate system. The Navier-Stokes equations that govern viscous compressible flows can then be expressed in their conservative form:

$$dU/dt + \partial F^e/\partial x_i = \partial F^v/\partial x_i + Q \quad (4)$$

where $U$, $F^e$, $F^v$ and $Q$ denote the flow variables, convective flux terms, viscous flux terms and source terms, respectively. The source terms $Q$ are deduced to the set of equations to account for the quasi three-dimensional effects of an axially varying stream tube thickness. The summation convention is adopted in the equations above. By expressing the equations in terms of total time derivatives the convective terms are adjusted to account for the convection associated with the moving point of interest. Similarly, the source terms are adjusted to account for a spatially varying $u_i$.

$$U = \begin{pmatrix} \varrho \\ \varrho u_i \\ e \\ \varrho u_j \\ e \\ e(u_j - u_i) \\ e(u_j - u_i) \\ e(u_j - u_i) + p \delta_{ij} \\ \delta_{ij} \end{pmatrix}, \quad F^e = \begin{pmatrix} 0 \\ \varrho u_i u_j \\ (e + p)u_i \\ \varrho u_i \delta_{ij} + (e + p)u_i \\ \varrho u_j \delta_{ij} + (e + p)u_j \\ \varrho u_j \delta_{ij} + (e + p)u_j \\ \varrho u_j \delta_{ij} + (e + p)u_j \\ \varrho u_j \delta_{ij} + (e + p)u_j \\ \varrho u_j \delta_{ij} + (e + p)u_j \end{pmatrix}, \quad F^v = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{pmatrix} \quad (5)$$

Here $\varrho$, $p$, $e$, $T$, $K$, $\delta_{ij}$ and $h$ denote density, static pressure, total energy, static temperature, effective thermal conductivity, the Kronecker delta and the stream tube thickness, respectively, and $\varrho u_j$ is the component of the fluid velocity in the direction of $x_j$. The influence of the stream tube thickness variation on the viscous stresses and heat flux is here neglected. The stream tube height also varies as a result of the transformation to the moving coordinate system. This effect is assumed to be small and is therefore neglected. The viscous stress tensor is given by

$$\sigma_{ii} = \mu \left( \partial u_i/\partial x_i + \partial u_i/\partial x_i - 2\partial u_k/\partial x_k \delta_{ik} \right) \quad (6)$$

where $\mu$ is the effective viscosity. The equation set is completed by the state equations:

$$p = (\gamma - 1)(e - \frac{1}{2}\varrho u_j^2), \quad T = \frac{1}{\varrho c_v} (e - \frac{1}{2}\varrho u_j^2) \quad (7)$$

Here $\gamma$ denotes the ratio of specific heats and $c_v$ is the specific heat at constant volume.

Turbulence is modeled by an algebraic mixing length eddy viscosity model (Baldwin and Lomax, 1978), in which the effective viscosity, $\nu$, and the effective heat conductivity, $k$, consist of a laminar part ($\nu_L$ and $k_L$) and a turbulent part ($\nu_T$ and $k_T$). Algebraic turbulence models are widely used for steady state calculations. The application of such a model to time dependent calculations is based on the assumption that the flutter period
is much longer than the time it takes for the eddy viscosity to adapt to a certain velocity profile. The model has two layers in which the turbulent eddy viscosity is given by:

\[ \mu_T = \begin{cases} \mu_{T1}, & \text{if } y \leq y_{crossover} \\ \mu_{T2}, & \text{if } y > y_{crossover} \end{cases} \tag{8} \]

where \( y \) is the normal distance from the nearest wall and \( y_{crossover} \) is the smallest value of \( y \) at which the values from the inner and outer formulas are equal. The eddy viscosity in the inner region is described by the mixing length formulation:

\[ \mu_{T1} = \rho \ell \left| \omega \right| \tag{9.a} \]

where \( \left| \omega \right| \) is the magnitude of the vorticity and

\[ l = k y \left( 1 - e^{-\nu^+ / A^+} \right) \tag{9.b} \]

