HEAT TRANSFER COMPUTATIONS FOR TURBINE BLADE AIRFOILS

Jamel SLIMANI and Pascale KULISA

Laboratoire de Mécanique des Fluides et d’Acoustique UMR CNRS 5509
Ecole Centrale de Lyon
36, Avenue Guy de Collongue BP 163
69 131 ECULLY Cedex
France

ABSTRACT
The design and optimization of turbine blades subjected to high temperature flows require the prediction of aerodynamic and thermal flow characteristics. A computation of aerothermal viscous flow model has been developed suitable for the turbine blade design process. The computational time must be reduced to allow intensive use in an industrial framework. The physical model is based on a compressible boundary layer approach, and the turbulence is a one-equation model. Special attention has been paid to the influence of wall curvature on the turbulence modelling. Tests were performed on convex wall flows to validate the turbulence model. Turbine blade configurations were then computed. These tests include most difficulties that can be encountered in practice: laminar-turbulent transition, separation bubble, strong accelerations, shock wave. Satisfactory predictions of the wall heat transfer are observed.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_f )</td>
<td>skin friction coefficient</td>
</tr>
<tr>
<td>( h )</td>
<td>heat transfer coefficient</td>
</tr>
<tr>
<td>( h_t )</td>
<td>stagnation enthalpy</td>
</tr>
<tr>
<td>( k )</td>
<td>turbulent kinetic energy</td>
</tr>
<tr>
<td>( l_\nu )</td>
<td>length scale, mixing length</td>
</tr>
<tr>
<td>( P_\nu )</td>
<td>turbulent Prandtl number</td>
</tr>
<tr>
<td>( P_c )</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>( Q )</td>
<td>additional variable</td>
</tr>
<tr>
<td>( R_c )</td>
<td>curvature radius</td>
</tr>
<tr>
<td>( R_e )</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>( R_e_1 )</td>
<td>Reynolds number, based on ( \sqrt{k} ) and ( \gamma )</td>
</tr>
<tr>
<td>( R_e_2 )</td>
<td>Outlet Reynolds number, based on the outlet conditions and the chord length</td>
</tr>
<tr>
<td>( R_i )</td>
<td>Richardson number</td>
</tr>
<tr>
<td>( s )</td>
<td>curvilinear abscissa along the blade</td>
</tr>
<tr>
<td>( s_{max} )</td>
<td>maximum value of ( s )</td>
</tr>
<tr>
<td>( S_t )</td>
<td>Stanton number</td>
</tr>
<tr>
<td>( T_t )</td>
<td>stagnation temperature</td>
</tr>
<tr>
<td>( T_f )</td>
<td>freestream turbulence intensity</td>
</tr>
<tr>
<td>( u )</td>
<td>streamwise velocity component</td>
</tr>
<tr>
<td>( U_e )</td>
<td>streamwise inviscid velocity</td>
</tr>
<tr>
<td>( U_t )</td>
<td>friction velocity</td>
</tr>
<tr>
<td>( \bar{u}\bar{v} )</td>
<td>Reynolds stress</td>
</tr>
<tr>
<td>( v )</td>
<td>normal to the wall velocity component</td>
</tr>
<tr>
<td>( x )</td>
<td>streamwise coordinate</td>
</tr>
<tr>
<td>( \gamma )</td>
<td>normal coordinate</td>
</tr>
<tr>
<td>( \gamma^+ )</td>
<td></td>
</tr>
<tr>
<td>( \delta )</td>
<td>boundary layer thickness</td>
</tr>
<tr>
<td>( \varepsilon )</td>
<td>dissipation rate</td>
</tr>
<tr>
<td>( \psi )</td>
<td>stream function</td>
</tr>
<tr>
<td>( \mu )</td>
<td>dynamic viscosity</td>
</tr>
<tr>
<td>( \mu_t )</td>
<td>eddy viscosity</td>
</tr>
<tr>
<td>( \nu )</td>
<td>kinematic viscosity</td>
</tr>
<tr>
<td>( \nu_t )</td>
<td>turbulent kinematic viscosity</td>
</tr>
<tr>
<td>( \rho )</td>
<td>density</td>
</tr>
<tr>
<td>( \rho_e )</td>
<td>inviscid flow density</td>
</tr>
<tr>
<td>( \tau )</td>
<td>additional variable</td>
</tr>
</tbody>
</table>

