ANALYSIS OF TRANSONIC TURBOMACHINERY FLOWS USING A 2-D EXPPLICIT LOW-REYNOLDS k-ε NAVIER-STOKES SOLVER

F. Dejean, C. Vassilopoulos, G. Simandirakis, K. C. Giannakoglou, and K. D. Papailiou
Laboratory of Thermal Turbomachines
National Technical University of Athens
Athens, Greece

ABSTRACT
An explicit, time-marching fractional-step solver for the calculation of the two-dimensional compressible Navier-Stokes equations is presented. The advantage of using a fractional-step analysis is its simplicity and the fact that greater time-steps are allowed, since the stability criterion is less strict compared to other explicit solvers. Turbulence is modeled through a low-Reynolds k-ε model, for which a novel artificial viscosity scheme is implemented, ensuring a smooth e-distribution close to solid walls. The method is used in order to numerically investigate the flow field in three different cascades, namely a highly loaded transonic linear turbine guide vane cascade in six different flow conditions, a transonic steam turbine cascade in two different flow conditions and a low supersonic compressor cascade. Calculations are performed using both H- and C-type grids.

NOMENCLATURE

- \( c \): speed of sound
- \( c_v \): specific heat at constant volume
- \( \varepsilon_1, \varepsilon_2, \varepsilon_3 \): empirical constants for the k-ε model
- \( e \): internal energy per unit mass \( (e = c_v T) \)
- \( E_t \): total energy per unit volume \( (E = \rho e + \frac{1}{2} \rho V^2 + p) \)
- \( f_{ij} \): empirical functions for the k-ε model
- \( G_{ij} \): contravariant metric tensor \( (i=1,2, j=1,2) \)
- \( J \): Jacobian of the curvilinear transformation
- \( k \): turbulent kinetic energy
- \( k_2, k_4 \): artificial dissipation constants \( (k_2 = 0.25, k_4 = 0.01) \)
- \( n \): normal to the wall direction
- \( p \): static pressure
- \( P_{eff} \): effective pressure \( (p_{eff} = p + \frac{4}{3} \rho k) \)
- \( P_{RL} \): production term in the k-ε equations
- \( Pr_{RL}, Pr_r, Pr_k, Pr_e \): Prandtl numbers \( (Pr_{RL} = 0.72, Pr_r = 0.9, Pr_k = 1.0, Pr_e = 1.3) \)
- \( Q \): conservative variable array \( \{ J_0, J_{10}, J_{01}, J_{11}, J_{00}, J_{02}, J_{20}, J_{21}, J_{12}, J_{22}, J_{13}, J_{31}, J_{23}, J_{32}, J_{14}, J_{41}, J_{24}, J_{42}, J_{34}, J_{43} \} \)
- \( Re \): Reynolds number
- \( T \): temperature
- \( u, v \): Cartesian velocity components
- \( \nu_s \): local velocity magnitude
- \( \chi, \gamma \): contravariant velocity components
- \( \xi, \eta \): Cartesian coordinates
- \( \xi_\eta \): dimensionless distance from the wall \( (\xi = \rho u \eta / \nu) \)
- \( T_{ij} \): molecular, turbulent and effective viscosity coefficients
- \( \epsilon \): isotropic turbulent energy dissipation
- \( \mu_{eff} \): molecular and turbulent viscosity coefficients
- \( \mu \): effective viscosity
- \( \nu \): turbulent viscosity
- \( \nu_s \): density
- \( \rho \): Cartesian stress tensor components
- \( \tau \): turbulence intensity
- \( \tau_1 \): normal stress tensor components
- \( \tau_2 \): shear stress tensor components
- \( \tau_3 \): shear stress tensor components

Subscripts
- \( 1 \): inlet
- \( 2 \): exit
- \( is \): isentropic flow
- \( w \): wall

Presented at the International Gas Turbine and Aeroengine Congress and Exposition
The Hague, Netherlands – June 13–16, 1994
INTRODUCTION

The design and analysis of turbomachinery components has become increasingly dependent on the solution of the Navier-Stokes equations. The accuracy of any computational tool depends on the discretization error in space and in time (for time accurate calculations) and on the successful selection of a turbulence model. Its efficiency in terms of robustness and computational speed is mainly influenced by the use of explicit or implicit schemes, within the frame of a time-marching procedure. At the present time, there exist contradictory assessments of explicit and implicit solvers, especially in relation to the modern computers of parallel and vector architecture.