In the outer region the eddy viscosity is described by the following expression:

\[ \mu_{T2} = \frac{C_{CL} \rho F_{WAKE} F_{KLEB}}{\gamma} \tag{10.a} \]

where

\[ F_{WAKE} = \text{MIN}(y_{MAX} F_{MAX}, C_{WK} \gamma y_{MAX} u_{DIFF}^2 / F_{MAX}) \tag{10.b} \]

The quantities \( F_{MAX} \) and \( y_{MAX} \) are determined from the function:

\[ F(y) = y \left| \omega \right| \left( 1 - e^{-\nu^+ / A^+} \right) \tag{10.c} \]

\( F_{MAX} \) is the maximum value of \( F(y) \) for a given velocity profile and \( y_{MAX} \) is the value of \( y \) at which the maximum occurs. Thus the model length scale is determined from the vorticity distribution. The Klebanoff intermittency factor is given by:

\[ F_{KLEB} = \left[ 1 + 5.5 \left( \frac{C_{KLEB}}{y_{MAX}} \right)^6 \right]^{-1} \tag{10.d} \]

The value of \( u_{DIFF} \) is the difference between the maximum and minimum velocities in a given profile. The various constants are taken as:

\[ A^+ = 26 \quad C_{CL} = 0.02688 \quad C_{KLEB} = 0.3 \quad k = 0.41 \quad C_{WK} = 0.25 \quad \text{and} \quad \beta_{PR} = 0.9 \]

where \( \beta_{PR} \) is the turbulent Prandtl number. A laminar Prandtl number of 0.72 is used. No special treatment is made of any separated zones. Transition location is currently specified as input data and the wake region is modelled by a separate wake formulation, implemented as described by Visbal and Shang (1985).

**NUMERICAL METHOD**

The general formulation (4) of the governing equations allows the total time derivative to describe the rate of change following a moving grid point.

\[ u_i^* = \frac{dx_i}{dt} \tag{11} \]

In the discretized equations the additional convective contributions due to \( u_i^* \) can be identified as the extra convective fluxes across each cell face due to the motion of the face, and the addition to the source terms is due to the area deformation of each cell.

**Explicit method**

The explicit solver, which is used on the outer unstructured mesh, uses the Taylor-Galerkin finite element technique (Sidén et al., 1990; Morgan and Peraire, 1987). The time domain is divided into finite levels, denoted by \( k \), and the equations are solved by marching forward in time starting from a given initial condition. To go from time level \( k \) to level \( k + 1 \) the following two steps are taken:

**Step 1:**

\[ U^{k+1/2} = U^k - \frac{\Delta t}{2} \frac{\partial F_{U}^c}{\partial x_i} + \frac{\Delta t}{2} Q^k \tag{12.a} \]

**Step 2:**

\[ \Delta U = -\frac{\Delta t}{2} \frac{\partial F_{U}^c}{\partial x_i} + \frac{\Delta t}{2} \frac{\partial F_{U}^v}{\partial x_i} + \Delta t \frac{\partial Q^k}{\partial x_i} + D \tag{12.b} \]

In the first step a predicted value of \( U \) is estimated halfway in between the two time levels. The predicted quantities are estimated by balancing the convective fluxes and source terms at level \( k \) and neglecting all viscous contributions. In the second step the predicted variables are used to estimate the convective fluxes and source terms. The viscous terms are lagged at time level \( k \). All quantities at the intermediate level \( k + 1/2 \) are interpolated on piecewise constant elements and the quantities at time levels \( k, k + 1 \) and so on, are interpolated on piecewise linear elements. The last term in equation (12.b) is a smoothing term of the same form as described by Morgan and Peraire (1987), which is added in order to prevent overshoots in the vicinity of shocks. Since all results presented in this paper are purely subsonic, at least in the unstructured region, a value of \( D \) equal to zero is used.

The explicit method is implemented on an unstructured grid, which for the cases presented in this report may seem unnecessary, but has the advantage of allowing spatial resolution in regions of shock waves.

