3-D implicit Navier-Stokes solver for internal turbulent compressible flows
V. Michelassi, F. Martelli

To cite this version:
V. Michelassi, F. Martelli. 3-D implicit Navier-Stokes solver for internal turbulent compressible flows. Journal de Physique III, EDP Sciences, 1993, 3 (2), pp.223-235. <10.1051/jp3:1993128>. <jpa-00248917>

HAL Id: jpa-00248917
https://hal.archives-ouvertes.fr/jpa-00248917
Submitted on 1 Jan 1993

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L’archive ouverte pluridisciplinaire HAL, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d’enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.
3-D implicit Navier-Stokes solver for internal turbulent compressible flows

V. Michelassi and F. Martelli

Energy Engineering Dept., University of Florence, Via S. Marta 3, Florence, Italy

(Received 15 February 1991, accepted 12 November 1992)

Abstract. — A new code aimed at the solution of three-dimensional internal laminar and turbulent compressible flows is presented. The solution procedure is based on the scalar form of the approximate factorization method. The Navier-Stokes equations in mass-averaged form are iterated in time until a steady solution is reached. Evidence is given to the implicit treatment of diffusive terms that proved able to speed up convergence and enhance the algorithm robustness. Turbulence effects are accounted for by the Baldwin-Lomax turbulence model and the $q - \omega$ two-equation model. For the first, an investigation on the model behavior in case of multiple boundaries is performed. The numerical code is validated by computing the flow in the Stanitz 90-deg turning elbow. Finally the code is applied to the computation of the flow in the VKI LS-82 three-dimensional linear cascade.

Introduction.

With the advent of more and more powerful supercomputers, the numerical solution of three-dimensional turbulent flows became possible [1, 14]. Although it is well known that lower order turbulence closures fail to reproduce secondary motions of Prandtl’s second kind (turbulence driven), they normally succeed in predicting secondary flows of the first kind (pressure driven). Therefore, for many complex configurations it is possible to obtain fairly accurate results with zero and two-equations turbulence models with a reasonable prediction of pressure driven secondary flows.

For three-dimensional blade-passage flows, a correct prediction of the wake behavior has been obtained by Yokota [2] by means of the standard high-Reynolds-number form of the $k - \varepsilon$ two-equation model with the wall function approach. For practical flow configurations, it is necessary to face quite long computing times mainly because of the large number of points usually required for a detailed description of the flow field. Moreover, the non-linearities associated with nearly all the turbulence models can play a significant role in slowing down the convergence rate to the steady state solution. In two dimensions, a wide variety of flow

---

conditions have been accurately solved by means of low-Reynolds-numbers forms of the $k - \varepsilon$ model (LR) in which the effect of laminar viscosity is explicitly accounted for [3-5]. In nearly all the flow conditions investigated, these forms proved to be more accurate than the standard high-Re formulation, provided that a sufficient number of grid points are located inside the viscous and buffer layers. Secondary motions and losses are mainly driven by what happens close to walls so that a correct description of this flow region is crucial for an accurate simulation of the flow pattern. Rodi [5] found that the use of the LR forms could predict secondary flows normally lost with the high Reynolds number form. Unfortunately, the first author [3] found some of these forms extremely stiff from a numerical point of view. The stiffness was mainly caused by the low-Reynolds-number effect terms in which exponential functions are introduced to model the wall effects. From this standpoint, it appeared worthwhile investigating some features of turbulence models for internal flows using an implicit algorithm. Since complex flow patterns, such as separation and viscous effects, are expected in internal flows, the implicit approach was selected to increase the robustness and convergence rate of the numerical procedure when using zero and two-equation turbulence models.

Description of the algorithm.