There are numerous reviews of CFD techniques in the field of turbomachines (e.g., Lakshminarayana, 1986 for the turbulence models and Lakshminarayana, 1991 for the computational tools) and several comprehensive books on CFD (Hirsch, 1990) and thus, widely presented knowledge will not be repeated here.

Implicit methods present the advantage of allowing larger time-steps compared to explicit ones, provided that an implicit treatment of the boundary conditions is incorporated, which in turn leads to a further complexity of the code structure. Moreover, for cascade calculations the implicit treatment of the periodic boundary conditions adds an extra difficulty, in comparison to explicit methods where periodicity is a trivial task. Simplicity and user-friendliness of an explicit code are its two considerable advantages. An additional advantage is the easy parallelization and vectorization of explicit methods, such as Runge-Kutta or predictor-corrector MacCormack schemes (Gentzsh, 1988). Furthermore, the extension of explicit solvers to three-dimensional or unsteady flows is straightforward.

Turbulence modelling plays its particular role in turbomachinery calculations where algebraic eddy viscosity models are preferable, especially in combination with explicit codes. When higher order turbulence models are incorporated, implicit solvers are generally preferred for compromising the increased stiffness of the turbulence equations. Despite this disadvantage, explicit k-e Navier-Stokes codes can be used to compute two- or three-dimensional viscous flows at high Reynolds numbers (Kunz and Lakshminarayana, 1992a and 1992b). Such explicit codes are divided into two main categories, the coupled and the decoupled explicit methods. The former use conservation equations, that summarize mean flow and turbulent quantities in the same unknown array, while in the latter the averaged k-e equations are numerically decoupled from the averaged mean flow equations. In regard to convergence or accuracy, the two methods are equivalent (Kunz and Lakshminarayana, 1993). In both coupled and decoupled solvers, the source terms in the turbulence equations influence the code stability, mainly during the early stages of the computation. The problem is discussed by Kunz and Lakshminarayana (1992a), who encourage code developers to invest in explicit solvers. On the other hand, the indisputable advantages of the explicit schemes brought to light mixed explicit-implicit solvers, where the mean flow equations are handled explicitly, while the problem related to the stiff source terms is circumvented using implicit schemes for the turbulence equations (Turner and Jennions, 1992).

In the present work, an explicit fractional step code with a two-equation k-e low-Reynolds closure (Jones and Launder, 1972) is presented. This code is named ATHENA (A Turbulent Hyperbolic Explicit Navier-Stokes Algorithm) and it has already been used in turbomachinery calculations (Simandirakis, 1992, Giannakoglou et al., 1991), using the Baldwin-Lomax turbulence model. In the present method, the conservative two-dimensional Navier-Stokes equations are split in a sequence of one-dimensional operators for the inviscid part, the viscous part and the source terms. Thus, instead of applying a pure two-dimensional scheme, a number of one-dimensional steps are executed, for which numerical stability constraints are less strict. For each one-dimensional step, a predictor-corrector MacCormack scheme is used, with the exception of the source terms treatment which is of a semi-implicit type. A second- and fourth-order dissipation scheme is added to prevent odd-even uncoupling and to accurately capture shock waves, which presents an important novelty in regard to turbulence equations. The latter are solved in a coupled way with the mean flow equations. Local time stepping is used to accelerate convergence, by defining a common CFL number for the ensemble of the equations.

The proposed method is validated in three two-dimensional cascades, for which experimental data is available. The three cascades are: (a) a highly loaded transonic linear turbine guide vane cascade (Arts et al., 1990) designed and studied at the Von Karman Institute for fluid dynamics, (b) a transonic steam turbine cascade for which experimental investigations were performed by Stastny and Safarik (1990) and (c) a low supersonic compressor cascade for which measurements were conducted in the transonic cascade wind tunnel of DFVLR (Starken and Schreiber, 1990). Two-dimensional flow patterns, blade velocity or pressure distributions and blade heat transfer are compared against experimental data.