**Implicit method**

The factorized implicit method of Beam and Warming (1978) is employed on the structured C-mesh in the region near the blade. The solution at each time level is achieved by performing three sequential steps.

i) Implicit sweep in the I direction

\[ \left( 1 + \psi \int \frac{\partial F_{V}^c}{\partial U} n_i d s_i + \psi \int \frac{\partial F_{V}^v}{\partial U} n_i d s_i \right) \Delta U^{**} = \psi \int \frac{\partial F_{V}^c}{\partial U} U_k n_i d s + \psi \int \frac{\partial F_{V}^v}{\partial U} U_k n_i d s + \frac{\Delta t}{(1 + \theta)} \frac{\partial Q}{\partial U} U_k + \frac{\theta}{(1 + \theta)} \Delta U^{k-1} + D^{(2)} + D^{(3)} + D^{(4)} \tag{13.a} \]

ii) Implicit sweep in the J direction

\[ \left( 1 + \psi \int \frac{\partial F_{V}^c}{\partial U} n_i d s + \psi \int \frac{\partial F_{V}^v}{\partial U} n_i d s \right) \Delta U^{*} = \Delta U^{**} \tag{13.b} \]

iii) Point implicit sweep of source terms

\[ \left( 1 - \frac{\Delta t}{(1 + \theta)} \frac{\partial Q}{\partial U} \right) \Delta U^{*} = \Delta U^{*} \tag{13.c} \]

where \( \psi = \Delta t/[(1 + \theta) A_r] \), \( A_r \) is the area of the \( r \)th computational cell, \( \theta \) is a time weighting parameter (equal to 0.5 for time accurate simulations), \( \partial F_{V}^c/\partial U \), \( \partial F_{V}^v/\partial U \) and \( \partial Q/\partial U \) are the Jacobians, \( n_i \) is the unit normal vector directed outwards on a cell face and \( \int d s_i \) and \( \int d s \) denote integration over cell faces with normals pointing in the I-direction and J-direction, respectively. \( D^{(2)} \) is a pressure sensing artificial viscosity term to
smooth the solution in the vicinity of shocks and $D^{(k)}$ is a fourth order smoothing term to provide a low amount of background smoothing in order to prevent node to node oscillations. The two smoothing terms in the $J$ direction (direction normal to the blade) are set proportional to the local Mach number to avoid the numerical diffusion to mask the physical diffusion through the boundary layer.

As seen from equations (13.a-c) the implicit operator is factorized into two block tridiagonal operators and one point implicit operator that are solved sequentially. It should be noted that the basic second order time accuracy is not lost by the factorization. First, the equations are solved implicitly in the $I$ direction by a direct matrix inversion technique, then similarly in the $J$ direction and finally the source terms are treated point implicitly. The $I$ and $J$ directions are curvilinear coordinate directions given by the structure of the mesh. Hence, a structured mesh is necessary. The spatial discretization is handled by using cell centered quantities and the flux across each cell face is estimated by averaging the flow variables from the neighboring cell centers.

The structured mesh is deformed such that the computational blade boundary at all points in time coincides with the actual physical blade boundary, whose motion is assumed known. Interior points in the structured mesh are moved using the transfinite mapping technique, to maintain good discretization practice. For such small blade excursions as flutter simulations present ($\leq 2\%$ of chord) it is unnecessary to deform the outer unstructured mesh, and it is therefore kept fixed throughout the simulation.

Boundary conditions

The boundary conditions on the upstream and downstream boundaries are applied by making use of characteristic theory (Giles, 1986). This boundary condition is non-reflecting for planar waves traveling in a direction normal to these boundaries. In general for a vibration cascade the far field acoustic field will be composed of one or more families of waves which are not necessarily traveling normal to the inflow and outflow boundaries. Therefore, in general, these boundary conditions will not be strictly non-reflecting. The accuracy of these boundary conditions and the error introduced in the resulting solution depends on the placement of these boundaries and the far field decay rate of these waves. On the upstream boundary the total pressure, $p_0$, the total enthalpy, $H$, and the tangential fluid velocity are prescribed, and on the downstream boundary the static pressure is prescribed. On the blade boundary no flux of mass is assumed across the blade boundary. However, for the energy equation, work done by pressure due to the surface movement must be retained. The wall static pressure is obtained by simply using the pressure in the cell center closest to the wall, which is consistent with the assumption of small transverse pressure gradients through the boundary layer. The wall friction is evaluated by extrapolating the velocity profile from the two cells closest to the wall. The treatment of the interface boundary between the two different types of meshes is handled by overlapping the two boundaries the equivalent distance of one computational cell. Fig 1 shows the details of the mesh in the vicinity of the interface boundary. The exact positions of the overlapping node points are not required to be the same for the two meshes, but the fluxes across any point on the boundary are obtained through one-dimensional linear interpolation between the node points. At each time step the solution on the unstructured mesh and the solution on the structured mesh are solved independently. After the time step is completed the solution at the boundary nodes is discarded and replaced by the interpolated solution from the other respective mesh. The use of this type of treatment of the interface boundary is only justified for CFL numbers less than unity, where the CFL number is based on the overlap cell size. If this condition is fulfilled any perturbation introduced on the boundary will propagate a distance smaller than a cell size, hence limiting the perturbation to the boundary cells. This perturbation can then be corrected by applying the correct boundary condition in a sequential step from the computed solution on the other respective mesh.