NAVIER-STOKES EQUATIONS. — The Boussinesq hypothesis relates the turbulent shear stresses to the mean strains via the « eddy viscosity ». Under this assumption, the three-dimensional mass-averaged compressible Navier-Stokes (N-S) equations can be written in divergence form and in a generalized curvilinear coordinate system $\xi, \eta, \zeta$. The resulting set of equations can be written in vector form:

$$\frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial \mathbf{E}}{\partial \xi} + \frac{\partial \mathbf{F}}{\partial \eta} + \frac{\partial \mathbf{G}}{\partial \zeta} = \frac{\partial \mathbf{E}_v}{\partial \xi} + \frac{\partial \mathbf{F}_v}{\partial \eta} + \frac{\partial \mathbf{G}_v}{\partial \zeta} + \mathbf{H}$$

where $\mathbf{Q}$ is the vector of unknowns, $\mathbf{E}$, $\mathbf{F}$, $\mathbf{G}$ are the flux vectors of the convective terms, $\mathbf{E}_v$, $\mathbf{F}_v$, $\mathbf{G}_v$ are the flux vectors of the diffusive terms and $\mathbf{H}$ is the sink-source terms vector [6]. According to the Boussinesq assumption, the diffusion coefficients for momentum and energy are defined as follows:

$$\mu_{\text{eff}} = \mu_{\text{I}} + \mu_{\text{t}}; \quad \mu_h = \frac{\mu_{\text{I}}}{Pr_{\text{I}}} + \frac{\mu_{\text{t}}}{Pr_{\text{t}}}$$

in which $\mu_{\text{I}}$ is the laminar viscosity, that was considered independent of the static temperature, and $\mu_{\text{t}}$ is the turbulent viscosity obtained from the turbulence model. In this set of calculations the turbulent Prandtl number, $Pr_{\text{I}}$ was set equal to 0.90 and the laminar Prandtl number, $Pr_{\text{t}}$, was 0.72.

The flux vectors are discretized using centered finite differences. The metrics are obtained from a chain rule expansion of $x_\xi, x_\eta, x_\zeta, y_\xi, y_\eta, y_\zeta, z_\xi, z_\eta, z_\zeta$. When the centered discretization is used for the metrics in 3D, it can be shown that the metrics invariants are not satisfied. To avoid large discretization error we followed the metrics averaging procedure described in [7].

BALDWIN-LOMAX TURBULENCE MODEL FOR MULTIPLE BOUNDARIES. — The two-layer algebraic model developed by Baldwin and Lomax [8] divides the flow field into an inner layer close to the wall, in which viscous effects are dominant, and an outer layer. The influence of the solid wall is damped according to the Van Driest exponential function. The model has been
originally developed for a single boundary layer, but in three-dimensional internal flows the presence of multiple boundaries must be considered when computing the turbulent viscosities. While the inner, or viscous layer, is driven by what happens on the closest wall only, for the outer layer an averaging procedure is necessary to account for the various wall effects. In the present set of calculations three approaches have been examined to account for the presence of more than one boundary layer.

- Wall Treatment 1. — Only the geometrically closest wall is considered to compute the outer viscosity. No averaging procedure is implemented.

- Wall Treatment 2. — The inner layers are driven by the closest wall only, while the viscosity in the outer layer is computed with a simple weighted average according to the inverse of each wall distance as follows

\[
\mu_{t, \text{outer}} = \frac{1}{N} \sum_{w} \frac{\mu_{w, \text{outer}}}{W_w}
\]

where \( W_w \) is the wall distance, \( w \) is the wall number and \( N \) is number of walls present in a cross section.

- Wall Treatment 3. — While computing the inner layer viscosities with the closest wall contribution only, the viscosity in the outer layer is computed as a simple weighted average according to the inverse of the value of the Van Driest damping expression for each wall.

**q - \( \omega \) Two-Equation Turbulence Model.** — In a previous investigation Michelassi [3] tested some LR forms. The results showed that these forms could be numerically stiff, mainly on account of the correction terms introduced to model the low-Re effects. Moreover, an initial consistent profile for the turbulent quantities is generally necessary to start the calculations. A first attempt to implement the Chien's and the Rodi's two-layer low-Reynolds number forms of the \( k - \omega \) mode [3] did not bring to any converged result mainly because of difficulties in specifying both a sufficient mesh refinement and proper initial profiles for complex three-dimensional flows. Coakley [9], reassembling the Jones and Launder LR model, proposed the \( q - \omega \) two-equation model in which the effect of molecular viscosity is directly modelled. This formulation ensured a better numerical behavior as compared with other LR formulations. In the transported quantities, \( q = \sqrt{k} \) and \( \omega = \varepsilon/k \), \( \varepsilon \) represents the isotropic part of the dissipation rate: this quantity does not account for any non-isotropic effect (for example, the presence of a wall) and tends to zero on solid boundaries (Conversely, the total dissipation rate tends to a finite value related to the wall shear stress). This choice allows one to use \( \omega \) as an unknown since, assuming that both \( k \) and \( \varepsilon \) are going to zero at the wall with the same speed, \( \omega \) tends to a finite value.

