Reassessment of the Wall Functions Approach for RANS Computations

Eric Goncalves da Silva, Robert Houdeville

► To cite this version:

Eric Goncalves da Silva, Robert Houdeville. Reassessment of the Wall Functions Approach for RANS Computations. Aerospace Science and Technology, 2001, 5 (1), pp.1-14. 10.1016/S1270-9638(00)01083-X . hal-01493440

HAL Id: hal-01493440

<https://hal.science/hal-01493440>

Submitted on 21 Mar 2017

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.

Reassessment of the Wall Functions Approach for RANS Computations

E. Goncalves ¹ and R. Houdeville ²

* ONERA-TOULOUSE, Aerodynamics and Energetics Modelling

Department,

2, av. Ed. Belin, B.P. 4025, 31055 Toulouse Cedex 4, France

Abstract

To assert the validity of the wall law approach in a RANS code, systematic comparisons to results obtained with the same turbulence models, integrated down to the wall with damping functions, in a fine mesh are presented. It is shown that a very simple representation of the velocity profile in the wall region gives good results for transonic flows over airfoils with shock wave/boundary layer interaction leading to separation. Moreover, it is also shown that the heat flux can be correctly predicted in separated regions. The case of the infinite swept wing near separation is also considered and gives excellent results.

Four popular turbulence models, $k-\varepsilon$, $k-\omega$, $k-l$ and Spalart Allmaras, have been used for the study but the approach can be extended to other models.

Keywords

Wall functions/Navier-Stokes/Turbulence models.

¹PhD Student. Tel : 05 62 25 25 25, e-mail : Eric.Goncalves@oncert.fr

²Research Master. Tel : 05 62 25 28 05 , e-mail : Robert.Houdeville@oncert.fr

Nomenclature

Latine alphabet

x, y, z	wall reference frame
X, Y, Z	global reference frame
C_f	shear stress coefficient, $2\tau_w/\rho_e u_e^2$
C_{f0}	shear stress coefficient, $2\tau_w/\rho_0 u_0^2$
C_h	heat flux coefficient, $q_w/\rho_e u_e C_p (T_w - T_f)$
C_p	specific heat at constant pressure
F_c, F_d	convective and diffusive flux densities
K_p	pressure coefficient, $(P - P_0)/0.5\rho_0 u_0^2$
$\vec{q}$	total heat flux, $\vec{q}^v + \vec{q}^t$
k	turbulent kinetic energy
l	turbulent length scale
P	static pressure
P_k	Turbulent kinetic energy production
Pr, Pr_t	Prandtl numbers
s	Reynolds analogy factor, $2C_h/C_f$
T	mean static temperature
U_τ	friction velocity, $\sqrt{\tau_w/\rho_w}$
$\vec{V}, \vec{u}$	velocity in global and wall reference frames

Greek alphabet

β_0	deviation angle between wall friction and velocity at the boundary layer edge
ε	dissipation rate
κ	Von Kármán constant
μ, μ_t	molecular and eddy viscosity

ν	kinematic viscosity coefficient
Ψ	turbulent length scale determining variable
ρ	density
θ	momentum thickness
$\overline{\overline{\tau}}$	total stress tensor, $\overline{\overline{\tau^v}} + \overline{\overline{\tau^t}}$
ω	specific dissipation

Subscripts and Superscripts

e	boundary layer edge value
w	wall value
0	reference value
$1, 2$	adjacent and second cell with respect to the wall
$+$	wall scale
v	viscous
t	turbulent

1 INTRODUCTION

In a turbulent boundary layer, the fast evolution of the velocity in the wall region imposes to use a very fine mesh to correctly represent the flow properties, in particular the skin friction coefficient and the turbulent quantities. It is commonly admitted that the height h_1^+ of the first cell, expressed in wall units, must be close to 1 or even as small as 0.3 to reach an acceptable accuracy [1]. Such a condition may be difficult to fulfil and always leads to a very large aspect ratio of the cells near the walls which is damaging for the accuracy of the numerical schemes. Moreover, the local integration time step must be very small in order to respect the CFL condition. This may drastically slow down

the convergence process.

During the first attempts to numerically solve the Reynolds averaged Navier-Stokes equations [RANS], the computer capacities were not sufficient to use such fine meshes near the walls. The use of a wall law approach to get through the wall region provided an acceptable temporary solution. This explains the extensive use of wall laws twenty years ago. However, two objections are commonly done against the wall law approach. The first one is that from a theoretical point of view, the wall law validity is less than certain in regions where a severe adverse pressure gradient exists, leading to separation. This may be a strong argument against the use of wall laws. On the other hand, the continuous development of turbulence models with wall damping functions in the transport equations and the spectacular progress of the computer capacities have allowed the integration of the equations down to the wall, with finer and finer grids in complex configurations including massive separated regions. In these conditions, the use of wall laws was less and less justified. In his 1984 paper entitled "Time to abandon wall functions?", Launder [2] clearly displays a widespread opinion among the turbulence modelling community [3, 4, 5]. The second objection deals with the mesh convergence and the sensitivity to the location of the first grid node, supposed to be in the log-region. We will see that this assumption is not necessary.

Today, despite the available capacities of computers, the full Navier-Stokes computation, down to the wall, of a realistic aircraft configuration remains difficult and expensive. Moreover, the turbulence models, always established for large values of the turbulent Reynolds number, display difficulties to correctly describe the wall region, partly due to the weakness of the wall damping functions. In parallel to

these difficulties, the improvements achieved in the use of wall laws in Navier-Stokes codes lead to ask again the question of their opportunity.