Periodicity condition

Steady state calculations and flutter calculations with zero interblade phase angle present no difficulty in the application of the periodicity condition. The periodic boundaries are defined by two sets of nodes. Each node on the lower periodic boundary has a corresponding node on the upper periodic boundaries, with the same axial coordinate but spaced one pitch apart in the circumferential direction. Any contribution to the flux sum that is added to a periodic boundary node is also added to its twin node. Thus, all nodes on the periodic boundary are treated exactly the same as the interior nodes.

For the cases with non-zero interblade phase angle, however, some mechanism must be introduced that takes the phase shift between two neighboring blades into account. This is done by using the direct store method (Gerolymos, 1988). The computed flow variables at the upper periodic boundary are stored in an array and applied at a later time, corresponding to the phase shift at the lower periodic boundary. In the same manner, the flow variables on the lower periodic boundary are stored and applied at a later time, corresponding to $2\pi$ minus the phase shift on the upper periodic boundary.

For the cases with non-zero interblade phase angle, however, some mechanism must be introduced that takes the phase shift between two neighboring blades into account. This is done by using the direct store method (Gerolymos, 1988). The computed flow variables at the upper periodic boundary are stored in an array and applied at a later time, corresponding to the phase shift at the lower periodic boundary. In the same manner, the flow variables on the lower periodic boundary are stored and applied at a later time, corresponding to $2\pi$ minus the phase shift on the upper periodic boundary.

$U^k_n = U^k_{1-n}BPA$, $U^k_1 = U^k_{N+1}BPA$ (14)

Subscripts $l$ and $u$ denote the lower and upper periodic boundary, respectively. $N$ is the number of time steps per flutter period and $IBPA$ is the number of time steps with which the motion of the lower blade precedes the motion of the upper blade. The implementation of this algorithm is straightforward and involves only minor changes of a code that already can handle zero interblade phase angles. The periodicity condition (14) is implemented by using characteristic theory. The characteristic variables $q_1$ and corresponding eigen values $\lambda_1$ are computed on the periodic boundary.

$q_1 = \frac{q - p/c}{c}$, $\lambda_1 = u_n$

$q_2 = u$, $\lambda_2 = u_n + c$

$q_3 = \frac{u_n + p/c}{c}$, $\lambda_3 = u_n - c$

$q_4 = \frac{u_n - p/c}{c}$, $\lambda_4 = u_n - c$

The linearized quantities, denoted by overbars, are taken from the current time level $k$, $u_n$ and $u_n$ denote the velocity in the direction normal and tangential to the boundary and $c$ is the speed of sound. There will be two sets of characteristic variables corresponding to the stored quantities and one set of characteristic variables corresponding to the extrapolated flow variables at time level $k + 1/2$. If eigen value $\lambda_1$ is non-negative, i.e. characteristic $q_1$ is transported out of the computational domain, the extrapolated quantity $q_1^*$ is used to compute the flux across the periodic boundary. On the other hand, if $\lambda_1$ is negative then the stored quantity $\lambda_1^*$ is used on the boundary. This treatment assumes small viscous effects on the periodic boundary and becomes questionable when there is considerable shedding from the trailing edge convected across the boundary. In the case of shedding, however, periodicity can no longer be guaranteed and the 'direct store' method is not applicable. This means that situations such as rotating stall cannot be addressed with this method.