**Approximate Factorization: Scalar Form.** — The approximate factorization method proposed by Beam and Warming [10] splits an \( n \)-dimensional operator into the product of \( n \) one-dimensional operators. This technique provides a strong link between the equations insofar as they are solved fully coupled. The main drawback of this method lies in the necessity of time-consuming block tri- or penta-diagonal matrix inversion. The problem becomes more evident in three dimensions where the coupled solution yields a \( 5 \times 5 \) block tridiagonal matrix. In order to make the algorithm more efficient and still maintain its strong implicit nature, Pulliam [7, 11] proposed the following scalar form of the approximate factorization:
\[ T_{\xi} \cdot [I + \Theta \Delta t (\delta_{\xi} A_{\xi})] \cdot N \cdot [I + \Theta \Delta t (- H/2 + \delta_{\eta} A_{\eta})] \cdot P \cdot [I + \Theta \Delta t (- H/2 + \delta_{\xi} A_{\xi})] \cdot T_{\xi}^{-1} \cdot \Delta Q = \text{RHS} \]  

\[ \text{RHS} = \Delta t \left( - \frac{\partial E}{\partial \xi} - \frac{\partial F}{\partial \eta} - \frac{\partial G}{\partial \zeta} + \frac{\partial \tilde{E}_v}{\partial \xi} + \frac{\partial \tilde{F}_v}{\partial \eta} + \frac{\partial \tilde{G}_v}{\partial \zeta} + \tilde{H} \right) \]

in which \( I \) is the identity matrix, \( \Theta \) is a parameter that weights the explicit-implicit nature of the space operator (in the present calculations, since the steady state solution was sought, \( \Theta = 1 \) was used), \( \delta \) is a centered difference operator, \( \Delta t \) is the time step, \( \Delta Q = Q^{n+1} - Q^n \), \( Q = J \tilde{Q} \cdot A_{\xi} \), \( A_{\eta} \), \( A_{\zeta} \) are the convective Jacobians eigenvalues, \( T \) are the eigenvector matrices and \( N = T_{\xi}^{-1} T_{\eta} \), \( P = T_{\eta}^{-1} T_{\zeta} \). For the NS solution \( \tilde{H} = H = 0 \). The two equations of the \( q - \omega \) model are solved in a sequential manner and decoupled from the flow variables mainly on account of the fact that the coupling is provided only by the diffusive terms coefficients and the sink and source terms. Due to this choice the scalar three-diagonal algorithm was implemented for the turbulence model solution. The only difference with respect to the solution of the N-S equations is the presence of the sink-source vector \( \tilde{H} \) in equation (1). \( H_{j} \) is computed neglecting the contribution of the damping function \( D \). Its form for the two \( q \) and \( \omega \) equation is:

\[
H_{j}^q = \frac{\partial H^q}{\partial (\rho J^{-1} q)} = \frac{1}{2} \frac{C_{1}}{\omega} DS - \frac{1}{3} P_d - \frac{\omega}{2}
\]

\[
H_{j}^\omega = \frac{\partial H^\omega}{\partial (\rho J^{-1} \omega)} = - \frac{2}{3} C_1 D P_d - C_2 2 \omega
\]

in which \( S \) and \( P_d \) are production terms [6]. In place of their exact form, Coakley proposes an approximation of the Jacobians based on the turbulent viscosities that should ensure the dominance of the main diagonal. Figure 1 shows the comparison of the convergence rates of

![Convergence tests for implicit treatment of q-\omega sink-source terms.](image)
every single variable obtained without any sink or source terms Jacobian, with the exact Jacobian and with Coakley's approximation in a typical internal flow geometry. Surprisingly, there is no big gain in introducing the Jacobian in the implicit side of the operator since with the three formulations it was always possible to obtain the same residual reduction in approximately 300 iterations.