Among these improvements in the wall laws formulation, one can cite Viegas and Rubesin [6, 7] who took into account the influence of the viscous sublayer on the logarithmic region and proposed a treatment for various turbulence models with transport equations. Smith [8, 9], for his $k - l$ model, performed a numerical integration of the momentum and energy equations from the wall to the first grid point, by only neglecting the advection terms. By doing so, he overcame the problem of the validity of the logarithmic law. Pironneau and Mohammadi [10, 11] used a wall condition only on the numerical fluxes, not exactly at the wall but in its vicinity, in the log-region. The advantage is to fully separate the application of the boundary conditions from the computation of the inner domain. This has been recently proposed again by Menter and Grotjans [12] who also used a numerical flux formulation for the turbulent quantities. Toussaint [13] used the numerical flux approach but kept the no-slip condition at the wall to perform turbomachinery applications with a mixing length model. Thangam et al. [14, 15] computed the backward step with the wall laws approach to test various $k - \varepsilon$ models at high turbulent Reynolds numbers. A wall law formulation with a varying sublayer thickness has been proposed by Ciafalo and Collins [16] in order to improve the heat transfer prediction in separated region.

In the following, only a simple wall law approach based on the no-slip condition, as proposed by Toussaint [13] is considered. The treatment of various turbulence models using one or two transport equations is given. The validation is first made for the flat plate case with emphasis on the

influence of the mesh and of the artificial dissipation treatment. The second validation case deals with transonic flows on airfoil with separation in adiabatic or isothermal conditions. Systematic comparisons to results obtained with the same code but integrating the RANS equations down to the wall are given. For these computations, a fine mesh is used.

2 EQUATIONS OF THE PROBLEM

The RANS equations coupled with a two-equation model can be expressed as :

$$\frac{\partial f}{\partial t} + \text{div}(F_c - F_d) = S \quad (1)$$

$$F_c = \begin{pmatrix} \rho \vec{V} \\ \rho \vec{V} \otimes \vec{V} + p \vec{I} \\ \rho E \vec{V} + p \vec{V} \\ \rho k \vec{V} \\ \rho \Psi \vec{V} \end{pmatrix} \quad (2)$$

$$F_d = \begin{pmatrix} 0 \\ \overline{\overline{\tau^v}} + \overline{\overline{\tau^t}} \\ (\overline{\overline{\tau^v}} + \overline{\overline{\tau^t}}) \cdot \vec{V} - \vec{q^v} - \vec{q^t} \\ (\mu + \mu_t / \sigma_k) \vec{\text{grad}} k \\ (\mu + \mu_t / \sigma_\Psi) \vec{\text{grad}} \Psi \end{pmatrix} \quad (3)$$

where $f = (\rho, \rho \vec{V}, \rho E, \rho k, \rho \Psi)^t$ denotes the conservative variables, F_c and F_d the convective and diffusive flux densities and S the source terms which concern only the transport equations. k is the turbulent kinetic energy and Ψ the length scale determining variable.

The exact expression of the eddy viscosity μ_t and the source terms depends on the turbulence model, as well as the σ_k and σ_Ψ constants.

The total stress tensor $\overline{\overline{\tau}}$ is evaluated following the Stokes hypothesis and the Boussinesq assumption. The total heat flux vector $\vec{q}$ is obtained from the Fourier law with the constant Prandtl numbers hypothesis.

$$\overline{\overline{\tau}} = \overline{\overline{\tau^v}} + \overline{\overline{\tau^t}} = (\mu + \mu_t) \left[\frac{1}{2} (\text{grad } \vec{V} + (\text{grad } \vec{V})^t) - \frac{2}{3} (\text{div } \vec{V}) \vec{I} \right] + \frac{2}{3} k \vec{I} \quad (4)$$

$$\vec{q} = \vec{q^v} + \vec{q^t} = - \left(\frac{\mu}{P_r} + \frac{\mu_t}{P_{rt}} \right) C_p \text{grad } T \quad (5)$$

2.1 Turbulence Models

The basic turbulence models used in the present study are the following :

- $k - l$ model

In this model proposed by Smith [8, 9], the second variable is a characteristic length scale which is proportional to the wall distance in the wall region.

- $k - \varepsilon$ model

The low Reynolds version was proposed by Jones and Launder [17] and the numerical values of the constants from this reference are used.

- $k - \omega$ model

This model, proposed by Wilcox [18], does not use wall damping functions. At a wall, the imposed boundary condition for ω is :

$$\lim_{y \rightarrow 0} \omega = \frac{6\nu}{\beta y^2}$$

- Spalart Allmaras model

This model uses a unique transport equation for the eddy viscosity $\tilde{\nu}$ [19, 20].

These models, integrated down to the wall, are used as reference computation results to assert the validity of the wall law approach.

2.2 Numerical resolution

The code used for the study, called CANARI [21, 22, 23], is based on a cell-centered finite-volume discretization in structured mesh. From the Ostrogradski formula, the integration of the RANS system (1) on a cell of volume Ω limited by a surface Σ and with an outer normal $\vec{n}$ leads to :

$$\frac{d}{dt} \int_{\Omega} f d\Omega + \oint_{\Sigma} F_c \cdot \vec{n} d\Sigma - \oint_{\Sigma} F_d \cdot \vec{n} d\Sigma = \int_{\Omega} S d\Omega \quad (6)$$

The time integration is done by a four steps Runge-Kutta method, with a local time step and an implicit residual smoothing. For the space discretization, the Jameson centered scheme is used. It is stabilized by a blend of 2^{nd} and 4^{th} differences artificial dissipation with the Martinelli correction. The corresponding χ_2 and χ_4 coefficients are taken equal to 1 and 0.016. To avoid unrealistically large values of the artificial dissipation in the vicinity of walls, with a coarse mesh, the conservative variables are extrapolated at the wall rather than using the physical values of the no-slip condition.

