Assessment of Several Turbulence Models Applied to Supersonic Flows in Three-Dimensions – Part I

Edisson S. G. Maciel
Aeronautical Engineering Division, ITA (Aeronautical Technological Institute), SP, Brazil

Email address
edisavio@edissonsavio.eng.br

Abstract
In the present work, the Van Leer and the Liou and Steffen Jr. flux vector splitting schemes are applied to the three-dimensional Favre-averaged Navier-Stokes equations. The Cebeci and Smith and Baldwin and Lomax algebraic models and the Jones and Launder and Wilcox and Rubesin two-equation models are used in order to close the problem. The physical problem under study is the supersonic flow around a blunt body. The results have demonstrated that the Van Leer scheme using the Wilcox and Rubesin turbulence model has yielded the best value of the stagnation pressure at the blunt body’s nose.

1. Introduction

Conventional non-upwind algorithms have been used extensively to solve a wide variety of problems ([1]). Conventional algorithms are somewhat unreliable in the sense that for every different problem (and sometimes, every different case in the same class of problems) artificial dissipation terms must be specially tuned and judicially chosen for convergence. Also, complex problems with shocks and steep compression and expansion gradients may defy solution altogether.

Upwind schemes are in general more robust but are also more involved in their derivation and application. Some upwind schemes that have been applied to the Euler equations are: [2-3]. Some comments about these methods are reported below:

[2] suggested an upwind scheme based on the flux vector splitting concept. This scheme considered the fact that the convective flux vector components could be written as flow Mach number polynomial functions, as main characteristic. Such polynomials presented the particularity of having the minor possible degree and the scheme had to satisfy seven basic properties to form such polynomials. This scheme was presented to the Euler equations in Cartesian coordinates and three-dimensions.

[3] proposed a new flux vector splitting scheme. They declared that their scheme was simple and its accuracy was equivalent and, in some cases, better than the [4] scheme accuracy in the solutions of the Euler and the Navier-Stokes equations. The scheme was robust and converged solutions were obtained so fast as the [4] scheme. The authors proposed the approximated definition of an advection Mach number at the cell face, using its neighbor cell values via associated characteristic velocities. This interface Mach number was so used to determine the upwind extrapolation of the convective quantities.

In relation to turbulent flow simulations, [5] applied the Navier-Stokes equations to transonic flows problems along a convergent-divergent nozzle and around the NACA 0012 airfoil. The [6] model was used to close the problem. Three algorithms were implemented: the [7] explicit scheme, the [8] implicit scheme and the [9] explicit scheme. The results have shown that, in general terms, the [7] and the [9] schemes have presented better solutions.
presented good results. The algorithms of [12] and of [14] were compared and presented good results.

In terms of two-equation models, [15] has presented in their two-equation turbulence model formulation. The steady state supersonic flow around a blunt body configuration was studied. The physical problem under study is second order accurate. The algorithm is accelerated to the steady state solution using a spatially variable time step. This technique has proved excellent gains in terms of convergence rate as reported in [20-21]. The results have demonstrated that the [2] scheme using the [19] turbulence model has yielded the best value of the stagnation pressure at the blunt body’s nose.

2. Three-Dimensional Navier-Stokes Equations

The three-dimensional flow is modeled by the Navier-Stokes equations, which express the conservation of mass and energy as well as the momentum variation of a viscous, heat conducting and compressible media, in the absence of external forces. The Navier-Stokes equations are presented in their two-equation turbulence model formulation. For the algebraic models, these two-equations are neglected and the [2-3] algorithms are applied only to the original five conservation equations. The integral form of these equations may be represented by:

\[
\frac{\partial}{\partial t} \int_V Q \, dV + \int_S (E_e - E_v) n_x + (F_z - F_v) n_y + (G_e - G_v) n_z \, dS + \int_M M \, dV = 0, \tag{1}
\]

where \( Q \) is written for a Cartesian system, \( V \) is the cell volume, \( n_x, n_y, \) and \( n_z \) are components of the unity vector normal to the cell boundary, \( S \) is the flux area, \( E_e, F_z, \) and \( G_v \) are the components of the convective, or Euler, flux vector, \( E_v, F_v, \) and \( G_v \) are solved using an upwind discretization on a structured mesh. The [11] and [6] algebraic models and the [18] and [19] \( k-\varepsilon \) and \( k-\omega \) two-equation models, respectively, are used in order to close the problem. The physical problem under study is the supersonic flow around a blunt body configuration. The implemented schemes are first-order accurate in space. The time integration uses a Runge-Kutta method of five stages and is second order accurate. The algorithm is accelerated to the steady state solution using a spatially variable time step. This technique has proved excellent gains in terms of convergence rate as reported in [20-21]. The results have demonstrated that the [2] scheme using the [19] turbulence model has yielded the best value of the stagnation pressure at the blunt body’s nose.

\[
Q = \begin{pmatrix}
\rho \\
\rho u \\
\rho v \\
\rho w \\
\rho \varepsilon \\
\rho s
\end{pmatrix},
E_e = \begin{pmatrix}
\rho u \\
\rho u^2 + p \\
\rho uv \\
\rho uw \\
(e + p)u
\end{pmatrix},
F_v = \begin{pmatrix}
\rho v \\
\rho vu + \tau_{xx} \\
\rho vv + \tau_{xy} \\
\rho vw + \tau_{xz} \\
(e + p)v
\end{pmatrix},
G_v = \begin{pmatrix}
\rho w \\
\rho wu + \tau_{yx} \\
\rho vv + \tau_{yy} \\
\rho wv + \tau_{yz} \\
(e + p)w
\end{pmatrix},
\]

\[
E_v = \begin{pmatrix}
0 \\
t_{xx} + \tau_{xx} \\
t_{xy} + \tau_{xy} \\
t_{zx} + \tau_{zx} \\
f_x
\end{pmatrix},
F_v = \begin{pmatrix}
0 \\
t_{xy} + \tau_{xy} \\
t_{yy} + \tau_{yy} \\
t_{yz} + \tau_{yz} \\
f_y
\end{pmatrix},
G_v = \begin{pmatrix}
0 \\
t_{xz} + \tau_{xz} \\
t_{yz} + \tau_{yz} \\
t_{zz} + \tau_{zz} \\
f_z
\end{pmatrix}.
\tag{2}
\]
where the components of the viscous stress tensor are defined as:

\[
\begin{align*}
t_{xx} &= \left[ 2\mu_M \left( \frac{\partial u}{\partial x} - 2/3 \mu_M \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) \right) \right] / \text{Re} ; \\
t_{xy} &= \mu_M \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) / \text{Re} ; \\
t_{xz} &= \mu_M \left( \frac{\partial u}{\partial z} + \frac{\partial v}{\partial x} \right) / \text{Re} ; \\
t_{yy} &= \left[ 2\mu_M \left( \frac{\partial v}{\partial y} \right) - 2/3 \mu_M \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) \right] / \text{Re} ; \\
t_{yz} &= \mu_M \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) / \text{Re} ; \\
t_{zz} &= \left[ 2\mu_M \left( \frac{\partial w}{\partial z} \right) - 2/3 \mu_M \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) \right] / \text{Re}.
\end{align*}
\]

