Assessment of Several Turbulence Models as Applied to Supersonic Flows in 2D – Part II

Edisson S. G. Maciel

Aeronautical Engineering Division (IEA), Aeronautical Technological Institute (ITA), São José dos Campos, SP, Brazil

Email address
edisavio@edissonsavio.eng.br

Abstract

In the present work, the Van Leer flux vector splitting scheme is implemented to solve the two-dimensional Favre-averaged Navier-Stokes equations. The Granville algebraic model and the Wilcox and Rubesin, Coakley, and Wilcox two-equation models are used in order to close the problem. The physical problem under study is the supersonic flow around a simplified version of the VLS (Brazilian “Satellite Launcher Vehice”) configuration. The results have demonstrated that the aerodynamic coefficients are better predicted by the Wilcox and Rubesin turbulence model. The stagnation pressure ahead of the VLS configuration is better predicted by the Wilcox turbulence model.

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 in [4]. The interested reader is encouraged to read this reference to become aware of the present study.

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.

[10] have performed a study involving three different turbulence models. In this paper, the Navier-Stokes equations were solved applied to the supersonic flow around a simplified configuration of the Brazilian Satellite Launcher, VLS. The algebraic models of [11] and of [6] and the one-equation model of [12] were used to close the problem. The algorithms of [13] and of [3] were compared and presented good results.

In terms of two-equation models, [14] have presented a work that deals with such models applied to the solution of supersonic aerospace flow problems. The two-dimensional Navier-Stokes equations written in conservative form, employing a finite volume formulation and a structured spatial discretization were solved. The [2] algorithm, first order accurate in space, was used to perform the numerical experiments. Turbulence
was taken into account using two k-ε turbulence models, namely: the [15-16] models. The steady state supersonic flow around a simplified version of the Brazilian Satellite Launcher, VLS configuration was studied. The results show that the pressure field generated by the [16] model was stronger than the respective one obtained with the [15] model, although the latter predicts more accurate aerodynamic coefficients in this problem. The [16] model predicted less intense turbulence kinetic energy- and dissipation-rate profiles than the [15] model, yielding less intense turbulence fields.

In the present work, the [2] flux vector splitting scheme is implemented, on a finite-volume context. The two-dimensional Favre-averaged Navier-Stokes equations are solved using an upwind discretization on a structured mesh. The [17] algebraic model and the [18-20] two-equation models are used in order to close the problem. The physical problem under study is the supersonic flow around a simplified version of the Brazilian Satellite Launcher, namely: the [15-16] models. The steady state supersonic flow around a simplified version of the Brazilian Satellite Launcher, VLS, configuration was studied. The results have shown that the pressure field generated by the [16] model was stronger than the respective one obtained with the [15] model, although the latter predicts more accurate aerodynamic coefficients in this problem. The [16] model predicted less intense turbulence kinetic energy- and dissipation-rate profiles than the [15] model, yielding less intense turbulence fields.

The main contribution of this paper is to apply the [2] flux vector splitting scheme, on a finite-volume context. The two-dimensional Favre-averaged Navier-Stokes equations are solved using an upwind discretization on a structured mesh. The [17] algebraic model and the [18-20] two-equation models are used in order to close the problem. The physical problem under study is the supersonic flow around a simplified version of the Brazilian Satellite Launcher, VLS configuration. The implemented scheme uses a MUSCL procedure to reach second order accuracy 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 [21-22].

The results have demonstrated that the [20] model has yielded critical pressure fields, but less intense than the one of the [19] model. The aerodynamic coefficients are better predicted by the [18] turbulence model; however, the [20] model provides the second best estimation. The stagnation pressure ahead of the VLS configuration is better predicted by the [20] turbulence model. Hence, the best choice corresponds to the [20] turbulence model for this study.

The 2-D flow is modeled by the Navier-Stokes equations, implemented scheme uses a MUSCL procedure to reach second order accuracy 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 [21-22].