INTRODUCTION
High performance gas turbines require a high turbine inlet temperature. The temperature levels reached in the first stages of turbines can lead to severe mechanical problems limiting the life of the components. Determining the best compromise between the thermal stresses and the
aerodynamical performances is a major objective of turbine designers (Vuillez and Petot, 1994) and (Whalen, 1995). The prediction of these flows using three-dimensional Navier-Stokes computations is now possible and current, due to modern computer capabilities (Amari and Amine, 1992), (Fougères and Heider, 1994), (Boyle and Giel, 1995). Nevertheless, the memory use and the CPU time are very important. This point is very critical for the thermal predictions which require very fine mesh near the wall. Thus, these types of calculation are not systematically used during the design process.

With this goal in mind, a computational code was developed in order to predict the heat transfers on the turbine blades (Kulisa et al., 1990). The CPU time must be reduced allowing intensive use while taking into account the majority of the physical phenomena. The present approach is based on the boundary layer concept. Although turbine blade flows are characterized by strong accelerations, adverse pressure gradients or separated regions may occur. A viscous-inviscid flow interaction was included to compute these regions. Special attention has been paid to ensure the numerical stability and the robustness of the calculation.

Wall curvature strongly affects the turbulent flow characteristics. Consequently, there is a significant change in the heat transfer level. A one-equation turbulence model is used and modified to account for curvature effects.

The code was calibrated and validated for various flow configurations to assess the accuracy of the aerodynamic results (Kulisa et al., 1992). In this paper, detailed aerothermal validations are made for convex wall configurations. Then, two turbine test cases are presented. The calculated heat transfer coefficient is compared with experimental data for a rotor turbine blade and a high pressure stator blade.

PHYSICAL MODEL

The viscous flow model is based on the two-dimensional turbulent Reynolds-averaged Navier-Stokes equations. A thin-layer approximation is used to eliminate the diffusion terms in the x-direction. The equations are written in a compressible form. The conservation equation for the stagnation enthalpy \( h_t \) is employed to incorporate the thermal effects. Moreover, a change of variables substitutes the stream function \( \psi \) for the velocity normal component:

\[
\rho v = - \frac{\partial \psi}{\partial x}
\]  
(1)

Thus, the partial differential equations for the viscous flow can be written in the following form:

\[
\rho u = \frac{\partial \psi}{\partial y}  
\]  
(2)

\[
\frac{\partial \psi}{\partial y} u_x - \rho \frac{\partial \psi}{\partial x} u_y = \rho U_e \frac{du}{dx} + \frac{\partial}{\partial y} \left[ \mu \frac{\partial u}{\partial y} - \rho u'v' \right]  
\]  
(3)

\[
\frac{\partial \psi}{\partial y} h_{t,x} - \rho \frac{\partial \psi}{\partial x} h_{t,y} = \frac{\partial}{\partial y} \left[ \mu h_{t,y} - q_v - \rho \frac{\partial h_{t,y}}{\partial y} \right]  
\]  
(4)

where \( q_v \) is the heat flux, expressed as a function of the static temperature by Fourier's law. The density \( \rho \) is not considered to be a global unknown. The nature of the fluid does not appear explicitly in the equations. This feature may be used in order to treat fluids other than perfect gases, with no modification to the equations. However, for the applications presented in this paper, the density will be calculated from the perfect gas law as a function of the stagnation enthalpy and the velocity. The molecular viscosity \( \mu \) is determined by Sutherland's law.

\( U_e \) is the velocity at the outer edge of the boundary layer. For the classical boundary layer procedure, the so-called direct mode, the inviscid velocity \( U_e \) is prescribed. This procedure is not stable when strong decelerations or separations occur, and this kind of flow may locally appear on a turbine blade. Thus, the inviscid velocity \( U_e \) is considered to be an additional unknown. A supplementary equation is solved which simulates the inviscid flow response to the viscous boundary layer presence. In fact, this interaction equation relates the inviscid velocity \( U_e \) to the development of the boundary layer. In the code, a switch allows a choice of the computation mode. If no separation regions exist, the direct mode is used and \( U_e \) is prescribed from the experimental distribution of the pressure along the blade surfaces. If a separation exists, \( U_e \) becomes a supplementary unknown and the coupling equation is solved together with the boundary layer equations. The unknowns are \( \psi, u, h_t, \{U_t\} \).