The components of the turbulent stress tensor (Reynolds stress tensor) are described by the following expressions:

\[
\begin{align*}
t_{xx}' &= \left[ 2\mu_M \left( \frac{\partial u}{\partial x} - 2/3 \mu_M \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) \right) \right] / \text{Re} - 2/3 \text{k} ; \\
t_{xy}' &= \mu_M \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) / \text{Re} ; \\
t_{xz}' &= \mu_M \left( \frac{\partial u}{\partial z} + \frac{\partial v}{\partial x} \right) / \text{Re} ; \\
t_{yy}' &= \left[ 2\mu_M \left( \frac{\partial v}{\partial y} \right) - 2/3 \mu_M \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) \right] / \text{Re} - 2/3 \text{k} ; \\
t_{yz}' &= \mu_M \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) / \text{Re} ; \\
t_{zz}' &= \left[ 2\mu_M \left( \frac{\partial w}{\partial z} \right) - 2/3 \mu_M \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) \right] / \text{Re} - 2/3 \text{k}.
\end{align*}
\]

Expressions to \( f_x, f_y \) and \( f_z \) are given below:

\[
\begin{align*}
f_x &= (t_{xx} + \tau_{xx}) u + (t_{xy} + \tau_{xy}) v + (t_{xz} + \tau_{xz}) w - q_x ; \\
f_y &= (t_{xy} + \tau_{xy}) u + (t_{yy} + \tau_{yy}) v + (t_{yz} + \tau_{yz}) w - q_y ; \\
f_z &= (t_{xz} + \tau_{xz}) u + (t_{yz} + \tau_{yz}) v + (t_{zz} + \tau_{zz}) w - q_z .
\end{align*}
\]

where \( q_x, q_y \) and \( q_z \) are the Fourier heat flux components and are given by:

\[
\begin{align*}
q_x &= -\gamma / \text{Re} (\mu_M / \text{Pr} + \mu_t / \text{Pr}_t) \frac{\partial e_i}{\partial x} ; \\
q_y &= -\gamma / \text{Re} (\mu_M / \text{Pr} + \mu_t / \text{Pr}_t) \frac{\partial e_i}{\partial y} ; \\
q_z &= -\gamma / \text{Re} (\mu_M / \text{Pr} + \mu_t / \text{Pr}_t) \frac{\partial e_i}{\partial z} .
\end{align*}
\]

The diffusion terms related to the k-s equations are defined as:

\[
\begin{align*}
\alpha_x &= 1 / \text{Re} (\mu_M + \mu_t / \sigma_k) \frac{\partial k}{\partial x} ; \\
\alpha_k &= 1 / \text{Re} (\mu_M + \mu_t / \sigma_s) \frac{\partial k}{\partial y} ; \\
\alpha_s &= 1 / \text{Re} (\mu_M + \mu_t / \sigma_s) \frac{\partial s}{\partial z} ; \\
\beta_x &= 1 / \text{Re} (\mu_M + \mu_t / \sigma_k) \frac{\partial s}{\partial x} ; \\
\beta_k &= 1 / \text{Re} (\mu_M + \mu_t / \sigma_s) \frac{\partial e_i}{\partial y} ; \\
\beta_s &= 1 / \text{Re} (\mu_M + \mu_t / \sigma_s) \frac{\partial e_i}{\partial z} .
\end{align*}
\]

In the above equations, \( p \) is the fluid density; \( u, v \) and \( w \) are Cartesian components of the velocity vector in the \( x, y \) and \( z \) directions, respectively; \( e \) is the total energy per unit volume; \( p \) is the static pressure; \( k \) is the turbulence kinetic energy; \( s \) is the second turbulent variable, which is the rate of dissipation of the turbulence kinetic energy (k-ε or k-ω² model) for this work; the \( t \)'s are viscous stress components; \( τ \)'s are the Reynolds stress components; the \( q \)'s are the Fourier heat flux components; \( M_k \) takes into account the production and the dissipation terms of \( k \); \( M_s \) takes into account the production and the dissipation terms of \( s \); \( \mu_M \) and \( \mu_T \) are the molecular and the turbulent viscosities, respectively; \( \text{Pr}_L \) and \( \text{Pr}_T \) are the laminar and the turbulent Prandtl numbers, respectively; \( \sigma_k \) and \( \sigma_s \) are turbulence coefficients; \( \gamma \) is the ratio of specific heats; \( \text{Re} \) is the viscous Reynolds number, defined by:

\[
\text{Re} = \rho V_{\text{REF}} / \mu_M ,
\]

where \( V_{\text{REF}} \) is a characteristic flow velocity and \( l_{\text{REF}} \) is a configuration characteristic length. The internal energy of the fluid, \( e_i \), is defined as:

\[
e_i = e / \rho - 0.5 (u^2 + v^2 + w^2) .
\]

The molecular viscosity is estimated by the empiric Sutherland formula:

\[
\mu_M = \beta T^{1/2} / (1 + S / T) ,
\]

where \( T \) is the absolute temperature (K), \( b = 1.458 \times 10^{-6} \) Kg/(m.s.K¹/²) and \( S = 110.4 \) K, to the atmospheric air in the standard atmospheric conditions ([22]).

The Navier-Stokes equations are dimensionless in relation to the freestream density, \( \rho_\infty \), the freestream speed of sound, \( a_\infty \), and the freestream molecular viscosity, \( \mu_\infty \). The system is closed by the state equation for a perfect gas:

\[
p = (\gamma - 1) \left[ e - 0.5 \rho (u^2 + v^2 + w^2) \right] - \rho_k ,
\]
considering the ideal gas hypothesis. The total enthalpy is given by \( H = (e + p)/\rho \).