2.3 Wall boundary condition

Only the wall boundary condition will be discussed here in details to clearly show the key points of the wall law approach and the underlying assumptions. With the cell-centered formulation, the mean values of the conservative variables over a cell are stored at the barycenter G (figure 1). The application of the wall boundary condition into the numerical scheme consists in taking into account the no-slip condition in the balance of the numerical fluxes in equation (6). To this end, the conservative quantities

are assumed constant over each cell interface and equal to the mean value of the adjacent cells. Let us now consider each equation of the problem.

Continuity equation : As $\vec{V}_W = \vec{0}$, no particular treatment is needed, whatever is the size of the cell.

Momentum equations : Because $\vec{V}_W = \vec{0}$, the computation of the convective flux densities reduces to estimate the pressure at the wall. Even for large values of the height of the cell, the assumption $P_W = P_G$ remains valid due to the small variation of the static pressure in a boundary layer.

The computation of the diffusive flux density corresponds to the integration of $\overline{\overline{\tau}} \cdot \vec{n} d\Sigma$ over the ABCD contour. To this end, $\overline{\overline{\tau}}$ is firstly assumed constant over each interface. A second assumption is needed because $\overline{\overline{\tau}}$ is known at the cell centers. For an inner interface, such as AB or BC, one simply has $\overline{\overline{\tau}}_{AB} = (\overline{\overline{\tau}}_{GL} + \overline{\overline{\tau}}_G)/2$. At the wall, an extrapolation is needed. By neglecting the advection terms in the x-momentum boundary layer equation, one gets :

$$\frac{\tau_{xy}}{\tau_w} = 1 + p^+ y^+ \quad ; \quad p^+ = \frac{\nu_w}{\rho_w U_\tau^3} \frac{\partial p}{\partial x} \quad (7)$$

This shows that the shear stress variation near a wall remains small even for p^+ values of order 10^{-2} . As a consequence, remembering that only the τ_{xy} component has a significant contribution in the diffusive fluxes balance, $\overline{\overline{\tau}}_W$ can be assumed equal to $\overline{\overline{\tau}}_G$ even for coarse meshes.

The actual problem comes from the computation of $\overline{\overline{\tau}}$ in each cell which implies the knowledge of the velocity gradient. To correctly evaluate this gradient, a severe mesh refinement is needed, leading to a mesh size less than 5, in wall units. The wall law approach precisely consists in estimating the stress tensor without referring to the velocity derivative.

Energy equation : The estimation of the convective flux balance is straightforward because $\vec{V}_W = \vec{0}$. For the diffusive flux density, the heat flux q_w at the wall must be known, which is the case for an adiabatic wall. For an isothermal wall, it is given by the Fourier law through the temperature gradient. Once again, this gradient computation requires a very fine mesh near the wall. To overcome this limitation, the heat flux at the wall must be evaluated without using the temperature derivative. Details of the methods are given in the next section.

Transport equations : Once again, the convective flux balance does not present any problem. The diffusive flux density calculation requires to evaluate the gradient of the transported turbulent quantities over each interface. Moreover, the computation of the source terms implies damping functions which vary rapidly near the wall. All of this imposes a strong mesh refinement [1]. To develop a wall law approach, various ways may be followed, all relying on physical descriptions of the turbulence development near a wall.

A first solution consists in integrating k down to the wall with $k_W = 0$. In a coarse mesh, a particular treatment is needed to correctly estimate the production term. The wall law provides $\overline{\tau_G}$ and the velocity gradient. The second turbulent quantity Ψ can be deduced from the value of k computed from the RANS integration and from a characteristic turbulence length scale. In this case, the wall condition for Ψ is no longer needed.

In the second solution, k is directly imposed at the center of the first cell. To this end, the Bradshaw relation is used and $-\rho\overline{u'v'}$ is assumed to be equal to τ_w by neglecting p^+ in (7). Rubesin and Viegas [6, 7] proposed a more realistic model of the evolution of the turbulent quantities in the first cell and Smith [8] gave an analytical solution to the k-equation based

on non-restrictive hypothesis.

In practice, it seems very difficult to justify one or the other of these solutions. Only comparisons with results obtained with fine meshes can assert the validity of the approach.

3 WALL LAW FORMULATION

3.1 Mean flow equations

For a two-dimensional incompressible turbulent boundary layer not too close to separation, the velocity profile can be simply represented as :

$$\begin{aligned} u^+ &= y^+ & \text{if } y^+ < 11.13 \\ u^+ &= \frac{1}{\kappa} \ln y^+ + 5.25 & \text{if } y^+ > 11.13 \end{aligned} \quad (8)$$