Classical boundary conditions are applied. No-slip condition is prescribed at the wall. For the thermal conditions, either the enthalpy or the heat flux distributions may be prescribed along the wall. At the outer edge, continuity conditions with the inviscid flow are ensured.

TURBULENCE CLOSURE

To close the system (2) to (4), the Reynolds stress \( u'v' \) and the turbulent heat flux \( \nu T' \) must be determined. The turbulence model is based on Boussinesq's hypothesis which relates the turbulent correlations to the gradients of the mean quantities. Thus, the algebraic eddy viscosity concept \( \mu_t \) is employed:

\[
-\nu u'v' = \mu_t \frac{\partial u}{\partial y} 
\]  
(5)

The curvature has an important effect on the flow characteristics. More particularly, the curvature affects the turbulence and the laminar-turbulent transition development. This leads to significant changes in the heat transfer rates. Therefore, the curvature effects must be taken into account in the model.

A convex wall produces a stabilizing effect, in that the turbulent kinetic energy is reduced. So and Mellor (1973) show lower levels of Reynolds stress throughout the boundary layer and particularly in the outer region. This reduction is immediate as soon as the curvature appears. Not only is the Reynolds stress dissipated but it is also cancelled.
by a negative production of \( \overline{uv} \). This leads to a decrease in the turbulent kinetic energy; these decreases are particularly significant in the outer of the boundary layer. In this region, the vortice length scales are related to the curvature radius and not to the boundary layer thickness. The inner layer remains a flat plate structure. Moreover, the laminar-turbulent transition is delayed on a convex surface. Gillis et al. (1980) extended the study to the recovery region, and show that recovery from curvature is very slow. 50 to 100 times the boundary layer thickness are needed to recover a flat plate structure, representing a memory effect on the flow turbulence.

On the other hand, a concave wall produces a destabilizing effect emphasizing the turbulent state of the flow. The laminar-turbulent transition moves upstream, and the flow may support stronger deceleration. Moreover, concave wall may lead to complex physical phenomena: Görtler vortices. The direct modelisation of the Görtler vortices cannot be incorporated in this simple boundary layer approach.

With regard to the thermal aspect, the curvature strongly affects the heat transfer rate. Thomann (1968), Mayle et al. (1979) observed 15% to 20% decrease in the heat transfer on convex surfaces relative to the flat plate case. Consequently, the Stanton number strongly decreases for convex wall and the concave wall raises it according to Gibson et al. (1981), Simon and Moffat (1983). In the recovery region, 15 to 20 boundary layer thickness downstream from a convex curvature, Simon and Moffat (1983) show that the Stanton number is still 15 to 20 percent below flat plate values.

For a mixing length model, Bradshaw (1969) and (1973) proposes a modification using a mixing length correction. This method is only valid for weak curved flows (\( \delta / R_c \leq 0.02 \)). For a two-equation model, Launder et al. (1977) modified the destruction of the dissipation rate \( \varepsilon \) by including a turbulent Richardson number. This model gives good results for boundary layer flows and channel flows. Nevertheless, the authors conclude that the computation effort is important for weak amelioration relative to a mixing length model.

In this article a one-equation model is used; this approach is midway between a mixing length model and a two-equation model. The transport equation of the turbulent kinetic energy \( k \) is solved:

\[
\frac{\partial k}{\partial \xi_j} = \frac{\partial}{\partial \xi_j} \left[ \left( \mu + \mu_k \right) \frac{\partial k}{\partial \xi_j} \right] + \nu \left( \frac{\partial u_i}{\partial \xi_j} + \frac{\partial u_j}{\partial \xi_i} \right) \frac{\partial u_i}{\partial \xi_j} - \varepsilon
\]  

The dissipation rate \( \varepsilon \) is calculated from \( k \) and a length scale \( L_1 \), by the well-known relation:

\[
\varepsilon = \frac{k^{3/2}}{C_f}
\]

in the outer part of the boundary layer.

\[C_f = \kappa R_c^{3/4} \]  

\[\kappa = 0.41 \text{ is the Von Karman constant} \]

\[A_c = 2 C_r \]  

\[C_r = 0.09.\]

The eddy viscosity is expressed from the velocity scale \( \sqrt{k} \) and a length scale \( L_1 \), by:

\[\nu_t = \rho C_r \sqrt{k} L_1\]

The length scale \( L_1 \) is determined by algebraic relations.