GOVERNING EQUATIONS AND TURBULENCE MODELLING

The conservative form of the two-dimensional, unsteady, Favre-averaged Navier-Stokes equations, with a low-Reynolds k-e closure may be written in the form

$$\frac{\partial \tilde{\mathbf{q}}}{\partial t} + \nabla \cdot \mathbf{\tilde{F}} = \mathbf{\tilde{g}}$$

where the unknown variable vector $\tilde{\mathbf{q}}$, the flux vectors $\tilde{\mathbf{F}}$ and $\mathbf{\tilde{g}}$ and the source term vector $\mathbf{s}$, are given below

$$\tilde{\mathbf{q}} = [\rho, \rho u, \rho v, E_c, \rho k, \rho e]^T$$

$$\tilde{\mathbf{F}} = [\rho \tilde{\mathbf{u}} , \rho \tilde{\mathbf{u}}^2 + P , \rho \tilde{\mathbf{u}} \tilde{\mathbf{v}} , E_c \tilde{\mathbf{u}} , \rho k \tilde{\mathbf{u}} , \rho e \tilde{\mathbf{u}}]^T$$

$$\mathbf{\tilde{g}} = [-P \nabla u , -P \nabla v , \rho \tilde{\mathbf{u}} \cdot \nabla \mathbf{\tilde{u}} , \kappa \nabla^2 \tilde{\mathbf{u}} , \rho \eta \nabla^2 \tilde{\mathbf{u}} , \rho \tilde{\mathbf{u}} \cdot \nabla \chi]^T$$
\[ f = \left[ \rho u, \rho u^2 - \sigma_{xx}, \rho u v - \sigma_{xy}, u \sigma_e, \nu \sigma_{xx}, \nu \sigma_{yy}, \nu \sigma_{xy} \right]. \]

\[ \rho k \left( \frac{1}{2} \mu + \frac{\mu_t}{Pr_t} \right) \frac{\partial}{\partial x} \left( \frac{\partial u}{\partial x} \right) \]

\[ g = \left[ \rho v, \rho u v - \sigma_{xy}, \rho v^2 - \sigma_{yy}, \nu v_x - \mu \sigma_{xy}, \nu \sigma_{yy} \right]. \]

\[ \sigma_{xx} = -\sigma_{yy} = \rho \mu \left( \frac{\partial u}{\partial x} \frac{\partial v}{\partial y} \right) \]

\[ \sigma_{xy} = \mu \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) \]

\[ \sigma_{yy} = -\rho \mu \left( \frac{\partial v}{\partial y} \frac{\partial v}{\partial y} + \frac{\partial u}{\partial x} \frac{\partial v}{\partial y} \right) \]

while the heat flux vector is

\[ (q_x, q_y) = \left( -Pr_{eff} \frac{\partial e}{\partial x}, -Pr_{eff} \frac{\partial e}{\partial y} \right). \]

The effective Prandtl number \( Pr_{eff} \) is defined as

\[ Pr_{eff}^{-1} = \gamma \left( \frac{\mu + \mu_t}{Pr_t} \right) \]

where the laminar viscosity \( \mu \) is calculated through the Sutherland's law.

The form of the low-Reynolds number turbulence model used herein, is the one introduced by Jones and Launder (1972), where the source terms have the following form

\[ s_k = P - \rho e - 2\mu \left( \frac{\partial k}{\partial n} \right)^2 \]

\[ s_e = c_e \frac{P - \rho e}{k} + c_f \rho \frac{\varepsilon^2}{k} + 2 \mu \mu_t \left( \frac{\partial V^2}{\partial n} \right)^2 \]

The production term \( P \), is given by

\[ P = \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + 2 \mu \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 \]

\[ - \frac{2}{3} \mu \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right)^2 - \frac{2}{3} \rho k \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) \]

and the turbulent viscosity \( \mu_t \) is obtained from the Prandtl-Kolmogorov relation,

\[ \mu_t = \frac{c_f \mu k^2}{\varepsilon} \]

For the selected model, the constants and functions used are given by

\[ f_2 = 1 - 0.3 \exp(-Re^2), \quad f_\mu = \exp \left[ \frac{-3.4}{\left( \frac{Re^2}{50} \right)} \right] \]

\[ Re = \frac{\rho k^2}{\mu e} \]

**GEOMETRICAL TRANSFORMATION**

Computations are performed in a body-fitted coordinate system \((\xi, \eta)\), which is numerically generated using a combination of algebraic and elliptic equations. Since cross-derivatives are retained during the discretization, the use of an orthogonal grid is not mandatory. However, the particular form of the low Reynolds terms in the k-e equations, recommends a grid as orthogonal as possible. The governing equations (1) are transformed in the computational plane \((\xi, \eta)\) and the resulting equations may be cast in the following conservative form

\[ \frac{\partial \mathbf{Q}}{\partial \tau} + \frac{\partial \mathbf{F}}{\partial \xi} + \frac{\partial \mathbf{G}}{\partial \eta} = \mathbf{S} \]