The main contribution of this paper is to apply the [2] flux vector splitting scheme to calculate turbulence in three-dimensions. It applies the [17-20] turbulence models, one algebraic and three two-equation models, to a three-dimensional problem. As the two-equation models are difficult to be implemented, due to the difficult in putting the equations on a formal formulation and in the difficult of implementing the boundary conditions, mainly in three-dimensions, this work assumes a character of originality and of complex implementation, which is simplified by the present description.

2. Navier-Stokes Equations

The 2-D 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] algorithm is applied only to the original four conservation equations. The integral form of these equations may be represented by:

\[ \frac{\partial}{\partial t} \int_V Q \, dv + \int_S \left[ (E_x - E_y) n_x + (F_x - F_y) n_y \right] dS + \int_S G \, dv = 0, \quad (1) \]

where Q is written for a Cartesian system, V is the cell volume, \( n_x \) and \( n_y \) are components of the unity vector normal to the cell boundary, S is the flux area, \( E_x \) and \( F_x \) are the components of the convective, or Euler, flux vector, \( E_y \) and \( F_y \) are the components of the viscous, or diffusive, flux vector and G is the source term of the two-equation models. The vectors Q, \( E_x \), \( E_y \), and \( F_x \) are, incorporating a k-\( \omega \) or k-\( \omega^2 \) formulation, represented by:

\[
\begin{bmatrix}
0 \\
t_{xx} + t_{xy} \\
t_{yy} + t_{yx} \\
f_x \\
f_y \\
\alpha_x \\
\alpha_y \\
\beta_x \\
\beta_y \\
G_x \\
G_y
\end{bmatrix}, \quad \text{and} \quad G = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}, \quad (2)
\]

where the components of the viscous stress tensor are defined as:

\[
t_{xx} = \left[ 2 \mu_{vis} \frac{\partial u}{\partial x} - 2/3 \mu_{vis} \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) \right]/Re; \\
t_{yy} = \mu_{vis} \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)/Re; \\
t_{xy} = \left[ 2 \mu_{vis} \frac{\partial v}{\partial y} - 2/3 \mu_{vis} \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) \right]/Re. \quad (3)
\]

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

\[
f_x = \left( t_{xx} + t_{xy} \right) u + \left( t_{xy} + t_{yy} \right) v - q_x; \quad (5)
\]

\[
f_y = \left( t_{yx} + t_{yy} \right) u + \left( t_{yy} + t_{yx} \right) v - q_y, \quad (6)
\]

where \( q_x \) and \( q_y \) are the Fourier heat flux components and are
given by:
\[
q_x = -\frac{\gamma}{Re} \left( \frac{\mu_s}{Pr_l} + \frac{\mu_t}{Pr_T} \right) \frac{\partial e}{\partial x} ;
\]
\[
q_y = -\frac{\gamma}{Re} \left( \frac{\mu_s}{Pr_l} + \frac{\mu_t}{Pr_T} \right) \frac{\partial e}{\partial y} .
\]

The diffusion terms related to the k-ε equations are defined as:
\[
\alpha_k = \frac{1}{Re} \left( \frac{\mu_s}{Pr_l} + \frac{\mu_t}{Pr_T} \right) \frac{\partial k}{\partial x} ;
\]
\[
\alpha_\epsilon = \frac{1}{Re} \left( \frac{\mu_s}{Pr_l} + \frac{\mu_t}{Pr_T} \right) \frac{\partial \epsilon}{\partial y} ;
\]
\[
\beta_k = \frac{1}{Re} \left( \frac{\mu_s}{Pr_l} + \frac{\mu_t}{Pr_T} \right) \frac{\partial s}{\partial x} ;
\]
\[
\beta_\epsilon = \frac{1}{Re} \left( \frac{\mu_s}{Pr_l} + \frac{\mu_t}{Pr_T} \right) \frac{\partial s}{\partial y} .
\]