**Inner part of the boundary layer**

The length scale is related to the Richardson number, as in Bradshaw's proposal.

\[L_1 = L_{fp} \frac{1}{1 + B R_i} \]

where \( L_{fp} \) is the classical flat plate mixing length:

\[L_{fp} = C l y(1 - \exp(-\frac{R_e}{A+}) \]  

\[C_l y, A+ = 25 \]  

\[\beta \] is a constant equal to 7 for convex wall and 5 for concave wall.

The Richardson number is defined as the ratio of the Coriolis force to the viscous force. The particular form of the Richardson number used is discussed by Bradshaw (1969) and is given by:

\[\frac{2 \frac{u}{R_e}}{\frac{\partial u}{\partial y}} \]

\[R_i = \frac{R_e}{2 R_e} \]

\[R_i \] is an effective curvature radius, determined by solving a first-order drag equation which simulated the longitudinal curvature radius change:

\[\frac{d(1/R_e)}{ds} = 1 + \frac{1}{105} \left[ 1 - \frac{1}{R_e} \right] \]

This equation is used by Cebeci et al. (1978), Johnston and Eide (1976).

To avoid negative values of the length, (Gillis et al., 1980), Ibrahim (1987), the Richardson number is limited by:

\[R_i \leq 0.3 \frac{2 \delta}{U e R_e} \]

**Outer part of the boundary layer**

\[\frac{\nu}{R_e} = 0.0025 \tanh(345 \frac{\delta}{R_e}) \]

Boundary conditions are applied for the \( k \)-equation:

\[k = 0 \text{ at the wall} \]

\[\frac{\partial k}{\partial y} = 0 \text{ at the outer edge} \]

For the initialisation, Bradshaw's law is used:

\[k = \frac{\nu_t}{\sqrt{C_r} \frac{\partial u}{\partial y}} \]
The transition from laminar to turbulent flow is activated by introducing an intermittency function defined by Dhawan and Narasimha (1958). Thus, the transition location can be prescribed for the boundary layer computation.

By analogy with the molecular heat flux, the turbulent heat flux is related to the mean temperature (or enthalpy) gradient by a turbulent thermal diffusivity $\alpha_t$, $\alpha_t$ is expressed as a function of $\mu_t$:

$$\alpha_t = \frac{\mu_t}{Pr_t}$$  \hspace{1cm} (16)

$Pr_t$ is the turbulent Prandtl number, assumed to be a constant ($=0.9$).

NUMERICAL METHOD

The equations are discretized by the second order implicit scheme of Keller (1970). The main advantage of this scheme is the derivative order reduction in the equations. Consequently, only first order derivatives still exist. Keller’s scheme, applied to Eq. (2)-(4), leads to the introduction of two additional variables $\tau$ and $Q$, thus eliminating the second order derivatives of $u$ and $h_t$:

$$\tau = \mu \frac{\partial u}{\partial y} - \rho u' v'$$  \hspace{1cm} (17)

$$Q = \mu \frac{\partial u}{\partial y} - q_y - \rho v' h_t'$$  \hspace{1cm} (18)

The following system is then obtained:

$$\rho u_t \frac{\partial u}{\partial t} - \frac{\partial}{\partial x} \left( \frac{\partial u}{\partial x} \right) = \frac{\partial}{\partial y} \left[ \frac{\partial u}{\partial y} - \rho u' v' \right]$$  \hspace{1cm} (19)

$$\frac{\partial}{\partial t} \left( \frac{\partial h_t}{\partial y} \right) + \frac{\partial}{\partial x} \left( \frac{\partial h_t}{\partial x} \right) = \frac{\partial}{\partial y} \left[ \frac{\partial h_t}{\partial y} - q_y - \rho v' h_t' \right]$$  \hspace{1cm} (20)

$$\tau = \mu \frac{\partial u}{\partial y} - \rho u' v'$$  \hspace{1cm} (21)

$$Q = \mu \frac{\partial u}{\partial y} - q_y - \rho v' h_t'$$  \hspace{1cm} (22)

The final unknowns are : $[\psi, u, h_t, \tau, Q, (Ue)]$. This new system is more complicated from an algebraic point of view because there are more equations and more unknowns. Nevertheless, only first order derivatives exist, and the discretization on a box introduces only the variables at the four grid points of this box, leading to a very compact scheme. Because of the parabolic nature of the boundary layer equations, the system (19) to (23) is solved using a space-marching method.