Fig 1. Computational mesh for V2 computation. The structured mesh consists of 314 X 29 cells and the unstructured region consists of 2134 node points.
In a typical turbomachinery flutter simulation we encounter two thousand to fifteen thousand time steps per flutter period. For each periodic boundary node and flow variable an array has to be kept in the computer memory with the same number of entries as there are time steps per flutter period. This is commonly the part of the program that requires the most computer storage. In practice this problem is solved by only storing every 114th time step. The complete time history for the intermediate time steps is later recovered by performing a cubic spline interpolation among the stored points.

**STEADY STATE TEST CASE**

The two-dimensional blade geometry considered in this chapter corresponds to a 'double circular arc (DCA) V2' compressor cascade designed for an inlet Mach number of 0.85 and a turning of 50 degrees. This geometry is representative for the hub section of current axial compressors. The experimental investigations were carried out in the high speed cascade wind tunnel in Braunschweig, West Germany, as reported by Hoheisel and Seyb (1986) The blade has a chord of 80 mm and an aspect ratio

V2: TP7360, \( M_1 = 0.6, \beta_1 = 49.5 \)

- Experiments by Hoheisel and Seyb (1986)
- Results by Walker (1986)
- Results by Stow (1989)

---

V2: TP8343, \( M_1 = 0.8, \beta_1 = 49.5 \)

- Experiments by Hoheisel and Seyb (1986)
- Results by Walker (1986)
- Results by Stow (1989)

![Fig 2](image1.png)

**Fig 2.** a) Isentropic Mach number distribution, b) boundary layer displacement thickness and c) momentum thickness distributions versus the axial chord position at an upstream Mach number of 0.6 and at design incidence.

![Fig 3](image2.png)

**Fig 3.** a) Isentropic Mach number distribution, b) boundary layer displacement thickness and c) momentum thickness distributions versus the axial chord position at an upstream Mach number of 0.8 and at design incidence.
of 3.75. Results consist of steady state surface pressure distributions along the chord and boundary layer measurements at midspan location. The experimental study has also considered variation of the axial velocity density ratio through the cascade by manipulating the tunnel side walls. Computed results are compared with measurements, Navier-Stokes computations by Ho (1990), who uses a one-equation model for the turbulent kinetic energy, and full velocity potential calculations by Walker (1986). The latter solver uses an integral boundary layer approach to obtain the blockage effect of the boundary layer. Measurements and computations have also been presented by Stow (1989).

Fig 4. a) Isentropic Mach number distribution, b) boundary layer displacement thickness and c) momentum thickness distributions versus the axial chord position at an upstream Mach number of 0.6 and at off-design incidence.

Analysis of compressor cascades often tends to be difficult due to the inherent problem of predicting the growth of turbulent boundary layers in strong adverse pressure gradients. An additional complication in this specific case is the laminar separation bubble that appears just downstream of the leading edge, even at design conditions. No special treatment of transition location is done in the runs presented here. Hence, transition to turbulence is assumed at the leading edge. Two cases at design incidence (49.5 degrees flow angle) and one case at large positive incidence (54.5 degrees flow angle) are considered. The two first cases (TP 7360 and TP 8343) have inlet Mach numbers of 0.6 and 0.8, respectively. In the last test case (TP 7345) the upstream Mach number is 0.6. The computational mesh is shown in Fig 1. The different solvers are compared in Figs 2 to 4, where the isentropic Mach number distribution, boundary layer displacement thickness and momentum thickness are plotted versus the axial chord position. At design incidence all computer programs perform fairly well. The low Mach number case (TP 7360) indicates good agreement between computations and measurements (Fig 2). However, at high subsonic Mach number (TP 8343) the flow expansion around the leading edge suction side gives rise to a shock at about 10% of chord. Results by Ho clearly show the boundary layer thickening as a result of the shock/boundary layer interaction. Considering the boundary layer displacement thickness and momentum thickness Ho gets very good agreement with measurements. The algebraic turbulence model used in this paper does not show the same effect but results are still in fairly good agreement with measured data. The same is true for the integral boundary layer method (Fig 3). At off-design incidence the integral boundary layer code predicts smaller boundary layer growth on the blade suction side than the measurements and the two other solvers indicate, hence, underestimating the profile loss (Fig 4).