3. Van Leer and Liou and Steffen Jr. Algorithms

The space approximation of the integral Equation (1) yields an ordinary differential equation system given by:

\[
R_{i,j,k} = R_{i-1/2,j,k} + R_{i+1/2,j,k} + R_{i,j+1/2,k} + R_{i,j-1/2,k} + R_{i,j,k-1/2} + R_{i,j,k+1/2} ,
\]

with \( R_{i,j,k} \) representing the net flux (residual) of the conservation of mass, conservation of momentum and conservation of energy in the volume \( V_{i,j,k} \). The residual is calculated as:

\[
V_{i,j,k} \frac{dQ_{i,j,k}}{dt} = -R_{i,j,k} ,
\]

with \( R_{i,j,k} \) representing the net flux (residual) of the conservation of mass, conservation of momentum and conservation of energy in the volume \( V_{i,j,k} \). The residual is calculated as:

V_{i,j,k} dQ_{i,j,k} /dt = -R_{i,j,k} ,

understood as a sum of the arithmetical average between the right (R) and the left (L) states of the cell face \((i+1/2,j,k)\), involving volumes \((i+1,j,k)\) and \((i,j,k)\), respectively, multiplied by the interface Mach number, plus a scalar dissipative term, as shown in [3]. Hence,

\[
M_{i+1/2,j,k} = M_L^+ + M_R^- ,
\]

where the separated Mach numbers are defined by [2]:

\[
M = \begin{cases} 
M_{\text{L}} & \text{if } M \geq 1; \\
0.25(M+1)^2 & \text{if } |M| < 1; \\
0 & \text{if } M \leq -1;
\end{cases}
\]

\[
M' = \begin{cases} 
M_{\text{L}} & \text{if } M \geq 1; \\
-0.25(M-1)^2 & \text{if } |M| < 1; \\
M_{\text{R}} & \text{if } M \leq -1.
\end{cases}
\]

\[
M_L \text{ and } M_R \text{ represent the Mach numbers associated with the left and the right states, respectively. The advection Mach number is defined by:}
\]

\[
M = \left( S_x u + S_y v + S_z w \right) / \left( |\mathbf{S}| \right).
\]

The pressure at the face \((i+1/2,j,k)\), related to the cell \((i,j,k)\), is calculated by a similar formula:

\[
p_{i+1/2,j,k} = \frac{p_L^{\infty} + p_R^{\infty}}{2} ,
\]

with \( p^{\infty} \) denoting the pressure separation and due to [2]:

\[
\text{Figure 1. Computational Cell.}
\]
purely upwind, using either the left state or the right state to
calculated. The normal distance from the wall to the studied cell is
space.

The above equations clearly show that to a supersonic cell
face Mach number, the [2] scheme represents a discretization
purely upwind, using either the left state or the right state to
the convective terms and to the pressure, depending of the
Mach number signal. This [2] scheme is first order accurate in
the convective terms and to the pressure, depending of the
shear stress at wall are calculated. After that, the
layer thickness, the
modeling function is to develop approximations to these

\[ \phi_{i+1/2,j,k}^{(2)} = \phi_{i+1/2,j,k}^{(0)} \]

The time integration is performed using an explicit
Runge-Kutta method of five stages, second order accurate,
and can be represented in generalized form by:

\[ \frac{\partial u}{\partial t} = \frac{1}{V} \frac{\partial}{\partial x} \left( \frac{1}{\sqrt{\rho}} \int u \left( \hat{n}, \text{dS} \right) \right) = \frac{1}{V} \int u \text{dS} \equiv \frac{1}{V} \left[ 0.5(u_{j,k} + u_{j+1,k})S_{j+1,j,k} + 0.5(u_{j,k} + u_{j-1,k})S_{j-1,j,k} + 0.5(u_{j,k} + u_{j,k+1})S_{j,k+1,j} + 0.5(u_{j,k} + u_{j,k-1})S_{j,k-1,j} \right] \]

\[ \phi_{i+1/2,j,k}^{(k)} = \phi_{i+1/2,j,k}^{(0)} - \alpha_k \Delta t_{i,j} \left[ R \left( Q_{i,j}^{(k-1)} \right) / V_{i,j} + M \left( Q_{i,j}^{(k-1)} \right) \right] \]

4. Turbulence Models

4.1. Cebeci and Smith Turbulence Model

The problem of the turbulent simulation is in the
calculation of the Reynolds stress. Expressions involving
velocity fluctuations, originating from the averaging process,
represent six new unknowns. However, the number of
equations keeps the same and the system is not closed. The
modeling function is to develop approximations to these
correlations. To the calculation of the turbulent viscosity
according to the [11] model, the boundary layer is divided in
internal and external.

Initially, the \( \nu_s \) kinematic viscosity at wall and the \( \tau_{x,y,w} \)
shear stress at wall are calculated. After that, the \( \delta \) boundary
layer thickness, the \( \delta_{lin} \) linear momentum thickness and the
\( \nu_{lin} \) boundary layer tangential velocity are calculated. So,
the \( N \) normal distance from the wall to the studied cell is
calculated. The \( N \) term is obtained from:

\[ N^+ = \sqrt{Re} \sqrt{\tau_{x,y,w} / \rho_w} \frac{N}{\nu_w} \]

where \( \rho_w \) is the wall density. The van Driest damping factor is

The definition of a dissipative term \( \phi \) determines
the particular formulation of the convective fluxes. The following
choice corresponds to the [2] scheme, according to [14]:

\[ Q_{i,j}^{(0)} = Q_{i,j}^{(k)} \]

\[ Q_{i,j}^{(k)} = Q_{i,j}^{(0)} - \alpha_k \Delta t_{i,j} \left[ R \left( Q_{i,j}^{(k-1)} \right) / V_{i,j} + M \left( Q_{i,j}^{(k-1)} \right) \right] \]

\[ Q_{i,j}^{(n+1)} = Q_{i,j}^{(k)} \]

with \( k = 1, ..., 5; \alpha_1 = 1/4, \alpha_2 = 1/6, \alpha_3 = 3/8, \alpha_4 = 1/2 \) and \( \alpha_5 = 1 \). The gradients of the primitive variables are calculated using
the Green theorem, which considers that the gradient of a
primitive variable is constant at the volume and that the
volume integral which defines the gradient is replaced by a
surface integral ([24]). To the \( \partial u / \partial x \) gradient, for example, it is possible to write:

\[ D = 1 - e^{(-N^+ \sqrt{\rho_w} \mu_w / \mu / A^+)} \]

where \( A^+ = 26 \) and \( \mu_w \) is the wall molecular viscosity.
After that, the \( V \nu / dN \) normal to the wall gradient of the
tangential velocity is calculated and the internal turbulent
viscosity is given by:

\[ \mu_{yi} = Re \rho (\kappa ND)^2 \nu T dN \]

where \( \kappa \) is the von Kármán constant, which has the value 0.4.
The intermittency function of Klebanoff is calculated to the
external viscosity by:

\[ g_{Kleb} (N) = \left[ 1 + 5.5 (N/\delta)^3 \right]^{-1} \]