In the above equations, ρ is the fluid density; u and v are Cartesian components of the velocity vector in the x and y directions, respectively; e is the total energy per unit volume; p is the static pressure; k is the turbulence kinetic energy; s is heat flux components; G is the molecular and the turbulent viscosities, respectively; Pr is the laminar and the turbulent Prandtl numbers, respectively; η is the ratio of specific heats; Re is the laminar Reynolds number, σ is the Reynolds stress components; the q’s are the Fourier fluid, ε is the total energy per unit volume; G is the molecular viscosity estimated by the empiric Sutherland formula:
\[
\mu_M = b T^{1/2} / (1 + S / T),
\]
where T is the absolute temperature (K), b = 1.458x10⁻⁶.

The problem of the turbulent simulation is in the calculation of the Reynolds stress. Expressions involving velocity fluctuations, originating from the average 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 [17] model, the boundary layer is divided in internal and external.

Initially, the \( \nu_\omega \) kinematic viscosity at wall and the \( \tau_{xy,w} \) shear stress at wall are calculated. After that, the \( \delta \) boundary layer thickness is calculated. So, the (N) normal distance from the wall to the studied cell is calculated. The N* term is obtained from:
\[
N^* = \sqrt{Re} \left( \frac{\tau_{xy,w}}{\rho_w} \right)^{1/2} \left( \frac{\nu_{xy}}{\nu_w} \right)^{1/2} N^*,
\]
where \( \rho_w \) is the wall density. The Van Driest damping factor is calculated by:
\[
D = 1 - e^{-\left( N^*/A_G \right)},
\]
where:
\[
A_G = 26 \left( \sqrt{1 + b + |b|} \right)^2
\]
and
\[
p^+ = \frac{1}{Re} \left( \frac{\nu_{xy}}{\tau_{xy,w}} \right)^{1/2} \frac{\partial p}{\partial x} .
\]

The ratio of tangential stress is given by:
\[
\frac{\tau_{xy}}{\tau_{xy,w}} = 1 + \frac{\eta^3}{\eta^3} (3 + 2 \xi) \eta^3 + (2 + \xi) \eta^3.
\]

The characteristic length is defined by
\[
I_{\text{mix}} = \sqrt{\frac{\tau_{xy}}{\tau_{xy,w}}} D. \quad (20)
\]

Hence, for the internal layer, one has:
\[
\mu_{Ti} = \text{Re} \rho C_{\text{mix}} F_{\text{wake}} F_{\text{Kleb}} (N; N_{\text{max}} / C_{\text{Kleb}}), \quad (21)
\]

In the external layer,
\[
\mu_{Te} = \text{Re} \rho \alpha C_{\text{cp}} F_{\text{wake}} F_{\text{Kleb}} (N; N_{\text{max}} / C_{\text{Kleb}}), \quad (22)
\]

with:
\[
F_{\text{wake}} = \text{MIN} \left[ N_{\text{max}} F_{\text{max}}; C_{\text{wk}} N_{\text{max}} U_{\text{diff}} / F_{\text{max}} \right], \quad (23)
\]

\[
C_{\text{Kleb}} = 2/3 - 0.01312 / (0.1724 + \beta); \quad (24)
\]

\[
\beta = -\frac{1}{\sqrt{\tau_{xy,w} / \rho_w}} \frac{1}{dx \text{ Re}}; \quad (25)
\]

\[
C_{\text{cp}} = (3 - 4C_{\text{Kleb}}) \left[ \frac{1}{2C_{\text{Kleb}}(2 - 3C_{\text{Kleb}})} \right]. \quad (26)
\]

Hence, \( N_{\text{max}} \) is the value of \( N \) where \( I_{\text{mix}} \| \| \| D \) reached its maximum value and \( I_{\text{mix}} \) is the Prandtl mixture length. The constant values are: \( \kappa = 0.4 \); \( \alpha = 0.0168 \); and \( C_{\text{wk}} = 0.25 \).