To extend the wall law to compressible boundary layers, van Driest [24, 25] introduced the velocity transformation :

$$\bar{u} = \int_0^u \sqrt{\frac{\rho}{\rho_w}} du \quad (9)$$

To integrate (9), the temperature evolution in the wall region must be known. The starting equation :

$$u\tau_{xy} - q_y = -q_w \quad (10)$$

is obtained by neglecting the advection terms in the total enthalpy equation. By expressing τ_{xy} and q_y , equation (10) leads to a simple differential equation in u which gives :

$$\begin{aligned} T &= T_w - A \frac{u^2}{2} \left(1 + \frac{2q_w}{\tau_w u} \right) \\ A &= \frac{\mu + \mu_t}{Cp \left(\frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right)} \end{aligned} \quad (11)$$

where A is assumed constant, which is exact only if the laminar and turbulent Prandtl numbers are equal. From $\rho T = \rho_w T_w$, the integration of (9) gives :

$$\begin{aligned}\bar{u} &= \frac{1}{\sqrt{B}} \left[\arcsin \left(\frac{2Bu + C}{\sqrt{4B + C^2}} \right) - \arcsin \left(\frac{C}{\sqrt{4B + C^2}} \right) \right] \\ B &= \frac{A}{2T_w} \quad ; \quad C = \frac{1}{T_w u} \left(T_w - T - A \frac{u^2}{2} \right)\end{aligned} \quad (12)$$

Knowing T and u in the first cell, relation (11) gives the heat flux at the wall for an isothermal wall or the wall temperature for an adiabatic wall.

The wall law treatment is now straightforward. From the thermal boundary conditions and the conservative quantities in the adjacent cell to the wall given by the Navier-Stokes solution, equation (12) supplies $\bar{u}$ at the center of the cell which is used in (8) to compute τ_w .

Extension to separated flow and three-dimensional flows may be simply done by taking u in relation (8) in the wall friction direction.

At first look, the application domain of the wall law seems very narrow: boundary layer with mild pressure gradient. This is true if equation (8) must be applied from the wall up to $y/\delta \simeq 0.15$. In the wall law approach, equation (8) is only needed to be valid in the first cell. Let us now consider the vicinity of separation. τ_w is very small and therefore y^+ in the first cell, providing its size is not too large. This means that the log law is no longer imposed but only the linear law. In a strong flow reversal domain, τ_w may become important and the log law is applied again without theoretical justification. Once again, only comparisons to fine mesh results can assert the validity of the approach.

3.2 Turbulent quantities

3.2.1 k equation

It has been seen that two different ways could be followed to introduce a wall law in the k -equation : either to impose the production of k or k itself in the adjacent cells to walls. These two approaches have been tested and they gave very similar results. Then, only the first one will be considered here.

It must be noted that the k -equation is treated the same way for all turbulence models. k is set equal to zero at the wall and its production is computed in the first cell as proposed by Viegas and Rubesin [6, 7] who explicitly take into account the viscous sublayer defined as $y_v^+ = 5$. With the notation given in figure 2, the production of k is given by :

$$(P_k)_1 = \frac{1}{y_e} \int_0^{y_e} \tau_{xy}^r \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) dy = \frac{\tau_w^{3/2}}{\kappa y_e \sqrt{\rho_w}} \ln \frac{y_e}{y_v} + \tau_w \frac{\partial v}{\partial x} \left(1 - \frac{y_v}{y_e} \right) \quad (13)$$

3.2.2 Second transport equation

$k - l$ model [Sm]

Near a wall, the second variable reduces to a classical mixing length. In the first cell, this leads to :

$$l_1 = \kappa y_1 \quad ; \quad \kappa = 0.41 \quad (14)$$

Except for the first cell, the model is unchanged.

$k - \varepsilon$ model [JL]

With the wall law approach, the high Reynolds version of the Jones Launder model is used. However, to express the dissipation in the first cell as a function of k , the Chen formulation is retained [5] :

$$\varepsilon_1 = \frac{k_1^{3/2}}{l_\varepsilon} \quad (15)$$

$$l_\varepsilon = \frac{\kappa y_1}{C_\mu^{3/4}} \left[1 - \exp\left(\frac{-Re_t C_\mu^{3/4}}{2\kappa}\right) \right] \quad ; \quad Re_t = \frac{\sqrt{k_1} y_1}{\nu_1} \quad (16)$$

$k - \omega$ model [W1]

In the first cell, ω is expressed as a function of k :

$$\omega_1 = \frac{\varepsilon_1}{C_\mu k_1} = \frac{\sqrt{k_1}}{C_\mu l_\varepsilon} \quad (17)$$

with l_ε given by (16).

3.2.3 Spalart Allmaras model [SA]

From the closure relations of the model :

$$\mu_t = \rho \tilde{\nu} f_{v1} \quad , \quad f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3} \quad , \quad \chi = \frac{\rho \tilde{\nu}}{\mu} \quad (18)$$

$\tilde{\nu}$ can be obtained at the center of the first cell if μ_t is known. To this end, a mixing length formulation is used with the velocity derivative deduced from the wall law :

$$\mu_t = \rho \kappa^2 y^2 (1 - e^{-y^+/26})^2 \frac{U_\tau}{\kappa y} \quad (19)$$

This leads to a fourth order equation with a unique real and positive solution :

$$\begin{aligned} (\rho \tilde{\nu})_1 &= \frac{1}{4} \left[\mu_{t1} + C_2 + (3\mu_{t1}^2 - C_2^2 + \frac{2\mu_{t1}^3}{C_2})^{1/2} \right] \\ C_0 &= \mu_1 \mu_{t1} C_{v1} \quad ; \quad C_1 = C_0^3 \sqrt{3} \left(\frac{256 C_0^3}{\mu_{t1}^6} + 27 \right)^{1/2} \\ C_\pm &= C_1/18 \pm 0.5 C_0^2 \quad ; \quad C_2 = \left[\mu_{t1}^2 + 4(C_-^{1/3} - C_+^{1/3}) \right]^{1/2} \end{aligned} \quad (20)$$

Except for the first cell, the model is unchanged.