COMPUTATIONAL RESULTS

The code was tested on various flows, laminar, transitional or turbulent, on flat plates or turbine blades. Detailed aerothermal results are presented for a turbulent boundary layer on convex wall in order to validate the curvature modelisation. Then, two turbine test cases are presented. These tests exhibit the major difficulties which can be encountered in reality : laminar-turbulent transition, strong deceleration due to a shock wave impingement, separation bubbles near the leading edge or strong acceleration leading to partial relaminarization. For these cases, the available experimental data are the distributions of the Mach number and of the convective heat transfer coefficient $h$. The coefficient $h$ is defined as the ratio of the heat flux divided by the difference between the freestream temperature and the wall temperature. $h$ is measured with $\Delta h = 1000 \pm 50 W / m^2 K$ uncertainty.

Convex wall flow

The computation was tested for turbulent flow on a convex wall followed by a flat plate (recovery region), to especially validate curvature modelisation. The curvature is strong $\delta / Rc = 0.10$ and extends from $s / s_{max} = 0.20$ to $s / s_{max} = 0.7$. This experiment was first investigated by Gillis et al. (1980) for the aerodynamic field.

![Figure 1: Skin friction evolution versus the non-dimensional curvilinear abscissa.](image1)

![Figure 2: Reynolds stress profiles in the curved region.](image2)
The skin friction evolution is presented in Fig. 1. The model reproduces the skin friction decrease in the curved zone. At the end of the curvature \((s / s_{\text{max}} \approx 0.7)\), the

calculated skin friction is underestimated. In the recovery region, the prediction is satisfactory. Detailed results of the turbulent boundary layer are presented now. The Reynolds stress evolutions in the direction normal to the wall are plotted for two streamwise positions in the curved zone \((s / s_{\text{max}} \approx 0.45)\) and \((s / s_{\text{max}} \approx 0.56)\) Fig. 2, and two positions in the recovery region \((s / s_{\text{max}} \approx 0.72)\) and \((s / s_{\text{max}} \approx 0.82)\) Fig. 3. In the curved zone, very satisfactory agreement may be observed. In the recovery region, the model for the outer layer does not reproduce a sufficient relaxation of the Reynolds stress. Nevertheless, the results are satisfactory.

![Figure 3: Reynolds stress profiles in the recovery region.](image)

![Figure 5: Stanton number evolution versus the non-dimensional curvilinear abscissa.](image)

Simon and Moffat (1983) extended the experimental study including thermal effects. The Stanton number distribution is presented in Fig. 4. The Stanton number decreases, due to the stabilizing effect of the convex wall and is well reproduced by the calculation. In the recovery region, the results agree with the experiments. Stagnation temperature profiles are presented for \(s / s_{\text{max}} = 0.63\) (curved region) and for \(s / s_{\text{max}} = 0.82\) (recovery region) in Fig. 5. The evolutions are well predicted by the computation.

![Figure 6: Rotor blade - Mach number evolution versus the non-dimensional curvilinear abscissa.](image)

Turbine rotor blade

The flow develops on a turbine rotor profile mounted in a transonic linear cascade configuration in the CT2 Von Karman Institute facility, (Arts, 1985). Measured Mach number distribution is plotted in Fig. 6 for the suction side and for the pressure side. The flow is transonic or supersonic on the major portion of the blade with the outlet Reynolds number equal to \(10^6\). The wall temperature reaches 296K, whereas the external flow temperature is 422K. Measured upstream freestream turbulence intensity is 0.8%.
On the suction side, the laminar-turbulent transition zone is located at the velocity peak, shown in Fig. 6, between 70% and 80% of the profile length. Measured and calculated heat transfer coefficient $h$ are plotted in Fig. 7, versus the non-dimensional curvilinear abscissa $s / s_{\text{max}}$. On the initial part of the blade, the flow is laminar and the acceleration leads to the decrease of $h$. During the laminar-turbulent transition, the heat transfer coefficient strongly increases. This effect is well reproduced by the computation. Very good agreement may be observed.