\( F_{\text{Kleb}} \) is the intermittent function of Klebanoff given by:
\[
F_{\text{Kleb}} (N) = \left[ t + 5.5(C_{\text{Kleb}} N / N_{\text{max}})^6 \right]^{-1}, \quad (27)
\]

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

\[
U_{\text{diff}} = \left( \sqrt{u^2 + v^2} \right)_{\text{max}} - \left( \sqrt{u^2 + v^2} \right)_{N = N_{\text{max}}}, \quad (28)
\]

Finally, the turbulent viscosity is chosen from the internal and the external viscosities:
\[
\mu_T = \text{MIN} (\mu_{Ti}, \mu_{Te}). \quad (29)
\]

\subsection*{3.2. Wilcox and Rubesin Turbulence Model}

In the [18] turbulence model, \( s = \omega^2 \). To define the turbulent viscosity, or eddy viscosity, it is necessary to define the turbulent Reynolds number:
\[
Re_T = k / (\nu_M \omega), \quad \text{with: } \nu_M = \mu_M / \rho. \quad (30)
\]

It is also necessary to determine the D damping factor:
\[
D = 1 - \alpha e^{1 - Re_T}. \quad (31)
\]

The turbulent viscosity is expressed in terms of \( k \) and \( \omega^2 \) as:
\[
\mu_T = \text{Re} D \omega k / \omega. \quad (32)
\]

The source term denoted by G in the governing equation contains the production and dissipation terms of \( k \) and \( \omega^2 \). To the [18] model, the \( G_k \) and \( G_{\omega^2} \) terms have the following expressions:
\[
G_k = -P_k - D_k \quad \text{and} \quad G_{\omega^2} = -P_{\omega^2} - D_{\omega^2}, \quad (33)
\]

where:
\[
P = \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \frac{\partial u}{\partial x}; \quad (34)
\]

\[
P_k = \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \frac{\partial u}{\partial x}; \quad (35)
\]

\[
P_{\omega^2} = \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \frac{\partial u}{\partial x}; \quad (36)
\]

with the second damping factor \( E \) defined as:
\[
E = 1 - \alpha e^{(-0.5Re_T)}. \quad (37)
\]

The closure coefficients adopted to the [18] model assume the following values: \( \sigma_k = 2.0 \); \( \sigma_{\omega^2} = 2.0 \); \( \beta = 0.09 \); \( \beta = 0.15 \); \( \alpha = 0.99174 \); \( \gamma_{\omega} = 0.9 \); \( \text{Pr}_{\text{kd}} = 0.72 \); \( \text{Pr}_{\text{ct}} = 0.9 \).

\subsection*{3.3. Coakley Turbulence Model}

The [19] model is a \( k^{1/2} - \omega \) one. The turbulent Reynolds number is defined as:
\[
R = \sqrt{k N / \nu_M}. \quad (38)
\]

The production term of turbulent kinetic energy is given by
\[
P = \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \frac{\partial u}{\partial y} / \text{Re}. \quad (39)
\]
The function $\chi$ is defined as

$$\chi = \frac{C_1 B}{\omega} - 1. \quad (39)$$

The damping function is given by

$$D = \frac{1 - e^{-\alpha R}}{1 + \beta \chi}. \quad (40)$$

The turbulent viscosity is defined by

$$\mu_T = \frac{0.5 C_1 DP}{\omega} \rho \sqrt{k/Re}; \quad D_k = 0.5 \left[ \frac{2}{3} \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) \right] \rho \sqrt{k/Re};$$

$$\mu_T = \left( \frac{C_1 C_2 P}{\omega^2} \right) \rho \sqrt{\omega/Re}; \quad D_\omega = \left[ \frac{2}{3} C_1 \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) \right] \rho \left( \frac{\omega}{C_2} \right) \sqrt{\omega/Re}, \quad (44)$$

where $C_1 = 0.405D + 0.045$. The closure coefficients adopted for the [19] model are: $\sigma_k = 1.0$; $\sigma_\omega = 1.3$; $C_1 = 0.09$; $C_2 = 0.92$; $\beta = 0.5$; $\alpha = 0.0065$; $Pr_d = 0.72$; $Pr_f = 0.9$.