where

\[ \mathbf{Q} = J \mathbf{q} \]

\[ \mathbf{F} = J \left[ \mathbf{\xi}_x \mathbf{f} + \mathbf{\xi}_y \mathbf{g} \right] \]

\[ \mathbf{G} = J \left[ \mathbf{\eta}_x \mathbf{f} + \mathbf{\eta}_y \mathbf{g} \right] \]

\[ \mathbf{S} = J \mathbf{s} \]
The Jacobian of the transformation, J is defined as
\[ J = \frac{\partial(x, y)}{\partial(\xi, \eta)} = x_\xi y_\eta - x_\eta y_\xi \]

where \( x_\xi, x_\eta \) etc. are the metrics of the transformation.

**BOUNDARY CONDITIONS**

All the flow cases examined in the present paper, have a subsonic axial velocity component at both the inlet and exit boundaries of the computational domain. Following the theory of hyperbolic systems, all but one dependent variables need to be provided at the inlet; these are the stagnation pressure and temperature and the inlet flow angle. At the exit boundary the static pressure is specified, which is used to determine the mass flow rate and to locate the shock waves. The non-specified dependent variables, at both inlet, and exit are extrapolated from the interior nodes.

At the inlet the turbulence intensity \( \tau_u \) is specified. Therefore assuming an upstream value for the turbulent viscosity (i.e. a \( \mu_c/\mu \) ratio), the inlet values of \( k \) and \( \varepsilon \) are calculated as follows
\[ k_{\text{inlet}} = \frac{1}{2} \left( \frac{\tau_u}{\nu_{\text{inlet}}} \right)^2 \]
\[ \varepsilon_{\text{inlet}} = \frac{k_{\text{inlet}}}{\mu_{\text{inlet}}} \frac{2}{\left( \frac{\mu}{\mu_t} \right)_{\text{inlet}}} \]

Along solid walls the velocity components together with \( k \) and \( \varepsilon \) are set to zero and the pressure is calculated through a redundant form of the normal momentum equation. Depending on the case examined, either the distribution of the wall temperature or a zero heat flux could be alternatively imposed. For cascade flow problems, periodicity conditions are imposed at each step of the algorithm; their implementation is easy, due to the explicit nature of the method.

**NUMERICAL PROCEDURE**

Following the fractional step concept (Laval, 1983), the finite-difference form of the discretized equation (4) is split into a sequence of multiple single-directional operators. Each operator corresponds to a different physical component of the equation (i.e. the inviscid and the viscous parts, as well as the source terms). As a consequence, the time evolution of the unknown vector array \( \bar{Q} \), is obtained by applying the sequence of operators, which is given below in a symbolic notation
\[ \bar{Q}^{n+2} = L^H_\xi L^H_\eta L^P_\xi L^P_\eta L^{ST} L^{LST} L^P_\xi L^P_\eta L^H_\xi L^H_\eta \bar{Q}^n \]  

The following comments on equation (6) must be highlighted:

(a) a double and inverse sequence of the one-dimensional operators leads to a second order accuracy in time, while the calculated quantities have a real physical meaning only at the expiration of a \( 2\Delta t \) time interval (i.e. from \( n \) to \( n+2 \) iteration level) (Abarbanel and Gottlieb, 1981).

(b) one-dimensional operators \( L \) are subscribed by either \( \xi \) or \( \tau \), depending on whether sweeps are performed along the \( \eta \) or \( \xi \) constant grid lines respectively.

(c) the \( L \) operators are superscribed by \( H, P \) or \( ST \) in order to distinguish the nature of the governing equation part which is resolved by each operator. Thus, an \( L^H \) operator represents the scheme used to resolve the inviscid part of the equations (\( H = \text{Hyperbolic} \)), an \( L^P \) solver solves for the viscous part (\( P = \text{Parabolic} \)), while the \( L^{ST} \) operator is the one which handles the source terms in the equations. For instance, the \( L^H_\xi \) operator is used to proceed from the \( \bar{Q}^n \) solution vector to the \( \bar{Q}^{n+1/2} \) intermediate one, by solving the one-dimensional equation
\[ \frac{\partial \bar{Q}}{\partial \tau} = \frac{\partial F^H}{\partial \xi} = 0 \]
where \( F^H \) stands for the inviscid part of the flux vector \( F \).

(d) the method is second order accurate in space, provided that an appropriate one-dimensional numerical scheme is used for the solution of each one-dimensional operator. In the present paper, the predictor-corrector MacCormack scheme is applied for both hyperbolic and parabolic operators (MacCormack, 1988).