UNSTEADY TEST CASE

This two-dimensional subsonic cascade configuration has been tested in a rectilinear cascade tunnel at the Office National d'Etudes et de Recherches Aerospatiales (ONERA). The configuration and experimental results are presented by Szeczenyi (1980) and compared with results of a full velocity potential flow solver (Whitehead, 1982). Both the experimental results and the computational results are compiled in the report by Bölcs and Fransson (1986). The cascade configuration consists of six fan stage tip sections. Each blade is symmetric around its mean line and has a chord of 0.09 meters and a span of 0.12 meters. The maximum thickness-to-chord ratio is 0.027, the gap-to-chord ratio is 0.95 and the stagger angle is 59.3 degrees. The exact cascade geometry and the profile coordinates are given by Bölcs and Fransson (1986). The unsteady measurements consist of 19 flush-mounted high response pressure transducers on one blade (9 on the pressure side and 10 on the suction side). Three aerelastic test cases are considered here, all subject to the same inlet Mach number ($M_1 = 0.5$) and torsional blade motion with a blade amplitude $a$ of 0.00524 rad and a reduced frequency $k$ of 0.37, but ranging from attached to partly separated flow ($i = 2, 4, 6$ and 6 degrees).

$$k = \frac{\omega c_b}{2v_{ref}}$$ (15)

Here, $v_{ref}$ denotes a reference velocity, which is taken as the upstream velocity for compressor cascades. $\omega$ and $c_b$ are the angular frequency and the blade chord, respectively. Unsteady results are presented in the form of real and imaginary parts of the fundamental component of the surface pressure coefficient $C_p$. These represent the components of the unsteady pressure that are in phase and out of phase with the blade motion, respectively. $C_p$ is obtained through Fourier decomposition of the time varying surface pressure, and is scaled by the blade amplitude.

The unsteady computations were carried out for four flutter periods after which the time varying moment coefficient indi-
cated convergence to a periodic solution. Eight thousand time steps per period were used which corresponds to a maximum CFL number of about 50. This time step was chosen solely on the basis of numerical stability, not considering temporal accuracy. A study of the time step's influence on the unsteady response is naturally of interest, but is beyond the scope of this paper. However, it should be noted that the fundamental frequency component is resolved using eight thousand time steps per period which means that even the sixth harmonic is quite well resolved by 125 time levels per period. The accuracy of the solution is to a great extent dependent on the importance of the high frequency components in the solution. The computational mesh consists of 300 X 24 cells in the structured region and 2652 node points in the unstructured region (Fig 5).

Fig 5. Computational mesh for the subsonic compressor cascade. The structured region consists of 300 X 24 cells and the unstructured region consists of 2652 node points.

A correct treatment of the physics of this test case requires dealing with several topics that are not fully understood. At low incidences the boundary layer separates at the leading edge on the suction side. As the incidence angle is increased the separation bubble increases in size. Some small distance downstream of the separation point there is a short transition region and further downstream the fully turbulent boundary layer reattaches to the blade. Judging from the measurements the unsteady blade surface pressure seems to reach a maximum at the position of the steady state reattachment point on the suction side. This peak in unsteady pressure is most likely caused by the motion of the reattachment point. The adopted turbulence model has shortcomings in predicting the eddy viscosity in regions of transition to turbulence and in boundary layers in large pressure gradients. Without trying to model these processes we cannot hope to predict the losses correctly for this cascade. In the computations presented here transition to turbulence is assumed at the leading edge. This treatment simplifies the computations considerably but, no doubt, neglects some of the physics. The calculations underpredict the viscous loss in all three cases. This is compensated by specifying a contraction ratio (average velocity density ratio) of 1.05. This value is the same for all three runs and is chosen such that the calculations match the measured upstream Mach number, incidence angle and the total to static pressure ratio across the cascade.