3.3 Laminar region

In a laminar region, μ_t is set to a small value, typically $10^{-3}\mu$, and the wall law reduces to $u^+ = y^+$, whatever the y^+ value. This is equivalent to compute the velocity gradient over two points in adjacent cells to a wall. No special treatment is done in the intermittency region.

4 APPLICATION RESULTS

4.1 Flat plate case

At first look, this case does not seem interesting because the validity of the wall law is clearly established for the flat plate. However, it allows a careful examination of the numerical problems and an easy study of the mesh convergence. Only the key points and the main conclusions will be given here.

The flow conditions are : $M_0 = 0.7$, $Re_L = 4.3 \cdot 10^6$, $T_i = 300K$ and the boundary layer is turbulent from the leading edge. The inlet and outlet section of the computational domain correspond to 0.125 and 0.5 of the length L of the plate, respectively. There are 45 nodes in the flow direction with a finer distribution near the leading edge. Four meshes are considered. They just differ by the number and the distribution of nodes in the normal direction. Figure 3 gives an idea of the mesh lines distribution near the wall :

- fine mesh : the geometric progression of the 81 nodes is equal to 1.129. The height of the first cell varies between $h_1^+ = 0.2$ near the leading edge to 1.2 at the trailing edge.
- mesh 1 : there are only 51 nodes in the normal direction following a bi-exponential distribution with reasons equal to 1.1 and 0.91. h_1^+ varies between 30 and 66 and the ratio of the height of the two first cells h_2/h_1 is equal to 1.1.
- mesh 2 : it is identical to mesh 1 except that the second line has been removed. h_1^+ varies between 50 and 135 and $h_2/h_1 = 0.576$.
- mesh 3 : The 51 nodes are also distributed over a bi-exponential progression with reasons equal to 1.09 and 1.004 but h_2/h_1 is equal to 0.233.

With the fine mesh, the transport equations of paragraph 2.1 are integrated down to the wall. These corresponding computations are used as reference results for the comparisons. The wall law approach is used only with the coarse meshes.

Results

In order to evaluate the influence of the wall law approach on the results, only the two most sensitive parameters to this treatment are discussed in details here. First of all, the skin friction coefficient $C_{f_0} = 2\tau_w/\rho_0 U_0^2$ is plotted in figure 4 for the various turbulence models. The results from a boundary layer computation with the Chien model [26] are also given. They are representative of commonly accepted results for a flat plate. The C_f values obtained with mesh 1 (regular geometric progression) are never correct. This may be due to the regular distribution of the size of the meshes near the wall ($h_2/h_1 = 1.1$) which imposes a too large size of the second mesh to correctly obtain the various gradients of the quantities needed to the models. For all models except the Spalart Allmaras, the skin friction coefficient computed with mesh 3 is very close to the results from the fine mesh. For the $k - \varepsilon$ model, the result is closer to the Chien boundary layer computation than the value obtained with the fine mesh.

The profiles of the τ_{xy} component of the stress tensor are presented in figure 5. The most striking result is the large oscillation of the shear stress in the second cell with the wall law approach. This cell corresponds to the first one for which τ_{xy} is computed from the velocity derivative. Near the wall, τ_{xy} is given by the wall law. The oscillation decreases slightly from mesh 1 to mesh 3 but never disappears. In fact, it is due to the 4th differences term of the artificial dissipation

used to insure the stability of the scheme. By removing this artificial dissipation in the first cell near the wall, the oscillation is displaced towards the third cell. We have seen that it disappears completely using the wall treatment proposed by Mohammadi [10, 11] which imposes a slip condition away from the wall but other problems occur in separated regions with this approach. In fact, the wall condition treatment for the artificial dissipation needed for centered schemes is important with the wall law approach. To compute this artificial dissipation, either the physical wall conditions or an extrapolation can be employed. The first solution leads to unacceptable results. This has also been noted by Zhu [27]. The second solution, with a 0-order extrapolation, is used for the present study.

To sum up, the flat plate case shows the great importance of the numerical aspects with respect to the physical one in the wall law approach. It also leads to a simple rule for the mesh construction to obtain the same precision as with a fine mesh : the second mesh size must be small enough to correctly represent the velocity gradient. The simplest way to satisfy this requirement is to build a fine mesh, with y^+ of order unity as for a classical computation down to the wall and to remove a certain number of cell layers near the wall. This number depends on the distribution of nodes in the boundary layer. To show this, more or less nodes have been removed from the fine mesh in figure 3 and the influence on the skin friction at $x/c = 0.97$ is considered. Only the results obtained with the $k - l$ model are given in table 1. This table shows that for a y_1^+ in the range 20 to 200, the skin friction variation remains less than 1%. The error reaches 3.2% for a y_1^+ equal to 360. The worst result is obtained for a y_1^+ in the range 4 to 12 which correspond to the buffer layer not correctly modeled by equations 8. In the table, the

CPU speed up, ratio between CPU time for a given case and the fine mesh computation, is also given. For a reasonable size of the first cells, it is of order 4 to 8. The table also shows that the wall law can also be used with a fine mesh without a lost of accuracy.

4.2 OALT25 airfoil

The computation conditions retained for the validation of the wall law approach correspond to a severe test case close to the onset of the buffeting regime : $Re_c = 20.15 \cdot 10^6$, $\alpha = 2.2^\circ$, $M_\infty = 0.77$, $T_i = 151K$, $P_i = 2.410^5 Pa$. Transition is imposed at $x/c = 0.05$ on both sides of the airfoil. Comparisons with experimental results obtained in the cryogenic transonic wind tunnel T2 at O.N.E.R.A. [28] will be presented.