### 3.4. Wilcox Turbulence Model

In the [20] turbulence model, $s = \omega$. The turbulent viscosity is expressed in terms of $k$ and $\omega$ as:

$$\mu_T = \frac{C_\mu D}{\omega} \frac{pk}{\omega}, \quad (45)$$

In this model, the quantities $\sigma_k$ and $\sigma_\omega$ have the values $1/\sigma^*$ and $1/\sigma$, respectively, where $\sigma^*$ and $\sigma$ are model constants.

To the [20] model, the $G_k$ and $G_\omega$ terms have the following expressions:

$$G_k = -P_k + D_k \quad \text{and} \quad G_\omega = -P_\omega + D_\omega, \quad (46)$$

where:

$$P_k = \mu_T \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \frac{\partial u}{\partial y} \frac{\partial u}{\partial y} / \text{Re} \quad \text{and} \quad D_k = \beta^* \frac{pk \omega^*/\text{Re}}{\text{Re}}; \quad (47)$$

$$P_\omega = \left( \frac{C_\omega}{k} \right) P_k \quad \text{and} \quad D_\omega = \beta \frac{P_k}{\omega^*/\text{Re}}, \quad (48)$$

where the closure coefficients adopted for the [20] model are: $\beta^* = 0.09$; $\beta = 3/40$; $\sigma^* = 0.5$; $\sigma = 0.5$; $\alpha = 5/9$; $Pr_d = 0.72$; $Pr_f = 0.9$.

### 4. Initial and Boundary Conditions

The initial and boundary conditions to the [17] turbulence model are the same for perfect gas formulation. Details of these conditions can be found in [24-25]. For the $k-\omega$ models, one has:

$$\mu_T = \frac{C_\mu Dpk}{\omega}, \quad (41)$$

with: $C_\mu$ a constant to be defined.

To the [19] model, the $G_k$ and $G_\omega$ terms have the following expressions:

$$G_k = -P_k - D_k \quad \text{and} \quad G_\omega = -P_\omega - D_\omega, \quad (42)$$

### 4.1. Initial Condition

Freestream values, at all grid cells, are adopted for all flow properties as initial condition, as suggested by [26-27]. Therefore, the vector of conserved variables is defined as:

$$Q_{ij} = \begin{cases} 1 & M_x \cos \alpha \quad M_y \sin \alpha \quad \frac{1}{\gamma (\gamma - 1)} + 0.5M_x^2 \quad t_u \quad s_u \end{cases}, \quad (49)$$

where $t_u$ is the freestream turbulent kinetic energy or the square root of this value and $s_u$ is the freestream turbulent vorticity or the squared of this value. These parameters assume the following values as using the [18] model: $t_u = 1.0 \times 10^{-6}$ and $s_u = \left( \frac{10 u_\infty}{v_{REF}} \right)^2$, with $u_\infty$ the freestream Cartesian component and $v_{REF}$ a characteristic length, the same adopted in the definition of the Reynolds number; the [19] model: $t_u = 1.0 \times 10^{-3}$ and $s_u = \left( \frac{10 u_\infty}{v_{REF}} \right)^2$; and the [20] model: $t_u = 1.0 \times 10^{-6}$ and $s_u = \left( \frac{10 u_\infty}{v_{REF}} \right)^2$.