(e) a common time step \( \Delta \tau^* \) is used for all \( L \) operators. This is given by
\[ \Delta \tau^* = \min \left[ \frac{1}{|V^i| + C_{fg}^H}, \frac{2 \rho Pr}{\gamma \mu \sigma (4 g^H + g^{12})} \right] \]
\[ i = 1, 2 \quad , \text{no summation} \]

and is the minimum of time steps, which result from the stability analysis performed for each operator separately. The total time increment per time-step is less strict compared to that dictated by any explicit two-dimensional stability criterion.

(f) with regard to the treatment of the source terms, a semi-implicit scheme is used. Thus, for the solution of the intermediate step.
the right-hand-side array \( \mathbf{S} \) is split in two parts \( \mathbf{S}^+ \) and \( \mathbf{S}^- \), which contain the positive and negative source terms respectively. The negative part is Newton linearized and the delta form of equation (8), which results from the aforementioned linearization about the time level \((n+4/5)\), can be written as

\[
\left[ I - \Delta t \left( \frac{\partial \mathbf{S}^-}{\partial \mathbf{Q}} \right)^{n+4/5} \right] \Delta \mathbf{Q}^{n+4/5} = \Delta t \mathbf{S}^{n+4/5} \quad (9)
\]

where

\[
\mathbf{Q}^{n+4/5} = \mathbf{Q}^{n+1/5} + \Delta \mathbf{Q}^{n+4/5}
\]

It is to be noted that since the source terms are handled as an intermediate step within the multi-step scheme (6), the \( L^ST \) operator is used to proceed from the \((n+4/5)\) to the \((n+6/5)\) time-step.

**Artificial Dissipation**

The use of centered finite-difference schemes on a collocated grid requires the addition of an extra amount of non-physical dissipation, in order to prevent odd-even uncoupling effects and to avoid oscillations close to shock waves or stagnation points. The extra dissipation is explicitly added to the solution array \( \mathbf{Q} \) at the end of a complete calculation period, corresponding to a time interval of \( 2\Delta t \), as follows

\[
\mathbf{Q}^{n+2} = \mathbf{Q}^{n+1} + \Delta \mathbf{Q}^{n+2} = \mathbf{Q}^{n+1} + \Delta \mathbf{Q}^{n+2}\quad (10)
\]

The quantity \( \Delta \mathbf{Q}^{n+2} \) represents a blend of second- and fourth-order derivatives of the solution array scaled by the inverse of the Jacobian \( J \). For the \( \xi \)-direction, the artificial dissipation term \( \Delta \mathbf{Q}^{n+2} \) yields

\[
\Delta \mathbf{Q}^{n+2} = \nabla_\xi \left[ 2 \Sigma_\xi(\sigma J) e^{(2)}_\xi \frac{\Delta \xi}{J} \left( \mathbf{Q}^{n+2} \right) \right]
\]

\[
-\nabla_\xi \left[ 2 \Sigma_\xi(\sigma J) e^{(4)}_\xi \Delta \xi \nabla_\xi \Delta \xi \frac{\mathbf{Q}^{n+2}}{J} \right]
\]

where \( \nabla_\xi \), \( \Delta \xi \) are the backward and forward difference operators in the \( \xi \)-direction and \( \Sigma_\xi \) is the forward averaging operator in the same direction.

**Mean Flow Equations**

Following the work by Jameson et al. (1981), the amount of extra diffusion added is controlled through two artificial coefficients \( e^{(2)}_\xi \) and \( e^{(4)}_\xi \), namely

\[
e^{(2)}_\xi = k_\xi \Delta t \max(\sigma, 0)
\]

\[
e^{(4)}_\xi = \max(0, k_\xi \Delta t, e^{(2)}_\xi)
\]

where \( \max(\cdot) \) stands for the local maximum value of \( \cdot \) in the \( \xi \)-direction. For the mean flow equations, the sensor \( \gamma \) is based on static pressure variations

\[
\gamma = \frac{|\nabla_\xi \Delta \xi \mathbf{P}|}{|\Delta \mathbf{P}|}
\]

where \( \Delta \mathbf{P} \) is the local average of the pressure \( \mathbf{P} \), in the \( \xi \)-direction. Special one-sided schemes are used for the numerator and denominator in expression (13), close to the boundaries of the domain. In equation (11), \( \sigma \) stands for the sum of the two one-dimensional spectral radii. Similar analysis is valid for the \( \eta \)-direction as well.