Meshes

The three C-type meshes used for the study are presented in figure 6. All of them extend to 10 chords with a vorticity boundary condition in order to simulate a uniform infinite flow [29]. There are 341 nodes in the flow direction (240 on the airfoil and 40 in the wake) with a finer distribution around the shock. The meshes only differ by the number and the size of cells in the normal direction :

- the fine mesh has 97 nodes in the crossflow direction and the size h_1^+ of the first cell varies between 0.4 and 0.7.
- the mesh 1 is obtained by removing 20 lines from the fine mesh near the wall. h_1^+ varies from 4 in the separated region to 70. The ratio h_2/h_1 ranges from 0.8 to 0.9.
- the mesh 2 has 81 nodes in the crossflow direction with $h_2/h_1 \simeq 0.35$. The size of the first cell near the wall is identical to that of mesh 2.

Pressure coefficient

A very small influence of the meshes with the wall law approach can be seen from figure 7. Compared to the results obtained with the fine mesh, only the $k - \omega$ model shows a discrepancy in the shock location. In fact, the wall law approach removes one of the difficulties with this model, i.e. the sensitivity of the $k - \omega$ model to the artificial dissipation near the wall.

Skin friction coefficient

The skin friction coefficient $C_{f_0} = 2\tau_w / \rho_0 U_0^2$ along the suction side of the airfoil is given in figure 7. At first look, there is a large discrepancy upstream of the shock, in the accelerated region where the boundary layer is thin. Two different points must be considered. Firstly, the comparison of the wall law approach with the fine mesh computation, for each model, clearly shows the influence of the mesh, weak for [SA] and important for [Wl]. Secondly, by comparing all the models to each other, the discrepancy is the most important for the fine mesh results and is reduced with the wall law approach with the mesh 2. This result clearly shows the interest to use wall laws.

In the rear part of the airfoil, downstream of the shock, all models predict the separation of the boundary layer, both with the fine mesh or using a wall law. The most interesting result is related to the Jones Launder model. With the fine mesh, the model predicts a reattachment of the boundary layer with a relatively large value of the skin friction contrarily to the other models. This behaviour is removed with the wall law approach. This has also been previously observed by Viegas and Rubesin [6]. To conclude this point, it can be noted that the convergence with the wall law approach remains good despite the importance of the separated

region, which is not the case with the fine mesh. To reach the same level of convergence between the two computations, 5 times less iterations are necessary with the wall law treatment.

Velocity profiles

The velocity profiles U/U_0 are given in figure 8 in semi-logarithmic coordinates at $x/c=0.84$ located in the separated region, downstream of the shock. The profiles obtained with the wall law approach are remarkably close to those obtained with the fine grid computation. This is true with the two coarse grids, except for the $k-\omega$ model. The [SA] model predicts the most important backflow velocity, followed by [Sm] and [Wl]. The [JL] model does not predict recirculation at this location but a null value of the skin friction coefficient.

These results clearly show that the hypothesis concerning the validity of the wall law in separated regions is not so critical.

4.3 OALT25 airfoil with an isothermal wall

Heat flux prediction with a wall law approach is a more difficult problem than the adiabatic cases because of the fast evolution of the temperature near the wall and the relative weakness of equation (11) because of the assumption that the A coefficient is constant. This has led Smith [8] to develop a method in which the momentum and energy equations are numerically integrated in the wall region, only neglecting the advection terms. To see if such a numerical effort is really needed, the OALT25 airfoil is again considered in the same conditions, but the wall temperature is imposed equal to $1.4T_i$. Only a calculation with the $k-l$ model is presented here.

Figure 9 shows that the pressure coefficient is correctly computed with

the two coarse meshes. Concerning the skin friction coefficient, plotted in the same figure, a very good agreement is obtained between the results of mesh 2 and the fine mesh computation. This confirms the validity of the van Driest transformation.

More interesting is the heat flux evolution presented in figure 9. The best results are obtained with mesh 2. In the separated region, downstream of the shock, the heat flux level is obtained with in 15% error. Such a result was not obvious due to the form of equation (11) which expresses q_w as a function of τ_w/u . This ratio is of the form 0/0 at separation. However, it is perfectly defined when expressed in term of wall distance. The Reynolds analogy factor $s = 2C_h/C_f$ is interesting to consider. For mild pressure gradients, it is roughly equal to 1.2. This is observed upstream the shock, in figure 9. In separated regions, s can take any value because the heat flux does not vanish as the wall friction.

As the heat flux prediction is more sensitive to the mesh point distribution near the wall, other calculations have been done with various sizes of the first cells. Good results are obtained providing that h_1^+ is less than 20 in separated regions. This means that the meshes must be more refined near the wall than for adiabatic computations. However this condition is not too difficult to fulfil because U_τ remains small in separated regions.

4.4 Infinite swept wing

The wall law formulation presented in section 3 is assumed valid for three-dimensional boundary layers providing the velocity is expressed in the wall friction reference frame. This implies that the flow direction is assumed constant in the first cell, which is clearly not correct. Once again, comparisons with fine mesh results are needed to quantify the effect of such an hypothesis. To do that, the OALT25 airfoil with a yaw

angle equal to 30° is considered with identical normal and total conditions as in section 4. This leads to $\alpha = 1.9^\circ$, $M_\infty = 0.89$ and $Re_c = 25.08 \cdot 10^6$. The meshes are unchanged. Only a calculation with the $k-l$ model is presented.

Results

Figure 10 shows that the normal pressure coefficient $K_{pn} = K_p / \cos^2 \beta$ is perfectly computed with the wall law approach. Moreover, we note that the three-dimensional effects are weak on the pressure field because K_{pn} is almost identical to the experimental two-dimensional K_p . On the contrary, the three-dimensional effects are clearly seen in the deviation angle β_0 which reaches 80° in the separated region.