### 4.2. 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 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:

\[ [18] \text{model: } k_{\text{ghost}} = 0.0 \quad \text{and} \quad \omega_{\text{ghost}} = \left(38/3 \sqrt{\nu M} \right) \left[ b \frac{d_n^2}{\gamma} \right]^\beta; \]

\[ [19] \text{model: } k_{\text{ghost}} = 0.0 \quad \text{and} \quad \omega_{\text{ghost}} = \left(38/3 \sqrt{\nu M} \right) \left[ b \frac{d_n^2}{\gamma} \right]^\beta; \]

\[ [20] \text{model: } k_{\text{ghost}} = 0.0 \quad \text{and} \quad \omega_{\text{ghost}} = \left(38/3 \sqrt{\nu M} \right) \left[ b \frac{d_n^2}{\gamma} \right]^\beta, \]

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: Five properties are specified and one extrapolated. This approach is based on information propagation analysis along characteristic directions in the calculation domain ([27]). In other words, for subsonic flow, five characteristic propagate information point into the computational domain. Thus five 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: Five characteristic propagate information outward the computational domain. Hence, the associated variables should be extrapolated from interior information. The characteristic direction associated to the “\( q_{\text{normal}} = 0 \)” velocity should be specified because it point inward to the computational domain ([27]). 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 \( k_{ff} = 0.01K \), or 1% of \( K \). The turbulence vorticity is determined by its freestream value.

5. Results

Tests were performed in a Dual Core processor of 2.3GHz and 2.0Gbytes of RAM microcomputer on a Windows 7.0 environment. The present author uses the Microsoft Developer Studio for FORTRAN 90 to implement the numerical algorithm and turbulence models. 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. The ratio of specific heats, \( \gamma \), assumed the value 1.4.

Figures 1 shows the entire VLS viscous mesh, whereas Fig. 2 shows the detail of the VLS close to the satellite compartment. A mesh of 253x70 points or composed of 17,388 rectangular cells and 17,710 nodes was generated, employing an exponential stretching of 5% in the \( \eta \) direction.
The initial data of the simulations is described in Tab. 1.

<table>
<thead>
<tr>
<th>$M_\infty$</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>40,000m</td>
<td>3.76m</td>
<td>$8.93\times10^5$</td>
</tr>
</tbody>
</table>

### 5.1. Granville Results

Figure 3 exhibit the pressure contours obtained by the second order [2] scheme. The contours are uniform and well defined. The normal shock wave at the blunt body nose is well captured. Figure 4 exhibits the Mach number contours obtained by the high resolution TVD [2] scheme. The viscous region close to the VLS walls is well captured; in other words, the heat conduction, through the Fourier law, is well captured by the turbulence model. The normal shock weave is well captured. The solution is free of pre-shock oscillations.

Figure 5 presents the translational/rotational temperature contours obtained by the [2] scheme. Temperatures above 864.8 K are obtained. The region downstream the satellite compartment appears with regions of discrete high dissipation, as can be noted in the figure. In other words, this means that circulation bubbles are formed as consequence of boundary layer displacement. These can be seen in Figs. 6 and 7, which highlight the circulation bubble formations downstream the satellite compartment and upstream the booster region. These regions are very discrete, but even so the numerical scheme was able to capture such phenomenon.
Figure 8. $-C_p$ distribution (G).

Figure 8 shows the $-C_p$ distribution along the blunt body wall. The $-C_p$ suffers a rapid increase at the satellite compartment and downstream it is variable. At the booster region, a rapid decrease in the $-C_p$ values with a recovery pressure at the ramp is observed. In all this distribution, no overshoots and undershoots are perceptible, even for a second-order scheme. This aspect highlights the MUSCL procedure as a good tool to provide clean profiles. Such procedure avoids the appearance of Gibbs phenomenon, typical of second order schemes, yielding good quality solutions.

5.2. Wilcox and Rubesin Results

Figure 9. Pressure contours (WR).

Figure 9 shows the pressure contours obtained by the [2] scheme as using the [18] turbulence model. The shock is well defined and homogeneous. As can be seen, there are qualitative differences between this plot and the [17] corresponding plot. The shock at the booster region is captured in the [17] scheme and is not captured in the [18] solution.

Figure 10. Mach number contours (WR).

Figure 10 exhibits the Mach number contours obtained by the [2] numerical scheme as using the [18] turbulence model. The present contours are smoother than the corresponding [17] solution. The normal shock at the blunt nose is well captured by the scheme. The [17] solution is less dissipative than the [18] solution.