With it, the external turbulent viscosity is calculated by:

\[ \mu_{Te} = Re(0.0168) \nu T LM \delta_{LM} g_{Kleb} \]

Finally, the turbulent viscosity is chosen from the internal
and the external viscosities:

\[ \mu_T = MIN(\mu_{Ti}, \mu_{Te}) \]
4.2. Baldwin and Lomax Turbulence Model

To the calculation of the turbulent viscosity according to the [6] model, the boundary layer is again divided in internal and external. In the internal layer,

\[ \mu_{T_i} = \rho u_{\text{mix}} \| \omega \| \]  and \[ l_{\text{mix}} = \kappa N \left( 1 - e^{-N/\Lambda_k} \right). \]  \hspace{1cm} (33)

In the external layer,

\[ \mu_{T_e} = \rho \alpha C_{cp} F_{\text{wake}} F_{\text{Kleb}} \left( N; N_{\text{max}} / C_{\text{Kleb}} \right), \]  \hspace{1cm} (34)

with:

\[ F_{\text{wake}} = \min \left[ N_{\text{max}} N_{\text{mix}} ; C_{wk} N_{\text{max}} U_{\text{diff}} / F_{\text{max}} \right]; \]

\[ F_{\text{max}} = l / \kappa \left[ \max \left( l_{\text{mix}} \| \omega \| \right) \right]. \]  \hspace{1cm} (36)

Hence, \( N_{\text{max}} \) is the value of \( N \) where \( l_{\text{mix}} \| \omega \| \) reached its maximum value and \( l_{\text{mix}} \) is the Prandtl mixture length. The constant values are: \( \kappa = 0.4 \), \( \alpha = 0.0168 \), \( A^v = 26 \), \( C_{cp} = 1.6 \), \( C_{\text{Kleb}} = 0.3 \) and \( C_{wk} = 1 \). \( F_{\text{Kleb}} \) is the intermittent function of Klebanoff given by:

\[ F_{\text{Kleb}} \left( N \right) = \left[ 1 + 5.5 \left( C_{\text{Kleb}} N / N_{\text{max}} \right)^6 \right]^{-1}, \]

\[ \| \omega \| \] is the magnitude of the vorticity vector and \( U_{\text{diff}} \) is the maximum velocity value in the boundary layer case. To free shear layers,

\[ U_{\text{diff}} = \left( \sqrt{u^2 + v^2 + w^2} \right)_{\text{max}} - \left( \sqrt{u^2 + v^2 + w^2} \right)_{N = N_{\text{max}}} . \] \hspace{1cm} (38)

4.3. Jones and Launder Turbulence Model

In the [18] turbulence model, \( s = \varepsilon \). To define the turbulent

\[ D_\varepsilon = \left[ -\frac{2}{3} C_{\varepsilon 1} \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) / \omega + 2C_{\mu} D_{V_M} \left( \frac{\partial^2 u}{\partial y^2} \right)^2 / \omega^3 - C_{\varepsilon 2} E_\varepsilon \right] \rho \omega^2 / Re ; \] \hspace{1cm} (47)

with the second damping factor \( E_\varepsilon \) defined as:

\[ E_\varepsilon = 1 - 2/9 e^{\left[ -4\text{Re}^2 / 36 \right]} . \]

The source term denoted by \( M \) in the governing equation contains the production and dissipation terms of \( \kappa \) and \( \varepsilon \). To the [18] model, the \( M_k \) and \( M_\varepsilon \) terms have the following expressions:

\[ M_k = -P_k - D_k \] \quad and \quad \[ M_\varepsilon = -P_\varepsilon - D_\varepsilon , \] \hspace{1cm} (48)

where:

\[ P = \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \rho \omega / \text{Re} ; \]

\[ D_k = \left[ \frac{2}{3} \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} / \omega + \frac{2C_{\mu} D_{V_M} \left( \frac{\partial^2 u}{\partial y^2} \right)^2 / \omega^3 - C_{\varepsilon 2} E_\varepsilon }{ \rho \omega^2 / \text{Re} } \right] \] \hspace{1cm} (42)

\[ D_\varepsilon = 1 - \omega e^{\left[ -4\text{Re}^2 / 36 \right]} . \] \hspace{1cm} (49)

The turbulent viscosity is expressed in terms of \( \kappa \) and \( \omega \) as:

\[ \mu_T = \text{Re} C_\mu D_{pk} / \omega \] \hspace{1cm} (50)

The source term denoted by \( M \) in the governing equation contains the production and dissipation terms of \( \kappa \) and \( \varepsilon \). To the [19] model, the \( M_k \) and \( M_\varepsilon \) terms have the following expressions:

\[ M_k = -P_k - D_k \] \quad and \quad \[ M_\varepsilon = -P_\varepsilon - D_\varepsilon , \] \hspace{1cm} (42)

viscosity, or eddy viscosity, it is necessary to define the turbulent Reynolds number:

\[ \text{Re}_T = k / (v_M \omega), \] \quad with: \( v_M = \mu_M / \rho \) . \hspace{1cm} (48)

4.4. Wilcox and Rubesin Turbulence Model

In the [19] turbulence model, \( s = \omega^2 \). To define the turbulent

\[ \text{Re}_T = k / (v_M \omega), \] \quad with: \( v_M = \mu_M / \rho \) . \hspace{1cm} (48)