Implicit treatment of diffusive terms. — For inviscid solutions, the only assumption done to derive the algorithm is that the differentiation of the eigenmatrices is neglected in (1). Additionally, when solving the N-S equations, the diffusive Jacobians are normally assumed to be negligible. This does not cause any stability or convergence problems for external flows where the diffusion dominated region is small, but it may cause troubles for internal flows where the diffusion dominated region is large. According to this, for internal flows it is convenient to introduce an approximation of the eigenvalues of the diffusive terms Jacobian and put it into equation (1). Two forms of this approximation have been considered.

• Pulliam's approximation. — Pulliam [10] proposes the following approximation of the diffusive Jacobian eigenvalues. In the \( \xi \) direction the approximation is:

\[
A_\xi^* = (\rho \mu_{\text{eff}} \text{Re}^{-1} (\xi_x^2 + \xi_y^2 + \xi_z^2) J^{-1}) \cdot D [1, 1, 1, 1]
\]  

where \( D \) stands for main diagonal only. In the present set of calculations it was found convenient not to weight the eigenvalues with the Jacobian of the coordinate transformation \( J^{-1} \). These expressions are included on the implicit side of (1): for instance, the \( \xi \) direction implicit operator is:

\[
[I + \Theta \Delta t (\delta_\xi A_\xi - \delta_\xi A_\xi^*)]
\]

• Present approximation. — The exact form of the diffusive terms Jacobian can be computed from the related flux vectors. For the \( \xi \) sweep the main diagonal of such matrix may be conveniently approximated as:

\[
A_\xi^* = D [0, \alpha_\xi, \alpha_\xi, \gamma \text{Pr}^{-1} \alpha_\xi]
\]  

in which:

\[
\alpha_\xi = \mu_{\text{eff}} \text{Re}^{-1} J^{-1} (\xi_x^2 + \xi_y^2 + \xi_z^2) \frac{\partial (\rho^{-1} J)}{\partial \xi}
\]

Regarding the extra-diagonal terms as negligible, the previous diagonal matrices may be a good approximation of the diffusive terms Jacobians and can be put into the implicit side of (1): for instance, the \( \xi \) direction implicit operator is:

\[
[I + \Theta \Delta t (\delta_\xi A_\xi - \delta_\xi A_\xi^*)]
\]

A comparison of the two approaches was performed on a simple straight channel geometry at \( \text{Re} = 1,000, \ M_{\text{inlet}} = 0.3 \) in laminar flow regime. In this flow configuration there were no differences in convergence rate between (2) and (3) and it was also possible to drop the diffusive terms on the implicit side of (1) without altering convergence. Differences started to appear at \( \text{Re} = 50 \) because of the highly diffusive nature of the flow. Figure 2a shows the best convergence history of the algorithm without any implicit treatment of the diffusive terms that was obtained at CFL = 5 (the lower curve refers to the averaged residuals, while the upper one refers to the maximum). In figures 2b and 2c the introduction of a diffusive Jacobian shows a gain in convergence rate. Nevertheless, there are no appreciable differences between the two
Fig. 2. — Convergence tests for implicit treatment of diffusive terms.

convergence rates obtained at CFL = 10 since both the curves, (b) and (c) show nearly the same slope. At least for this class of flows the approximate implicit treatment of diffusive terms given by equation (3) did not prove to be more efficient than (2). Further testing is necessary in order to verify this result at higher Mach numbers when differences in the convergence between (2) and (3) may appear.

Results.

Stanitz elbow. — The flow in the accelerating rectangular elbow with 90° turning and variable cross section described by Stanitz [12] has been computed. This test case provides a good set of measurements including wall pressure distribution and visualization of secondary flows. The shape of the elbow was analytically computed by Stanitz to give no separation with a strong area reduction and a specified pressure distribution on the side wall under incompressible flow conditions. Among the various flow conditions investigated in [12], we selected the one with $M_{exit} = 0.4$ and with no spoiler at the duct inlet with a thin initial boundary layer. The calculations were performed on a $51 \times 41 \times 31$ grid. The Reynolds number, Re, based on the total conditions at the inlet section is approximately $2.5 \times 10^6$. These calculations were mainly aimed at the proper prediction of the wall pressure distribution that is affected by the growth of secondary velocities and a qualitative comparison of the secondary flows predicted by the Baldwin-Lomax zero-equation model and the $q-\omega$ two equation model.