Figure 11. Temperature contours (WR).

Figure 11 presents the temperature field obtained by the [2] scheme as using the [18] turbulence model. Qualitatively, this plot has differences in relation to the [17] plot. The [17] solution.

Figure 12. Circulation bubble formation (Cockpit-WR).
solution is less dissipative. Moreover, the regions of high dissipation are concentrated at the end satellite compartment and at the beginning booster region. Temperatures close to 731.6 K are reached. Figures 12 and 13 show details of the cockpit and booster regions of the VLS. Both regions present small separation regions at the corner (cockpit) and at the ramp (boosters), causing the formation of separation bubbles. The region of circulation bubble formation at the boosters is bigger than this one at the satellite compartment region.

Figure 12. Circulation bubble formation (Boosters-WR).

Figure 13. Circulation bubble formation (Boosters-WR).

Figure 14. –Cp distribution (WR).

Figure 15. Pressure contours (C).

Figure 16. Mach number contours (C).

Figure 17. Temperature contours (C).

Figure 14 presents the –Cp distribution at wall of the VLS configuration, generated by the [18] turbulence model. This curve presents a reduction of –Cp close to the booster region and after that the pressure coefficient is recovered at the booster end. The –Cp distribution along the VLS central body is relative smooth, without oscillations in the solution.

5.3. Coakley Results

Figure 15 shows the pressure contours generated by the [2] scheme as using the [19] turbulence model. This curve is very similar to the [17] and [18] models. The shock is well captured and the solution is homogeneous, without pre-shock oscillations.

Figure 16 exhibits the Mach number contours obtained as using the [19] turbulence model. The solution is very close to the [17] solution, indicating the good performance of this algebraic turbulence model. It is possible to note that this solution present less dissipation, as in the [17] solution. Regions of discrete formation of separation bubbles are perceptible at the downstream region of the satellite
compartment and at the booster region. It is possible to see in Figs. 18 and 19 that circulation bubbles are well formed.

Figure 17 presents the translational temperature contours originated by the [2] scheme as using the [19] turbulence model. Temperatures near 822.3 K are observed in the field, but smaller than the [17] result. Regions of high temperature are observed at the blunt body nose and at the satellite compartment end.

![Figure 18. Circulation bubble formation (Cockpit-C).](image)

![Figure 19. Circulation bubble formation (Boosters-C).](image)

Figures 18 and 19 corroborate what was observed in the aforementioned paragraph. Circulation bubbles formation is originated at regions of high heating and generate loss of energy by the bubbles displacement and energy exchange due to collisions.

Figure 20 exhibits the –Cp distribution originated by the [19] turbulence model. The cockpit upstream region presents a pressure distribution in steps and also presents the reduction of pressure close to the boosters region, with the consequent increase of such pressure, obtained in all other solutions. The –Cp profile of [19] is more strength than the –Cp profile of the [17] and [18] turbulence models. However, it is equivalent in qualitative terms in relation to the [18] turbulence model.

![Figure 20. –Cp distribution (C).](image)

5.4. Wilcox Results

Figure 21 shows the pressure contours obtained by the [2] algorithm as the [20] turbulence model is employed. The curves of contours are well defined and the solution quality is the same as in the [17] and [19] solutions. The shock at the booster region is well captured. No overshoots or undershoots
are present, corroborating the idea of this scheme prevents Gibbs phenomenon.

Figure 22 exhibits the Mach number contours obtained by the [20] turbulence model. The contours present more dissipative features than the [17] contours. No pre-shock oscillations are perceptible. The subsonic region is formed at the blunt nose as expected.

Figure 23 presents the translational temperature contours originated from the [20] turbulence model. The temperature peak is observed near to 835.2 K, which is less than the [17] captured field and more than the [18-19] capture fields. It is not possible to detect regions of great heating near the satellite compartment end or at the booster regions. Opposed to this observation, circulation bubble formations were detected in these regions. Figures 24 and 25 show these regions. They are discrete, but observable, and ratifies the expected behavior.