Discussion of steady state results

Steady state surface pressure coefficients are compared with experiments and the full velocity potential solver (Figs 6 to 8). The Navier-Stokes solver (VOLFAP) is in general better in predicting the surface pressure distribution since it, at least to some degree, accounts for viscous effects. However, although the trends of the measurements are predicted there are quite

CASE 1: $M_1 = 0.5, \ i = 2$ deg

![Fig 6. Steady state static pressure coefficient versus the true chord position for the subsonic compressor cascade with two degrees incidence angle and an upstream Mach number of 0.5.](image)

CASE 2: $M_1 = 0.5, \ i = 4$ deg

![Fig 7. Steady state static pressure coefficient versus the true chord position for the subsonic compressor cascade with four degrees incidence angle and an upstream Mach number of 0.5.](image)

CASE 3: $M_1 = 0.5, \ i = 6$ deg

![Fig 8. Steady state static pressure coefficient versus the true chord position for the subsonic compressor cascade with six degrees incidence angle and an upstream Mach number of 0.5.](image)
large differences locally. It is suspected that these differences are caused by the shortcomings of the turbulence model, although a thorough investigation of different specifications of transition locations is needed to verify this.

In the high incidence case the predicted surface pressure is fairly constant in the separated region, which is typical for a turbulent separation (Fig 8). The measurements, however indicate a large variation of pressure throughout the separated region. The error in predicted loss also affects the overall loading on each profile. This is especially evident in the vicinity of the leading edge where the predicted surface pressure on the suction side is shifted compared with that measured. This shift becomes smaller further downstream. The importance of correct loss prediction for this test case can be seen, for example, in the calculation with an incidence angle of two degrees (Fig 6). The measured static-to-static pressure ratio is in this case slightly less than unity which, for inviscid flow, would yield a negative incidence angle and negative lift since the profile is symmetric. The actual incidence angle is, however, two degrees and the measured surface pressure indicates considerable positive lift.

Discussion of time dependent results

Real and imaginary parts of the surface pressure distribution are compared with measurements and the linearized full velocity potential solver (Fig 9-11). Before a detailed discussion of the results is presented it is again worth noticing that the experiments were carried out with only one blade oscillating and all other blades fixed, presumably under the assumption that the aerodynamic blade-to-blade coupling is small. Both the Navier-Stokes computations and the full potential calculations presented by Bolcs and Fransson (1986) were carried out using an interblade phase angle of 180 degrees, hence, assuming that all blades in the cascade are moving.

CASE 1:  $M_1 = 0.5, \ i = 2 \text{ deg}$

For the case with two degrees incidence angle (Fig 9) both computational methods produce similar unsteady surface pressure distributions and they compare fairly well with measured data. However, there are discrepancies especially in the amplitude in the leading edge region. Bolcs and Fransson (1986) point out that by breaking down the computed results into influence coefficients for each blade closer agreement with the experiments, in which only one blade was oscillating, is achieved. Considering the case with four degrees incidence angle the very low measured static pressure close to the leading edge on the suction side seems to indicate a leading edge separation bubble (Fig 7).

As the blade oscillates the time varying incidence angle results in a periodically growing and shrinking separated region. This in turn, gives rise to a large variation in unsteady pressure at the location of the boundary layer reattachment point (Fig 10). The same tendency is also predicted by the Navier-Stokes solver although the amplitude is about twice as high as measured and the width of the peak is somewhat larger. If we finally consider the case with six degrees incidence angle the unsteady pressure distribution looks much like the previous case, with the difference being that the peak in unsteady pressure has moved further downstream and is wider than before. The Navier-Stokes code also here predicts the correct trend but overestimates the amplitude in the separated region. The linearized full potential method assumes the flow to be attached and neglects all pressure fluctuations associated with the unsteady separation (Fig 11).

Fig 12 shows the integrated moment coefficient $C_M$ as a function of the incidence angle.

\[
C_M = - \int C_p(R \times \mathbf{n}) \, ds \quad (16)
\]

CASE 2: $M_1 = 0.5, \ i = 4 \text{ deg}$
**CASE 3**: $M_1 = 0.5$, $i = 6 \text{ deg}$

- Measured pressure side pressure coefficients (Szechenyi, 1980)
- Measured suction side pressure coefficients (Szechenyi, 1980)
- Full potential results by Bőlcs and Fransson (1986)

Above, $C_p$ is the unsteady surface pressure coefficient as displayed in Figs 9 to 11. $\mathbf{R}$ is a dimensionless vector from the pivot axis to an arbitrary point on the blade surface, $\mathbf{n}$ is the unit normal vector on the surface and the integral is evaluated around the blade surface. $C_M$ is defined positive in the counter clockwise direction. Both computational methods compare well with the measurements considering the phase of $C_M$. However, there are quite large differences in amplitude for the large incidence cases.