It is also necessary to determine the D damping factor:
\[
P = \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \frac{\partial u}{\partial y} + \left( \frac{\partial u}{\partial y} + \frac{\partial w}{\partial z} \right) \frac{\partial w}{\partial y} + \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) \frac{\partial z}{\partial z} ;
\]

\[
P_k = \frac{DP}{\omega^2} \rho \omega \kappa / Re ;
\]

\[
D_k = \left[ -\frac{2}{3} \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) \right] \rho \omega \kappa / Re ;
\]

\[
P_{\omega^2} = \left( \frac{\gamma_{EP}}{\omega^2} \right) \rho \omega^3 / Re ;
\]

\[
D_{\omega^2} = \left[ -\frac{2}{3} \gamma_{\omega} \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) / \omega - \left( \beta + \frac{2}{\sigma_{\omega}} \left( \frac{\partial \sqrt{\omega}}{\partial y} \right) \right) \right] \rho \omega^3 / Re ,
\]

with the second damping factor \( E \) defined as: \( E = 1 - \omega e^{(-0.5 \alpha \omega)} \). The closure coefficients adopted to the [19] model assume the following values: \( \sigma_k = 2.0 \); \( \sigma_{\omega^2} = 2.0 \); \( \beta^* = 0.09 \); \( \beta = 0.15 \); \( \alpha = 0.99174 \); \( \gamma_{\omega} = 0.9 \); \( \Pr_{d{\omega}} = 0.72 \); \( \Pr_{d{\omega}} = 0.9 \).

### 5. Spatially Variable Time Step

The basic idea of this procedure consists in keeping a constant CFL number in all calculation domain; thus allowing that appropriated time steps to each specific mesh region could be used during the convergence process. Hence, to a viscous simulation and according to the [25] work, it is possible to write:

\[
\Delta t_{i,j,k} = \left( \frac{CFL(\Delta t_c, \Delta t_v)}{\Delta t_c + \Delta t_v} \right)_{i,j,k} ,
\]

with \( \Delta t_c \) being the convective time step and \( \Delta t_v \) being the viscous time step. These quantities are defined as:

\[
(\Delta t_c)_{i,j,k} = \frac{V_{i,j,k}}{\lambda_c} ,
\]

\[
(\lambda_c)_{i,j,k} = \max \left( \lambda_{i-j/2,k} , \lambda_{i+1/2,j,k} , \lambda_{i,j+1/2,k} , \lambda_{i,j-k/2} , \lambda_{i-j/2,k} , \lambda_{i+1/2,j,k} \right) ;
\]

\[
(\lambda_{\text{max}})_{i,j,k} = \max \left( u_{\text{int}} n_x + v_{\text{int}} n_y + w_{\text{int}} n_z + a_{\text{int}} \right) S_{\text{int}} ;
\]

\[
(\Delta t_v)_{i,j,k} = K_v \frac{V_{i,j,k}}{\lambda_v} - \left( p1_{i,j,k} = \frac{\gamma^{3/2} M_{\infty}^2}{(Re \Pr_{d{\omega}}) V_{i,j,k}} ;
\]

\[
(p2)_{i,j,k} = \frac{\mu_{i+j/2,k}+\mu_{i+j/2,k}}{\mu_{i+j/2,k}+\mu_{i+j/2,k}} S_{i+j/2,k} + \frac{\mu_{i+j/2,k}+\mu_{i+j/2,k}}{\mu_{i+j/2,k}+\mu_{i+j/2,k}} S_{i+j/2,k} + \frac{\mu_{i+j/2,k}+\mu_{i+j/2,k}}{\mu_{i+j/2,k}+\mu_{i+j/2,k}} S_{i+j/2,k} ;
\]

\[
(\lambda_v)_{i,j,k} = \left( p1 \times p2 \right)_{i,j,k} ,
\]

where interface properties are calculated by arithmetical average, \( M_{\infty} \) is the freestream Mach number, \( \mu \) is the fluid molecular viscosity and \( K_v \) is equal to 0.25, as recommended by [25].

### 6. Initial and Boundary Conditions

The initial and boundary conditions to the [6, 11] turbulence models are the same for perfect gas formulation. Details of these conditions can be found in [26-27]. For the \( k-\varepsilon \), one has:

#### 6.1. Initial Condition

Freestream values, at all grid cells, are adopted for all flow properties as initial condition, as suggested by [28-29]. Therefore, the vector of conserved variables is defined as:
\[
Q_{i,j,k} = \begin{cases} 
1 & M_u \cos \alpha - M_w \sin \alpha \cos \theta - M_w \sin \alpha \sin \theta \frac{1}{(\gamma - 1)} + 0.5M_w^2 f_1 K - f_2 K \end{cases} \)
\]

where \(\alpha\) is the angle of attack, \(\theta\) is the longitudinal angle, \(K\) is the kinetic energy of the mean flow and \(f_1\) and \(f_2\) are fractions. The kinetic energy of the mean flow is determined, considering the present dimensionless, as \(K = 0.5M_w^2\). The values adopted for \(f_1\) and \(f_2\) in the present work were 0.005 and 0.2, respectively.

### 6.2. Boundary Conditions

The boundary conditions are basically of four types: solid wall, entrance, exit and far field. These conditions are implemented with the help of ghost cells.

1. Wall condition: At a solid boundary the non-slip condition is enforced. Therefore, the tangential velocity component of the ghost volume at wall has the same magnitude as the respective velocity component of its real neighbor cell, but opposite signal. In the same way, the normal velocity component of the ghost volume at wall is equal in value, but opposite in signal, to the respective velocity component of its real neighbor cell.

The normal pressure gradient of the fluid at the wall is assumed to be equal to zero in a boundary-layer like condition. The same hypothesis is applied for the normal temperature gradient at the wall, assuming an adiabatic wall. The normal gradient of the turbulence kinetic energy at the wall is also assumed to be equal to zero.

From the above considerations, density, pressure and turbulence kinetic energy at the ghost volume are extrapolated from the respective values of its real neighbor volume (zero order extrapolation). The total energy is obtained by the perfect gas law and the rate of dissipation of the turbulence kinetic energy is determined by the following expression, considering production-destruction equilibrium:

\[
(p\rho)_{\text{ghost}} = C_{\mu}^{1/4} k_w^{3/2} (0.41d_n),
\]

where \(k_w\) is the wall turbulence kinetic energy and \(d_n\) is the distance of the first mesh cell to the wall.