On the pressure side, the transition region is short, as shown in Fig. 6, and is located very near the leading edge, at the velocity peak ($s / s_{\text{max}} = 0.1$). One of the difficulties in this test case occurs in this region. The laminar boundary layer deceleration results in a separation zone and induces the heat transfer coefficient $s / s_{\text{max}} = 0.1$, as shown in Fig. 8. The transition is located in this separation; it leads to the viscous layer reattachment and thus to the strong increase of $h$. Although the complete rise of $h$ up to the experimental point is not reflected by the computation, the physical phenomena are well reproduced. The evolution of the calculated skin friction $C_f$, plotted in Fig. 9, shows the separation bubble ($C_f$ becomes negative). In this case, the coupling mode is essential to obtain a stable calculation.

**Turbine stator blade**

The same VKI facility was used to test a high pressure turbine nozzle guide vane (Arts and Bourguignon, 1990) and (Arts and Lapidus, 1992). The Mach number evolution is plotted in Fig. 10 for the suction side and for the pressure side. A weak shock is identified on the suction side around 50% of the total length. On the pressure side, shown in Fig. 10, a velocity peak is observed at $s / s_{\text{max}} = 0.10$. Downstream, the flow is strongly accelerated up to the trailing edge. This acceleration is expected to affect the complete development of the laminar-turbulent transition, and thereby the level of the heat transfer. The freestream temperature reaches 298K and the wall temperature is 420K. Different Reynolds numbers and freestream turbulence intensities were investigated. An intermediate measured freestream turbulence level, $T_u=4\%$, was
selected and calculations were performed for the two outlet Reynolds numbers $Re_2 = 2.25 \times 10^6$ and $3.10^6$.

**Highest Reynolds number $Re_2 = 3 \times 10^6$.** For the highest Reynolds number, the measured and calculated heat transfer coefficients are plotted in Fig. 11 for the suction side and in Fig. 12 for the pressure side. On the suction side, the acceleration is responsible for the heat transfer decrease along the laminar part of the boundary layer. The laminar-turbulent transition occurs at $s/\text{max} = 0.25$ on a short length and increases the heat transfer level. The computation correctly predicts the evolutions. For the turbulent part of the flow, an overpredicted level of $h$ is observed.

On the pressure side, transition occurs immediately downstream of the leading edge. Because of the high Reynolds number, the transition is complete. The calculated results are in good agreement with the data.

**Lowest Reynolds number $Re_2 = 2.25 \times 10^6$.** For the lowest value of the Reynolds number, $Re_2 = 2.25 \times 10^6$, the measured and calculated heat transfer coefficients are presented in Fig. 13 for the suction side and in Fig. 14 for the pressure side. The experimental transition occurs later on the suction side, at $s/\text{max} = 0.26$, as indicated by measured leveling-off of $h$. As the flow accelerates further downstream, the complete transition process is delayed up to the shock wave position. This stabilization in a transitional state is confirmed by the almost constant level of $h$ in this region. The turbulence modelling is not able to simulate this physical phenomenon. Consequently, the calculated heat transfer coefficient continues to decrease. Through the shock, the heat transfer evolution is well predicted. After the shock wave ($s/\text{max} = 0.5$), a fully turbulent boundary layer develops.

On the pressure side, the heat transfer has the same evolution as for $Re_2 = 3 \times 10^6$, except the main level of $h$ is lower. The computed results agree well with the experimental data.
For the computations, the mesh density is approximately the same, containing 50 planes in the x-direction and 40 grid points in the y-direction normal to the wall. The CPU times are less than 1 minute on a workstation Silicon Graphics Indigo2.

CONCLUSIONS

A modelisation of viscous flows on turbine blades was developed based on boundary layer equations. Curvature effects were introduced in the one-equation turbulence model. The results are satisfactory in the curved region but discrepancies may be observed in the recovery region where history effects exist.

The capability of the method to predict the heat transfer on real turbine blade configurations, is presented with physical phenomena well reproduced. Calculation stability is demonstrated, even for a separation bubble and strong decelerations due to shock wave impingement. The weakness of the turbulence model may be noted for strongly accelerated flows. Future work will focus on this subject.

ACKNOWLEDGMENTS

We would like to thank the SNECMA for the financial support of the code development and Société Européenne de Propulsion (SEP) and Centre National d’Études Spatiales (CNES) for the financial support of the curvature model study.

REFERENCES