The skin friction coefficient components along the perpendicular and parallel directions to the leading edge (C_{f_x} and C_{f_z}) are presented in figure 11. The results with the wall law approach are close to those obtained with the fine mesh. Compared with the two-dimensional case, the recirculating flow has disappeared. In the accelerated flow region, mesh 2 gives slightly better results than mesh 1, as previously observed.

Finally, figure 12 shows the two components U and W of the velocity profiles at $x/c=0.84$, downstream the shock location. The wall law approach gives exactly the same results as those obtained with the fine mesh. This confirms the weak influence of the assumptions made on the velocity profile.

5 CONCLUSION

Although the use of wall laws in Navier-Stokes codes is not new, the validity of such an approach for configurations in which it is clearly established that the classical logarithmic velocity profile does not exist,

is still worthy to be confirmed. This is all the more important that turbulence models with transport equations are used. In order to draw conclusions as clearly as possible, the results obtained with the wall law approach have been compared with those obtained from computations with fine meshes for which the turbulence models, including wall damping functions, are integrated down to the wall.

For the present study, the wall law itself has been chosen as simple as possible, as well as the treatment of the transport equations. The flat plate case has shown the importance of the numerics with respect to the physics. It has also shown that some care has to be taken for the construction of meshes in order to reach high quality results. With these constraints, the results are not sensitive to the size of the first cell and the mesh convergence can be asserted.

The main configuration retained for the validation includes various difficulties. It corresponds to a transonic airfoil with a shock wave/boundary layer interaction leading to separation. Computations have also been done with heat flux at the wall and with sweep angle. It has been clearly established that the wall law approach gives excellent results and that the underlying hypothesis are less strong than they seem to be. An interesting result is that the use of a wall law does not change the behaviour of the turbulence models with respect to each other and does not deteriorate them. On the contrary, it allows to correct some of their weaknesses coming from the wall damping functions.

To conclude, the use of a wall law approach in a RANS solver proved to be very attractive, not only for the cost saving by a factor between 5 and 8, and for the evident robustness improvement, but also for the quality of the results.

References

- [1] J.E. Bardina, P.G. Huang, and T.J. Coakley. Turbulence modeling validation, testing and development. Technical report, NASA, April 1997. TM 110446.
- [2] B.E. Launder. Numerical computation of convective heat transfer in complex turbulent flows: time to abandon wall functions? *Int. J. Heat Mass Transfer*, 27(9):1485–1491, 1984.
- [3] V.C. Patel, W. Rodi, and G. Scheuerer. Turbulence models for near-wall and low Reynolds number flows: A review. *AIAA Journal*, 23(9):1308–1319, 1985.
- [4] V.C. Patel. Perspective: Flow at high Reynolds number and over rough surfaces - Achilles heel of CFD. *Journal of Fluids Engineering*, 120:434–444, september 1998.
- [5] H.C. Chen and V.C. Patel. Near-wall turbulence models for complex flows including separation. *AIAA Journal*, 26(6):641–648, june 1985.
- [6] J.R. Viegas and M.W. Rubesin. Wall-function boundary conditions in the solution of the Navier–Stokes equations for complex compressible flows. In *AIAA 83-1694, 16th Fluid and Plasma Dynamics Conference – Danver, Massachusetts*, July 12–14 1983.
- [7] J.R. Viegas, M.W. Rubesin, and C.C. Horstman. On the use of wall functions as boundary conditions for two-dimensionnal separated compressible flows. In *AIAA 85-0180, 23rd Aerospace Sciences Meeting – Reno, Nevada*, January 14-17 1985.
- [8] B.R. Smith. The $k - kl$ turbulence model and wall layer model for compressible flows. In *AIAA 90-1483, 21st Fluid and Plasma Dynamics Conference – Seattle, Washington*, July 18–20 1990.

- [9] B.R. Smith. A near wall model for the $k - l$ two equation turbulence model. In *AIAA 94-2386, 25th Fluid Dynamics Conference – Colorado Springs, Colorado*, June 20-23 1994.
- [10] B. Mohammadi and G. Medic. A critical evaluation of the classical $k - \varepsilon$ model and wall-laws for unsteady flows over bluff bodies. Technical report, INRIA, Décembre 1996. N° 3056, Thème 4.
- [11] B. Mohammadi and O. Pironneau. Unsteady separated turbulent flows computation with wall-laws and $k - \varepsilon$ model. *Comput. Methods Appl. Mech. Engrg.*, 148:393–405, 1997.
- [12] H. Grotjans and F. Menter. Wall functions for general application CFD codes. In *ECCOMAS, published by John Wiley & sons, Ltd*, 1998.
- [13] C. Toussaint and E. Laroche. Introduction de lois de paroi dans le code CANARI Cell-Vertex avec modèle de Michel. Technical report, ONERA, Décembre 1998. RT 39/7103 DAAP/Y.
- [14] S. Thangam and C.G. Speziale. Turbulent flow past a backward-facing step: A critical evaluation of two-equation models. *AIAA Journal*, 30(5):1314–1320, may 1992.
- [15] V. Yakhot, S.A. Orszag, S. Thangam, T.B. Gatski, and C.G. Speziale. Development of turbulence models for shear flows by a double expansion technique. *Physic of Fluids*, 4(7):1510–1520, 1992.
- [16] M. Ciafalo and M.W. Collins. $k - \varepsilon$ predictions of heat transfer in turbulent recirculating flows using an improved wall treatment. *Numerical Heat Transfer*, 15:21–47, 1989.