The properties of the first real volumes \((j = 1)\) are necessary to be determined, aiming to guarantee that the u profile is correctly calculated by the numerical scheme. The \(u\) component of these cells is determined by the “wall law”. It is initially necessary to calculate the wall shear stress, which is defined as:

\[
\tau_w = \rho u C_{\mu}^{0.25} k_w^{0.5} / u^+,
\]

where \(u^+\) is defined as:

\[
\begin{align*}
\quad u^+ &= d^+, \quad d^+ < 5; \\
\quad u^+ &= -3.05 + 5 \ln d^+, \quad 5 \leq d^+ < 30; \\
\quad u^+ &= 5.5 + 2.5 \ln d^+, \quad 30 \leq d^+ < 2000,
\end{align*}
\]

with: \(d^+ = \rho C_{\mu}^{0.25} k_w^{0.5} d_n / \mu_M\). The value of \(u\) of the real volume at the wall is obtained from:

\[
u = d_n \tau_w / \mu_M + u_{\text{ghost}}.
\]

The \(v\) component is extrapolated from the ghost volume, with opposite signal, and the pressure is extrapolated from the real volume at \(j = 2\). The turbulence kinetic energy is defined by its value at wall and the total energy of this volume is determined by the state equation for a perfect gas. The rate of dissipation of the turbulence kinetic energy in this volume is determined by Eq. (64).

2. Entrance condition:

2.1. Subsonic flow: Six properties are specified and one extrapolated. This approach is based on information propagation analysis along characteristic directions in the calculation domain (29). In other words, for subsonic flow, six characteristic propagate information point into the computational domain. Thus six flow properties must be fixed at the inlet plane. Just one characteristic line allows information to travel upstream. So, one flow variable must be extrapolated from the grid interior to the inlet boundary. The pressure was the extrapolated variable from the real neighbor volumes, for the studied problem. Density and velocity components adopted values of freestream flow. The turbulence kinetic energy and the rate of dissipation of the turbulence kinetic energy were fixed with the values of the initial condition, with the modification of \(K = 0.5u^2\). The total energy is determined by the state equation of a perfect gas.

2.2. Supersonic flow: In this case no information travels upstream; therefore all variables are fixed with their of freestream values.

3. Exit condition:

3.1. Subsonic flow: Six characteristic propagate information outward the computational domain. Hence, the associated variables should be extrapolated from interior information. The characteristic direction associated to the “(qnormal-a)” velocity should be specified because it point inward to the computational domain (29). In this case, the ghost volume pressure is specified from its initial value. Density, velocity components, the turbulence kinetic energy, and the rate of dissipation of the turbulence kinetic energy are extrapolated. The total energy is obtained from the state equation of a perfect gas.

3.2. Supersonic flow: All variables are extrapolated from interior grid cells, as no flow information can make its way upstream. In other words, nothing can be fixed.

4. Far field condition: The mean flow kinetic energy is assumed to be \(K = 0.5u^2\) and the turbulence kinetic energy at the far field adopts the value \(kff = 0.005K\), or 0.5% of \(K\). The turbulence dissipation energy is assumed \(\epsilon_{ff} = 0.20K\) or 20% of \(K\).

For the \(k-\alpha\), one has:
6.3. Initial Condition

Freestream values, at all grid cells, are adopted for all flow properties as initial condition, as suggested by [28-29]. Therefore, the vector of conserved variables is defined as:

$$ Q_{i,1} = \begin{bmatrix} 1 & M_w \cos \alpha & M_w \sin \alpha \cos \theta \end{bmatrix}^T, \quad \text{where } t_\alpha = \text{the freestream turbulent kinetic energy and } s_{\infty} = \text{the freestream turbulent vorticity or the squared of this value. These parameters assumes the following values as using the [19] model: } t_\alpha = 1.0 \times 10^{-6} \text{ and } s_{\infty} = \left[ \frac{10 u_{\infty}}{l_{\text{REF}}} \right]^2. $$

with \( u_{\infty} \) the freestream u Cartesian component and \( l_{\text{REF}} \) a characteristic length, the same adopted in the definition of the Reynolds number.

6.4. Boundary Conditions

The boundary conditions are basically of four types: solid wall, entrance, exit and far field. These conditions are implemented with the help of ghost cells.

(1) Wall condition: At a solid boundary the non-slip condition is enforced. Therefore, the tangent velocity component of the ghost volume at wall has the same magnitude as the respective velocity component of its real neighbor cell, but opposite signal. In the same way, the normal velocity component of the ghost volume at wall is equal in value, but opposite in signal, to the respective velocity component of its real neighbor cell.

The normal pressure gradient of the fluid at the wall is assumed to be equal to zero in a boundary-layer like condition. The same hypothesis is applied for the normal temperature gradient at the wall, assuming an adiabatic wall. The normal gradient of the turbulence kinetic energy at the wall is also assumed to be equal to zero.

From the above considerations, density and pressure are extrapolated from the respective values of its real neighbor volume (zero order extrapolation). The total energy is obtained by the state equation for a perfect gas. The turbulent kinetic energy and the turbulent vorticity at the ghost volumes are determined by the following expressions:

$$ k_{\text{ghost}} = 0.0 \quad \text{and} \quad \omega_{\text{ghost}}^2 = \left( \frac{38}{3} \nu M \right) \left( \frac{1}{\beta d_n^2} \right)^2, \quad (69) $$

where \( \beta \) assumes the value 3/40 and \( d_n \) is the distance of the first cell to the wall.

(2) Entrance condition:

(2.1) Subsonic flow: Six properties are specified and one extrapolated. This approach is based on information propagation analysis along characteristic directions in the calculation domain ([29]). In other words, for subsonic flow, six characteristic propagate information pointing into the computational domain. Thus six flow properties must be fixed at the inlet plane. Just one characteristic line allows information to travel upstream. So, one flow variable must be extrapolated from the grid interior to the inlet boundary. The pressure was the extrapolated variable from the real neighbor volumes, for the studied problem. Density and velocity components adopted values of freestream flow. The turbulence kinetic energy and the vorticity were fixed with the values of the initial condition. The turbulence kinetic energy receives the value 0.01 of \( K \). The total energy is determined by the state equation of a perfect gas.

(2.2) Supersonic flow: In this case no information travels upstream; therefore all variables are fixed with their of freestream values.

(3) Exit condition:

(3.1) Subsonic flow: Six characteristic propagate information outward the computational domain. Hence, the associated variables should be extrapolated from interior information. The characteristic direction associated to the “(normal=0)” velocity should be specified because it points inward to the computational domain ([29]). In this case, the ghost volume pressure is specified from its initial value. Density, velocity components, the turbulence kinetic energy, and the vorticity are extrapolated. The total energy is obtained from the state equation of a perfect gas.