**Turbulence Equations**

For the turbulence equations the added artificial dissipation has the general form of equation (11), similar to the one used for the mean flow equations. Nevertheless, a stability analysis of the \( k \) and \( e \) equations, would provide different lower value spectral radii than those characterizing the system of mean flow equations. This could lead to unequal time-steps for the mean flow and the turbulence equations, provided that a common CFL number is used. If this is the case, a consistent time evolution for both the mean flow and the turbulence equations would require the use of a dual CFL number (Yokota, 1990).

In the present work, the fact that different radii are employed is overlooked and a common CFL number is used, as discussed previously.

In this context, the quantity \( \sigma \) in equation (11) retains its expression as in the mean flow equations and the expression (12) is also valid. The novel aspect concerning the artificial dissipation scheme for the \( k \) and \( e \) equations, is the new sensor \( \gamma \) that influences directly only the second-order term. (Simandirakis, 1992). Thus, for the \( k \) and \( e \) equations the sensor \( \gamma \) is defined as

\[
\gamma = \frac{|\nabla_\xi \Delta \xi e|}{|\Delta e|}
\]

The advantage of the proposed definition of \( \gamma \), which is based on \( e \) rather than \( P \)-variations, is related to the transient behaviour of the turbulent energy dissipation. A temporary unbalance between dissipation and generation of turbulence could generally cause unrealistic negative values of \( e \) close to the walls. Users of the low-Reynolds \( k-e \) model (e.g. Gerolymos,
1990) often bound the turbulence production over dissipation ratio between two extreme values (for example 0.1 and 10). The scheme proposed in equation (14) was proved to be very efficient for protecting the calculation from attaining unrealistic values of $\varepsilon$.

RESULTS AND DISCUSSION

The Highly Loaded Transonic Turbine Guide Vane Cascade

In this section, a highly loaded transonic linear turbine guide vane cascade, for which an experimental investigation was conducted in the von Karman Institute for Fluid Dynamics (Arts et al., 1990), is examined. The blade chord was 67.6 mm, the solidity was 0.85 and the stagger angle was 55 deg. Measurements exist for different isentropic exit Mach numbers ($M_{2ls}$), inlet turbulence intensities $T_u$ and Reynolds numbers ($Re$). In all these cases, the inlet flow angle was set to zero.

![Fig. 1](image1.png)  
**Fig. 1** The C-type grid used for the highly loaded transonic guide vane cascade.

![Fig. 2](image2.png)  
**Fig. 2** A blowup of the computational grid shown in fig. 1, close to the trailing edge.

Six different cases were numerically studied in order to validate the code. These cases correspond to four different isentropic exit Mach numbers between 0.875 and 1.07, three different Reynolds numbers ($5 \times 10^5$, $10^6$ and $2 \times 10^6$) and two different inlet turbulence intensities (1% and 4%). Names and principal characteristics of each case are tabulated below:

<table>
<thead>
<tr>
<th>CASE</th>
<th>$M_{2ls}$</th>
<th>$T_w$</th>
<th>$Re$</th>
<th>$T_u$</th>
</tr>
</thead>
<tbody>
<tr>
<td>MUR45</td>
<td>0.875</td>
<td>300 K</td>
<td>$10^6$</td>
<td>1%</td>
</tr>
<tr>
<td>MUR47</td>
<td>1.020</td>
<td>300 K</td>
<td>$10^6$</td>
<td>1%</td>
</tr>
<tr>
<td>MUR210</td>
<td>1.070</td>
<td>297.35 K</td>
<td>$10^6$</td>
<td>1%</td>
</tr>
<tr>
<td>MUR213</td>
<td>1.070</td>
<td>298.25 K</td>
<td>$10^6$</td>
<td>4%</td>
</tr>
<tr>
<td>MUR228</td>
<td>0.930</td>
<td>302.85 K</td>
<td>$5 \times 10^5$</td>
<td>1%</td>
</tr>
<tr>
<td>MUR247</td>
<td>0.930</td>
<td>302.15 K</td>
<td>$2 \times 10^6$</td>
<td>1%</td>
</tr>
</tbody>
</table>