- [17] W.P. Jones and B.E. Launder. The prediction of laminarization with a two-equation model of turbulence. *Int. J. Heat Mass Transfer*, 15:301–314, 1972.
- [18] D.C. Wilcox. Reassessment of the scale-determining equation for advanced turbulence models. *AIAA Journal*, 26(11):1299–1310, November 1988.
- [19] P.R. Spalart and S.R. Allmaras. A one-equation turbulence model for aerodynamic flows. In *AIAA 92-0439, 30th Aerospace Sciences Meeting – Reno, Nevada*, January 6-9 1992.
- [20] P.R. Spalart and S.R. Allmaras. A one-equation turbulence model for aerodynamic flows. *La Recherche Aéronautique*, (1):5–21, 1994.
- [21] V. Couaillier. Numerical simulation of separated turbulent flows based on the solution of RANS/low Reynolds two-equation model. In *AIAA 99-0154, 37th Aerospace Sciences Meeting & Exhibit – Reno, Nevada*, January 11-14 1999.
- [22] N. Liapis and V. Couaillier. Unsteady Euler and Navier-Stokes flow simulations with an implicit Runge-Kutta method. In *2nd European Comput. Fluid Dynamics Conference, Stuttgart, published by John Wiley & sons*, 1994.
- [23] A.M. Vuillot, V. Couaillier, and N. Liapis. 3D turbomachinery Euler and Navier-Stokes calculations with multidomain cell-centered approach. In *AIAA Paper 93-2576*, 1993.
- [24] E.R. van Driest. Turbulent boundary layer in compressible fluids. *J. Aeronaut. Sci.*, 18:145–160, 1951.
- [25] E.R. van Driest. On turbulent flow near a wall. *J. Aeronaut. Sci.*, 23:1007–1011, 1957.

- [26] K. Y. Chien. Predictions of channel and boundary-layer flows with a low-Reynolds-number turbulence model. *AIAA*, 20(1):33–38, 1982.
- [27] J. Zhu and T.H. Shih. An NPARC turbulence module with wall functions. In *AIAA Paper 96-0382*, 1997.
- [28] D.Caruana, C.Bulgubure, and A.Mignosi. Experimental study on transonic shock wave/turbulent boundary layer interactions and separation instabilities. Suction and Reynolds number effects. In *20th Congress of the International Council of Aeronautical Sciences. Sorrento, Napoli, Italy.*, September 8-13 1996.
- [29] J.L. Thomas and M.D. Salas. Far-field boundary conditions for transonic lifting solutions to the Euler equations. *AIAA Journal*, 24(7):1074–1080, july 1986.

List of Figures

1	Adjacent cell to a wall	28
2	Viegas and Rubesin model	29
3	Wall region enlargement of the meshes for the flat plate case	30
4	Flat plate case - skin friction coefficient - $R_\theta = 6230$. . .	31
5	Flat plate case - shear stress profiles - $R_\theta = 6230$	32
6	OALT25 airfoil - fine mesh and enlargement of the trailing edge region	33
7	OALT25 airfoil - pressure and skin friction coefficients . .	34
8	OALT25 airfoil - velocity profiles at $x/c = 0.84$	35
9	OALT25 airfoil with isothermal wall condition - pressure coefficient, skin friction coefficient, heat flux and Reynolds analogy factor	36
10	OALT25 airfoil with yaw angle - Kp_n and β_0 , angle between wall friction and velocity at the boundary layer edge	37
11	OALT25 airfoil with yaw angle - skin friction components in directions perpendicular and parallel to the leading edge	38
12	OALT25 airfoil with yaw angle - Velocity profiles in par- allel and normal directions to the leading edge	39

Figure 1 – *Adjacent cell to a wall*

Figure 2 – *Viegas and Rubesin model*

Figure 3 – *Wall region enlargement of the meshes for the flat plate case*

Figure 4 – Flat plate case - skin friction coefficient - $R_\theta = 6230$

Figure 5 – Flat plate case - shear stress profiles - $R_\theta = 6230$

Figure 6 – *OALT25* airfoil - *fine mesh* and *enlargement of the trailing edge region*

Figure 7 – *OALT25* airfoil - pressure and skin friction coefficients

Figure 8 – *OALT25* airfoil - velocity profiles at $x/c = 0.84$

Figure 9 – OALT25 airfoil with isothermal wall condition - pressure coefficient, skin friction coefficient, heat flux and Reynolds analogy factor

Figure 10 – *OALT25* airfoil with yaw angle - K_{pn} and β_0 , angle between wall friction and velocity at the boundary layer edge

Figure 11 – *OALT25* airfoil with yaw angle - skin friction components in directions perpendicular and parallel to the leading edge

Figure 12 – *OALT25* airfoil with yaw angle - Velocity profiles in parallel and normal directions to the leading edge

mesh	y_1^+	C_f (x10 ³)	$\Delta C_f / C_{f_{ref}}$ (%)	nodes removed	nodes in the BL	CPU speed up
45x81	0.1-0.5	2.82	0	0	46	1
45x81	0.1-0.5	2.795	0.9	0	46	0.97
45x71	4-12	2.68	4.9	10	37	1.75
45x61	18-50	2.84	0.7	20	27	4.1
45x51	40-100	2.85	1.05	25	22	5.5
45x56	80-190	2.84	0.7	30	18	7.9
45x46	145-360	2.91	3.2	35	12	13.3

Table 1 – *Influence of the first cell size on the skin friction at $x/c=0.97$
- $k-l$ model*