(3.2) Supersonic flow: All variables are extrapolated from interior grid cells, as no flow information can make its way upstream. In other words, nothing can be fixed.

(4) Far field condition: The mean flow kinetic energy is assumed to be \( K = 0.5u^2 \) and the turbulence kinetic energy at the far field adopts the value \( k_{\text{ff}} = 0.01K \), or 1% of \( K \). The turbulence vorticity is determined by its freestream value.

7. Results

Tests were performed in an INTEL Core i7 processor of 2.10GHz and 8.0Gbytes of RAM microcomputer in a Windows 7.0 environment. Three orders of reduction of the maximum residual in the field were considered to obtain a converged solution. The residual was defined as the value of the discretized conservation equation. The entrance or attack angle was adopted equal to zero, as well the longitudinal angle. The ratio of specific heats, \( \gamma \), assumed the value 1.4.

Figure 2 shows the blunt body configuration, whereas Fig. 3 shows the blunt body mesh. A mesh of 53x50x10 points or composed of 22,932 hexahedron cells and 26,500 nodes was generated, employing an exponential stretching of 5.0% in the \( \eta \) direction.
The initial data of the simulations is described in Tab. 1.

### Table 1. Initial Conditions.

<table>
<thead>
<tr>
<th>$M$</th>
<th>$\alpha$</th>
<th>$\theta$</th>
<th>Altitude</th>
<th>$L_\infty$</th>
<th>Re</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.0</td>
<td>0.0°</td>
<td>0.0°</td>
<td>40,000m</td>
<td>2.0m</td>
<td>4.75x10^5</td>
</tr>
</tbody>
</table>

### 7.1. Cebeci and Smith Results

Figures 4 and 5 present the pressure contours obtained by the [2] and [3] schemes, respectively, as using the [11] turbulence model in three-dimensions. Both fields are homogeneous and the pressure contours generated by the [2] scheme is more strength than the respective one generated by the [3] scheme.

Figures 6 and 7 show the Mach number contours obtained by the [2] and [3] numerical schemes, respectively, as using the [11] turbulence model. Both Mach number fields are free of pre-shock oscillations and are homogeneous. The differences between these fields are only in qualitative terms. It is possible to see that the [2] solution develops a region of low Mach number contours close to the wall, resulting from the boundary layer formation. On the contrary, the [3] solution does not yields this region.

Figures 8 and 9 exhibit the translational temperature contours obtained by the [2] and [3] schemes, respectively, as using the [11] turbulence model. The temperature contours generated by the [3] scheme is more intense than the respective one of the [2] scheme. However, the [2] solution presents a zone of high dissipation close to the wall, whereas the [3] scheme does not. There are qualitative differences between the two solutions, but both present homogeneous contours, without oscillations. Some problems with the [3]
solution in the \( k = \text{constant} \) planes are observed, which prejudices the solution repetition in these planes. The [2] solution does not present such problems.

Figure 8. Temperature contours ([2]).

Figure 9. Temperature contours ([3]).

Figure 10. \(-C_p\) distributions.

Figure 11. Wall temperature distributions.

Figure 10 exhibits the \(-C_p\) distributions generated by the [2] and [3] schemes as using the [11] turbulence model. The \(-C_p\) plateau of the [2] scheme is higher than the \(-C_p\) plateau of the [3] scheme. The \(-C_p\) peak at the body’s nose is the same for both schemes. Figure 11 presents the wall temperature distributions generated by the [2] and [3] numerical schemes as using the [11] turbulence model. The temperature distribution is smoother for the [2] scheme. The temperature distribution of the [3] scheme is more intense than the respective one of the [2] scheme. The maximum temperature obtained by the [2] scheme is about 732.0 K, whereas that obtained by the [3] scheme is about 837.0 K.

7.2. Baldwin and Lomax Results

Figure 12 and 13 present the pressure contours obtained by the [2] and [3] schemes, respectively, as using the [6] turbulence model. Again, the pressure contours generated by the [2] scheme is more strength than the one generated by the [3] scheme. Good homogeneity is observed in both solutions.

Figure 12. Pressure contours ([2]).
Figures 14 and 15 show the Mach number contours generated by the [2] and [3] schemes, respectively, as using the [6] turbulence model. Both solutions are identical in quantitative terms, only being different in qualitative terms. The subsonic region close to the body’s wall is again observed in the [2] solution. The subsonic region at the body’s nose is observed, as resulting from the shock slowdown. No pre-shock oscillations are observed in both solutions.

Figures 16 and 17 exhibit the translational temperature contours obtained by the [2] and [3] schemes, respectively, as using the [6] turbulence model. The [3] solution presents higher temperatures in the field than the [2] solution. However, the [2] solution presents a zone of high dissipation close to body’s wall, resulting from intense heat energy exchange and boundary layer interaction.

Figure 18 presents the –Cp distribution generated by the [2] and [3] schemes as using the [6] turbulence model. As can be seen the –Cp plateau obtained by the [2] scheme is higher than the respective one of the [3] scheme. The –Cp peak is
approximately the same for both solutions. Figure 19 shows the wall translational temperature distributions originated by the [2] and [3] schemes as using the [6] turbulence model. The [2] temperature distribution is smoother than the [3] one. The temperature at the body’s nose is higher in the [2] solution. Both temperature distributions increase along the body.

7.3. Jones and Launder Results

Figures 20 and 21 exhibit the pressure contours generated by the [2] and [3] schemes, respectively, as using the [18] turbulence model. The pressure field generated by the [2] scheme is higher than the respective one generated by the [3] scheme. Both pressure fields present good homogeneity properties. Figures 22 and 23 show the Mach number contours obtained by the [2] and [3] schemes, respectively, as using the [18] turbulence model. Both fields are identical in quantitative terms, the difference being in qualitative terms. Particularly, the zone of low Mach number close to the body’s wall is only perceptible in the [2] solution. The region of subsonic Mach number at the body’s nose is well captured by both schemes.

No pre-shock oscillations are observed in both figures. The shock wave is well captured in both solutions.
Figures 24 and 25 exhibit the translational temperature contours obtained by the [2] and [3] schemes, respectively, as using the [18] turbulence model. The temperature field generated by the [3] scheme is slightly more intense than the respective one of the [2] scheme. Good homogeneous properties are observed in both figures. The zone of intense energy exchange, close to the body’s wall, is observed in the [2] solution. Moreover, the zone of intense temperature is observed at the body’s nose in both solutions, as expected. Good symmetry properties are noted in both solutions.