The first set of tests concerns the Baldwin-Lomax model with different wall treatments. With the experimental total pressure profile specified at the inlet section the computed static pressure profiles are compared with the measurements in figure 3. The plots refer to the static pressure distribution in the section corners on the side wall and the symmetry plane defined as:

$$P_s = \frac{P - P_{exit}}{P_{\text{total}} - P_{exit}}$$

The wall treatments # 1 and # 2 do not show large differences even if the two approaches are considerably different. For both the techniques the agreement with experimental results is fairly good on the pressure side of both the side wall and the symmetry plane. The suction side
on both the planes shows that the static pressure is overestimated. This is probably due to the presence of computed secondary flows being much stronger than the experimental ones. The computed pressure drop induced by the presence of the bend and the strong flow acceleration is smoother than the measured one. No big differences are found for the pressure side of both the side wall and the symmetry plane while, with wall treatment \# 3, the suction side appears in better agreement with experiments. Still the pressure minimum located at \( S = 2 \), where secondary velocities on the cross section start to develop, is not captured. This test suggests that the averaging technique based on the Van Driest damping expression for the mixing length can give reasonable predictions.

The flow simulation with the \( q - \omega \) model required an investigation on the inlet turbulence level, the results of which are summarized in figure 4. The use of the \( q - \omega \) model brings some improvement in the agreement with experiments mainly on the suction side of both the symmetry plane and side wall. The direct comparison of the turbulent calculations with the inviscid computation (eul), in which the same pressure distribution is found on both the side wall and the symmetry plane due to the absence of secondary motions, shows large differences on the suction side only starting from \( S = 2 \) where experiments deviate from the inviscid solution. The 1% turbulence level, not reported in figure 4, appeared to bring still a too high momentum diffusion so that this level was progressively decreased from 0.5% to 0.1%. The static pressure distribution on the pressure wall is mainly driven by convective phenomena, while the distribution on the suction wall is largely influenced by diffusion processes: this is the reason why the pressure side distribution is always well reproduced and is nearly independent of the inlet turbulence level. The final result obtained with 0.1% turbulence level shows a fairly good agreement with experiments on both the suction and pressure sides and seems to reproduce quite correctly the location and the influence of secondary flows.

VKI LS-82 CASCADE. — The Von Karman Institute blade cascade LS-82 has been proposed for testing flow solvers in subsonic and supersonic flow conditions: details on the blade geometry and test conditions may be found in reference [13]. The linear cascade is originally three-dimensional, but the reported isentropic Mach number \( (M_{is}) \) refers only to the midspan section. The experimental flow conditions are specified through \( M_{is} = 0.7, 0.84, 1.00, 1.12, \)

|Fig. 3. — \( C_p \) distribution by the Baldwin-Lomax turbulence model. |
Fig. 4. — $C_p$ distribution by the $q - \omega$ turbulence model.

1.43. Among these we selected the first shock-free configuration at $M_{is} = 0.7$ that was computed by both Baldwin-Lomax and the $q - \omega$ models, while computing the supersonic conditions obtained at $M_{is} = 1.12$ and 1.43 by the Baldwin-Lomax models with wall treatment # 3. This test allowed verifying the robustness of the method for turbine cascades in which further numerical problems are found due to the presence of periodic boundaries and stagnation points.

The computations were carried out on two grids, a coarse $38 \times 18 \times 13$ one and a refined $75 \times 35 \times 25$ grid that is shown in figure 5a. The mesh was generated with a mixed elliptic-algebraic approach [3] that gives non periodic grids. This type of grid needs a special treatment of the periodic boundaries. In the present calculations the flow variables on the two periodic boundaries were computed separately. At the end of every iteration the flow variables on these boundaries were made to coincide by taking an average of the computed values. This explicit way to ensure periodicity slightly slowed down the convergence rate. Figures 5b, c, d show the local Mach number isolines computed with the Baldwin-Lomax model for the three $M_{is}$ values investigated here. The $M_{is} = 0.7$ configuration is shock free and the boundary layer on the blade suction side remains very thin. With $M_{is} = 1.12$ a shock is present at approximately 60% of the axial chord. This results in a growth of the boundary layer thickness that may be easily seen in the plot. It is interesting to observe that the use of non periodic grids with an interpolated periodic condition ensures a perfect matching between the upper and lower sides of the grid.