A single computational grid was used for all six calculations performed and this is presented in figure 1. This was a C-type grid having 241x45 nodes, which was highly stretched close to the blade walls through an elliptic grid generator. The latter assures a quasi-orthogonal and clustered grid using appropriate source terms which were calculated along the inner and outer boundaries of the C-type topology and were exponentially distributed in the inner field (Sorensen, 1980). In this way, the maximum non-dimensional distance $y^*$ of the first grid node from the wall is kept lower than 1.5, which is dictated by the turbulence model in use. A blowup of the computational grid close to the trailing edge is shown in figure 2.
Figure 3 shows the iso-Mach contours for the case MUR45 while figures 4a and 4b present the predicted isentropic Mach number distributions on the pressure and the suction side of the blade along with experimental data, for the cases MUR45 and MUR47. Both distributions are in satisfactory agreement with measurements but a slight discrepancy of the predicted Mach number at the last part of the suction side is observed. This is due to the grid quality in this part of the domain; more specifically, the high stagger angle of the blade and the unequal lengths of the pressure and suction side force grid lines, springing from the last part of the suction side, to skew considerably in order to meet the corresponding nodes on the pressure side.

Figures 5a to 5d present the calculated and the experimental blade heat transfer distributions for the cases MUR210, MUR213, MUR228 and MUR247 respectively. By comparing figures 5a and 5b, the effects of two different inlet turbulence intensities could be analyzed, while all other flow parameters were kept constant. A parametric study for different flow Reynolds numbers, under the same $M_{215}$ and $\tau_u$ was conducted. In all cases, an expected high value of the heat transfer flux, close to the blade leading edge, is identified. This value decreases rapidly along both sides of the blade, due to the development of a laminar boundary layer (Arts and De Rouvreit, 1992). For the case MUR213 (figure 5b), there is a satisfactory agreement between prediction and experiment; especially the suction side is predicted very accurately, with the exception of the near trailing edge zone, where the measured heat transfer continues to diminish and the boundary layer appears to separate. In figure 5a, which corresponds to a lower level of inlet turbulence, the pressure side is predicted with less accuracy. According to the experiments, the inlet turbulence level $\tau_u$ mainly affects the laminar part of the boundary layer and as a consequence, figure 5a exhibits a lower level of heat transfer on the pressure surface. This observation cannot be confirmed using the present runs, since the difference between the predicted heat transfer rates on the pressure side is not significant. The authors believe that the imposed value of $(\mu_T/\mu)$
Fig. 6 Exit flow angle for the highly loaded transonic guide vane cascade.

The inlet turbulence ratio at the inlet is incompatible with the inlet level of turbulence intensity and their combination leads to an unrealistic set of inlet $k$ and $e$ values. Figures 5c (low Re) and 5d (high Re) in comparison with the already examined figure 5a (mild Re), show the effect of the Reynolds number on the blade heat transfer for a common inlet $\tau_w$ value ($\tau_w=1\%$). As the Reynolds number increases, the overall level of heat transfer increases too and this could be seen in the aforementioned figures. On the suction side, high values of the heat transfer are identified in all figures, due to important viscous phenomena in this region; the finite size of the blade trailing edge leads to a considerable local acceleration which explains the increase in the heat flux.

Figure 6 summarizes the predicted exit flow angles, which are compared with the corresponding experimental data. The comparison is performed in terms of the $M_{21s}$ for two different Reynolds numbers $Re=5.10^5$ and $Re=10^6$. Numerical predictions are very close to measurements with an accuracy of approximately half a degree.

The SE1050 Transonic Steam Turbine
The second case examined, was the SE1050 cascade, designed for the last stage of a SKODA steam turbine. Cascade tests with air were reported by Stastny and Safarik (1990) while flow visualizations obtained by Schlieren and interferometric techniques, were also available. The detailed geometry and test conditions are available in the above reference.

Among the flow conditions examined in the original paper, two of them corresponding to transonic flows were considered herein. Thus, calculations were performed for two different isentropic exit Mach numbers (a) $M_{21s}=1.189$ and (b) $M_{21s}=0.906$. The calculations were carried out using the same H-type grid illustrated in figure 7. The grid has 125x71 nodes and was appropriately clustered close to the solid walls,
The computational grid for the SE1050 steam turbine cascade.

Fig. 7 The computational grid for the SE1050 steam turbine cascade.

A blowup of the grid shown in Fig. 7, close to the leading edge.

Fig. 8 A blowup of the grid shown in Fig. 7, close to the leading edge.

resulting to a $y^*$ value of less than 1.5. A blowup of the grid close to the leading edge is shown in figure 8. For the two cases, the stagnation conditions were taken to be $T_{tot}=290$ K and $P_{tot}=99000$ N/m² and the incidence is set to zero.