The periodic variation of the leading edge separation bubble, for the case with six degrees incidence angle, is shown in a sequence of iso-Mach number plots (Fig 13). The blade-to-blade periodicity is shown in Fig 14 by a plot of iso Mach-number contours at a point in time for the case with two degrees incidence.

**Fig 11.** a) Real and b) imaginary parts of the fundamental component of the unsteady static pressure coefficient are plotted versus the true chord position for the subsonic compressor cascade with six degrees incidence angle and an upstream Mach number of 0.5.

**Fig 12.** Aerodynamic moment coefficient and phase lead versus the incidence angle.

**Fig 13.** The periodic leading edge separation bubble is shown by contours of Mach number at different point in time for the subsonic compressor cascade with six degrees incidence angle and an upstream Mach number of 0.5.
The differences between the Navier-Stokes computations and the measurements can to some extent be explained by the fact that only one blade was oscillating in the experiments, whereas all blades move in the computer simulations. Bolcs and Fransson (1986) show an improved agreement with the measurements by decomposing the linearized full potential results to response coefficients for each individual blade. The decomposed response coefficients would better correspond to the case with only one blade oscillating. This did to some extent explain the differences between the full potential results and the measurements in the case with two degrees incidence. However, more likely to have an effect, at least for the high incidence cases, is the fact that the turbulence model does not very accurately describe the complex flow in the leading edge region. For steady state flows transition to turbulence can be simulated by explicitly specifying the location for transition to turbulence. For unsteady flows, however, the time varying incidence angle will force the transition location to move in a periodic fashion, which requires the complete time history of the transition location to be specified. This is the reason why, in this paper, transition to turbulence is assumed at the leading edge. An alternate approach could be to use a one- or two-equation model, instead of a simple algebraic turbulence model, which does not require the transition location to be specified. Finally, it is possible that the steep pressure gradients in the leading edge region give rise to short wave length pressure waves which are not sufficiently resolved in the time domain by the numerical scheme used. This issue could be resolved by studying the unsteady response for a range of different time steps.

CONCLUSION

A time-accurate Navier-Stokes solver has been tested against both measurements and a linearized full potential flow solver with emphasis on blade flutter simulations. In a first steady state case the importance of solving the Navier-Stokes equations at off-design flow conditions is shown for a subsonic DCA compressor cascade. The Navier-Stokes method produces better boundary layer predictions at high incidences than the integral boundary layer method. The second test case considers a subsonic compressor cascade in which one blade is oscillating. Three incidence angles are considered ranging from attached to partially separated flows. The measured unsteady surface pressures indicate a maximum in amplitude in the vicinity of the leading edge separation bubble. This maximum is also predicted by the Navier-Stokes solver but not by the linearized full velocity potential code.

Although inviscid, and especially linearized inviscid, unsteady flow solvers have several computational benefits compared with the viscous flow solver presented in this paper, it is shown that viscous effects due to, for example, regions of separated flow can be of major importance in the case of blade flutter predictions and, hence, cannot be neglected. However, although the Navier-Stokes results qualitatively resemble the unsteady experimental data there remain quite large differences locally. Some of the differences can be attributed to different conditions for the experiments and computations, such as the issue of interblade phase angle discussed earlier. Another source of error is the modeling of turbulence in the separated region near the leading edge. This is substantiated by an underprediction of the steady state total pressure loss over the cascade.

More work is needed to isolate the causes for the discrepancies between the unsteady experimental results and the computations and to apply the method to a wider range of test cases. This must be done before it is shown that this method can be used to predict the flutter behavior of compressor and fan blades at off-design flow conditions.

ACKNOWLEDGEMENTS

The computer time for the calculations was provided in part by the National Supercomputer Centre in Sweden. The author would also like to acknowledge the support of VOLVO FLYGMOTOR AB.

REFERENCES