The strong shock encountered for $M_{is} = 1.43$ yields a very complex flow pattern with a long separated region (Fig. 6) that starts at the shock and propagates to the trailing edge. While flow separation is expected after a strong shock, the measurements [13] show that the separated
Fig. 5. — Grid and Local Mach Number isolines. a) $75 \times 35 \times 25$ grid. b) $M_{\infty} = 0.7$. c) $M_{\infty} = 1.12$. d) $M_{\infty} = 1.43$. 

JOURNAL DE PHYSIQUE III — T. 3, N° 2, FEBRUARY 1993
Fig. 6. — Flow separation at $M_{is} = 1.43$.

region, if any exists, does not propagate to the trailing edge. This is probably due to the algebraic turbulence model that predicts high turbulence viscosities in the region after the shock. The viscous formulation allows capturing the two counter-rotating vortices at the blade trailing edge, despite the not highly refined grid in the flow direction. In figure 7 this recirculation is shown for $M_{is} = 1.12$ where the shock does not induce the flow separation on the suction side experienced at $M_{is} = 1.43$.

The $M_{is}$ distribution over the blade midspan surface along the chord are given in figures 8a, b, c, d for the three flow conditions. Figure 8a compares the profiles computed with the algebraic model at $M_{is} = 0.7$ with the measurements [13]. While the pressure side distribution always agrees with experiments, on the suction side the coarse grid predicts the peak in velocity after the experimental peak position, while the refinement ensured by the $75 \times 35 \times 25$ grid allows for placing the peak in good agreement with measurements. The $M_{is}$ levels after 30 % of the chord are underestimated by the refined grid and overestimated by the coarse grid: this is due to the fact that the boundary layer is very poorly modelled by the coarse grid so that the code nearly behaves like an inviscid solver. Figure 8b shows the comparison of the $M_{is}$ distribution obtained by the Baldwin-Lomax and the $q - \omega$ model, for which the inlet turbulence level, as specified in the experiments, is 1 %. The two-equation model clearly shows a better fit with measurements after 30 % of the chord. On the blade nose the Baldwin-Lomax model seems to perform marginally better. In figure 8c the distribution obtained at $M_{is} = 1.12$ shows the presence of a shock computed with the refined grid.
Fig. 7. — Trailing edge separation.

Fig. 8. — a) $M_a = 0.7$ (Baldwin-Lomax). b) $M_a = 0.7$ ($q - \omega$). c) $M_{is} = 1.12$ (Baldwin-Lomax). d) $M_a = 1.43$ (Baldwin-Lomax).
Although on the pressure side the agreement with experiments is again good, on the suction side the experimental profile evidences a complex system of shocks that are probably caused by the presence of a small separated region that was not predicted by the model. Similar problems in the numerical prediction of such complex flow pattern have been encountered by many of the participants to the VKI workshop [13]. The same observations may be repeated for the $M_{is} = 1.43$ case (Fig. 8d) in which the shock magnitude is well reproduced by both the coarse and the refined grids. The refined grid ensures a better reproduction of the flow pattern in the nose region of the suction side. The fit with experiments after the shock is clearly poor. This is due to the presence of the large separated region, visible in figure 6, that gives a flow deceleration in the flow region. In the remaining part of the blade, the agreement with measurements is again quite good.

Figure 9 shows a projection on the tangential plane of the flow components in a cross section located at 50% of the chord. The presence of the passage vortex is well predicted in proximity to the side wall. Due to the large blade aspect ratio of the experimental tests the vertex is located close to the side wall and weakly affects the core flow region.

![symmetry plane](image)

**Fig. 9.** — Secondary flow visualization at 50% of the chord.

**Concluding remarks.**

The introduction of an approximate treatment of the diffusive terms in the scalar form of the approximate factorization method proved to increase convergence of the algorithm for internal flow configurations. Nevertheless, the diffusive Jacobians influence must be verified in a wider range of Mach numbers.

The Baldwin-Lomax turbulence model proved to give results in acceptable agreement with experiments provided that the presence of multiple boundaries is properly accounted for. The LR $q-\omega$ two-equation model allowed for computing complex three dimensional configurations giving a satisfactory description of the flow field with a manageable number of grid points and only a small increase of computational time with respect to the algebraic model.

This attempt to solve an accelerating cascade flow proves the robustness of the method, but further tests are necessary to calibrate the turbulence model response to strong surface curvature, and shock and trailing edge separation.
References