Figure 9 presents the iso-Mach contours for the more interesting case ($M_{2,15}=1.189$). By comparing this figure with the interferometric picture of the flow, which exists in the original paper, an accurate prediction of the general flow characteristics (location of the sonic line and the shock waves) is observed. It must be taken into account that the back pressure in this case was higher than the one corresponding to the limit loading. Thus, one of the shocks springing from the trailing edge remains in the passage and reflects on the flat part of the suction side. The same case was also examined by the authors, using a C-type grid and the Baldwin-Lomax turbulence model (Giannakoglou et al., 1991) and provided the same conclusions. A direct comparison between the predicted and measured critical Mach number distributions along the pressure and suction sides of the blade for the two flow cases, are presented in figures 10a and 10b. The agreement between predictions and measurements are very satisfactory, even along the suction side where the various local peaks of the critical Mach number distributions, were successfully captured.

Calculated iso-Mach contours for the SE1050 steam turbine cascade (case a, increment 0.05).

Fig. 9 Calculated iso-Mach contours for the SE1050 steam turbine cascade (case a, increment 0.05).

The Low Supersonic Compressor Cascade MCA

The last case examined concerns the flow through the low supersonic compressor cascade MCA (Starken and Schreiber, 1990). The experimental investigation was carried out in the Transonic Cascade Wind Tunnel of DFVLR. The blade section corresponds to the rotor midspan section of the axial compressor test case E/CO-4. The blade chord was 90mm, the stagger angle was 48.51 deg and the solidity was equal to 1.66085. The H-type grid which was used, had 155x55 nodes and it is presented in figure 11; the use of an H-type grid was the most appropriate due to the thin leading and trailing edges of the blade.

The inlet Mach number was $M_1=1.086$ and the inlet flow angle was 58.5 deg. Using the data provided in the original paper, the exit isentropic Mach number was calculated and used as input for the present calculation. This resulted to an increased inlet Mach number, compared to the one previously mentioned. Figure 12 presents the predicted iso-Mach contours; a shock appears upstream of the leading edge, which is almost
Fig. 10 Critical Mach number distribution along the blade for cases (a) and (b).

Fig. 11 The grid for the low supersonic compressor cascade MCA.

Fig. 12 Calculated iso-Mach contours for the low supersonic compressor cascade MCA (increment 0.05).

Fig. 13 Isentropic Mach number distribution along the blade for the low supersonic compressor cascade MCA.

perpendicular to the blade mean-line and reaches the suction side at approximately 40%. The thickening of the boundary layer which develops along the suction side after its interaction with the shock wave, is clear. Figure 13 shows the predicted and measured isentropic Mach number distributions on the blade surface. The level of the isentropic Mach number on the suction side, upstream of the shock wave, is higher compared to the experimental data, as discussed previously. Figure 14 shows the pitchwise Mach number distribution at an axial distance 45% downstream of the trailing edge, where the predicted Mach number in the heart of the wake appears to be much lower than the measured one.
Fig. 14 Mach number distribution at an axial position 45% downstream of the low supersonic cascade MCA.

CONCLUSIONS

An explicit time-marching fractional-step scheme was developed for the solution of the Favre-averaged Navier-Stokes equations in two-dimensional cascades. Turbulence was modelled using the low-Reynolds version of the k-e model. Mean-flow and turbulence equations are solved in a coupled way, through an appropriate sequence of one-dimensional operators assuring second order accuracy in time. The main conclusion of the present paper is the ability of explicit methods to handle the two-equation turbulence models, provided that a particular treatment of the source terms is implemented. This particular treatment is, in fact, a semi-implicit scheme which distinguishes positive and negative entries. A new form of the second- and fourth-order artificial dissipation scheme was introduced for the turbulence equations, which protects the turbulent energy dissipation from becoming negative close to solid walls. An additional advantage of the proposed method is the less strict stability criterion, with respect to other 2-D explicit solvers. From the programmer's point of view, the method is very simple and particular conditions, like periodicity, were easily implemented.

Three different two-dimensional cascades were used in order to validate the developed software. Both compressor and turbine cascades, in transonic flow conditions, were examined using either H- or C-type meshes. In all cases, the shock systems and other general flow characteristics were captured accurately.

ACKNOWLEDGEMENTS

A part of the work presented in this paper was funded by the BRITE Project P-1529, titled "Full Navier-Stokes equations applied to Turbomachinery problems".

REFERENCES