Figure 26 presents the $-C_p$ distribution obtained by the [2] and [3] schemes as using the [18] turbulence model. Opposed to the observed in the ultimate solutions, the [3] scheme presents higher $-C_p$ plateau than the [2] scheme, although both solutions present the same $-C_p$ peak, at the body’s nose.

Figure 27 shows the wall temperature distributions obtained by the [2] and [3] schemes as using the [18] turbulence model. Both solutions present the same temperature peak at the leading edge and both solutions suffer a reduction in temperature distribution along the body’s length, although the [2] solutions recovery some at the body’s trailing edge. The [3] solution suffers the biggest reduction, reaching
a temperature value of approximately 471.0 K.

7.4. Wilcox and Rubesin Results

Figures 28 and 29 show the pressure contours obtained by the [2] and [3] schemes, respectively, as using the [19] turbulence model. As can be observed, the [2] scheme predicts again more severe pressure field than the [3] scheme. The shock is well captured and good symmetry characteristics are noted. Good homogeneity in both solutions is also observed.

Figures 30 and 31 present the Mach number contours generated by the [2] and [3] schemes, respectively, as using the [19] turbulence model. Good symmetry properties are observed, without pre-shock oscillations. Good homogeneity properties are noted. The Mach number fields are identical in quantitative terms, although in qualitative terms some discrepancies are observed. The zone of low Mach number close to the body’s wall is only noted in the [2] solution.

Figures 32 and 33 show the translational temperature contours obtained by the [2] and [3] schemes, respectively, as using the [19] turbulence model. As can be observed, the [3] temperature field is more intense than the [2] temperature field, as observed in the other solutions. The zone of intense energy exchange observed close to the body’s wall is only captured by the [2] scheme. Moreover, the intense temperature region at the body’s nose is again only observed in the [2] solution, being more discrete in the [3] case. Good symmetry properties are observed in both figures.

Figures 34 and 35 present the Mach number contours generated by the [2] and [3] schemes, respectively, as using the [19] turbulence model. As observed in the majority of the solutions, the [2] scheme presents again a pressure plateau higher than the [3] scheme does, although both –Cp peaks at the body’s nose are approximately the same. Figure 35 shows the temperature distributions at wall generated by the [2] and [3] schemes as using the [19] turbulence model. The [2] temperature distributions is smoother than the [3] one. Both temperature
distributions keep approximately the same along the body’s length. The temperature values at the body’s trailing edge are approximately 767.0 K to the [3] scheme and 715.0 K to the [2] scheme.

Figure 32. Temperature contours ([2]).

Figure 33. Temperature contours ([3]).

7.5. Quantitative Analysis

A possibility to quantitative comparison of the turbulent cases is the determination of the stagnation pressure ahead of the configuration. [30] presents a table of normal shock wave properties in its B Appendix. This table permits the determination of some shock wave properties as function of the freestream Mach number. In front of the blunt body configuration, the shock wave presents a normal shock behavior, which permits the determination of the stagnation pressure, behind the shock wave, from the tables encountered in [30]. So it is possible to determine the ratio $\frac{p_r}{p_{\infty}}$ from [30], where $p_r$ is the stagnation pressure in front of the configuration and $p_{\infty}$ is the freestream pressure (equals to $\frac{1}{\gamma}$ to the present dimensionless).

Hence, to this problem, $M_\infty = 3.0$ corresponds to $\frac{p_r}{p_{\infty}} = 12.06$ and remembering that $p_{\infty} = 0.714$, it is possible to conclude that $p_r = 8.61$. Values of the stagnation pressure to the turbulent cases and respective percentage errors are shown in Tab. 2. They are obtained from Figures 4, 5, 12, 13, 20, 21, 28 and 29. As can be observed, the [2] scheme using the [19] turbulence model has presented the best result, with a percentage error of 0.58%.

Table 2. Values of the stagnation pressure and respective percentage errors.

<table>
<thead>
<tr>
<th>Model:</th>
<th>Scheme:</th>
<th>$p_r$</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>[3]</td>
<td>8.84</td>
<td>2.67</td>
</tr>
<tr>
<td></td>
<td>[3]</td>
<td>8.34</td>
<td>3.14</td>
</tr>
<tr>
<td>[18]</td>
<td>[2]</td>
<td>8.35</td>
<td>3.02</td>
</tr>
<tr>
<td></td>
<td>[3]</td>
<td>8.15</td>
<td>5.34</td>
</tr>
<tr>
<td>[19]</td>
<td>[2]</td>
<td>8.56</td>
<td>0.58</td>
</tr>
<tr>
<td></td>
<td>[3]</td>
<td>8.15</td>
<td>5.34</td>
</tr>
</tbody>
</table>

Finally, Table 3 exhibits the computational data of the present simulations. It can be noted that the most efficient is the [3] scheme using the [11] turbulence model. All schemes used a CFL number of 0.10, not necessarily being the maximum CFL number of each one.
As final conclusion of this study, the [19] turbulence model was the best when comparing these four turbulence models: [11], [6], [18] and [19]. In a next paper, the present author will study more four different turbulent models to this same problem trying to identify the best of each group of four and to perform a final analysis to found the best one.

8. Conclusions

In the present work, the [2-3] flux vector splitting schemes are implemented, on a finite-volume context. The three-dimensional Favre-averaged Navier-Stokes equations are solved using an upwind discretization on a structured mesh. The [11] and [6] algebraic models and the [18] and [19] two-equation models are used in order to close the problem. The physical problem under study is the supersonic flow around a blunt body configuration. The implemented schemes are first-order accurate in space. The time integration uses a Runge-Kutta method of five stages and is second order accurate. The algorithm is accelerated to the steady state solution using a spatially variable time step. This technique has proved excellent gains in terms of convergence rate as reported in [20-21].

The results have demonstrated that the [2] scheme using the [19] turbulence model has yielded the best value of the stagnation pressure at the blunt body’s nose and is the best choice for this study. The most efficient scheme has been the [3] one using the [11] turbulence model.

Acknowledgment

The author thanks the CAPES by the financial support conceded under the form of a scholarship. He also acknowledges the infrastructure of ITA that allowed the realization of this work.

References