Figure 26 shows the –Cp distribution obtained with the [20] turbulence model. The step profile at the blunt nose region and the hole region at the booster region are again observable. The pressure recovery at the booster region is typical of all solutions in this study. This –Cp profile is not so strength than the [19] profile.

As main conclusion of this four turbulence models, the [17] model is the less dissipative in qualitative terms. However, the [19] turbulence model gives more severe –Cp conditions than the others.

5.5. Quantitative Analysis

Table 2 shows the lift and drag aerodynamic coefficients calculated by the [2] scheme in the turbulent cases. As the geometry is symmetrical and an attack angle of zero value was adopted in the simulations, the lift coefficient should have a zero value. The most correct value to the lift coefficient is due to the [18] turbulence model.

Another possibility to quantitative comparison of the laminar and turbulent cases is the determination of the stagnation pressure ahead of the configuration. [28] 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 VLS 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 [28]. So it is possible to determine the ratio $p_{\infty}/p_\infty$ from [28], where $p_0$ is the stagnation pressure in front of the configuration and $p_{\infty}$ is the freestream pressure (equals to $1/\gamma$ to the present dimensionless).

Hence, to this problem, $M_\infty = 3.0$ corresponds to $p_0/p_{\infty} = 12.06$ and remembering that $p_{\infty} = 0.714$, it is possible to conclude that $p_0 = 8.61$. Values of the stagnation pressure to the turbulent cases and respective percentage errors are shown in Tab. 3. They are obtained from Figures 3, 9, 15, and 21. As can be observed, the [17] and [20] turbulence models present the best result, with a percentage error of 4.70%.

Table 3. Values of the stagnation pressure and respective percentage errors.

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>$p_{\infty}$</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>[17] turbulence model</td>
<td>8.20</td>
<td>4.70</td>
</tr>
<tr>
<td>[18] turbulence model</td>
<td>8.10</td>
<td>5.80</td>
</tr>
<tr>
<td>[19] turbulence model</td>
<td>8.10</td>
<td>5.80</td>
</tr>
<tr>
<td>[20] turbulence model</td>
<td>8.20</td>
<td>4.70</td>
</tr>
</tbody>
</table>

Finally, Table 4 exhibits the computational data of the present simulations. It can be noted that the most efficient is the [20] turbulence model.

As a final conclusion of this study, the [20] turbulence model was the best when comparing these four turbulence models: [17-20]. This choice is based on the second best estimative of the aerodynamic coefficients and the best estimative to the stagnation pressure. In a next paper, the present author will study 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 find the best one.

Table 4. Computational data.

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>CFL</th>
<th>Iterations</th>
</tr>
</thead>
<tbody>
<tr>
<td>[17] turbulence model</td>
<td>0.05</td>
<td>9,315</td>
</tr>
<tr>
<td>[18] turbulence model</td>
<td>0.20</td>
<td>9,450</td>
</tr>
<tr>
<td>[19] turbulence model</td>
<td>0.10</td>
<td>5,500</td>
</tr>
<tr>
<td>[20] turbulence model</td>
<td>0.10</td>
<td>5,005</td>
</tr>
</tbody>
</table>

6. Conclusion

In the present work, the [2] flux vector splitting scheme is implemented, on a finite-volume context. The two-dimensional Favre-averaged Navier-Stokes equations are solved using an upwind discretization on a structured mesh. The [17] algebraic model and the [18-20] two-equation models are used in order to close the problem. The physical problem under study is the supersonic flow around a simplified version of the VLS configuration. The implemented scheme uses a MUSCL procedure to reach second order accuracy 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 [21-22].

The results have demonstrated that the [20] model has yielded critical pressure fields, but less intense than the one of the [19] model. The aerodynamic coefficients are better predicted by the [18] turbulence model; however, the [20] model provides the second best estimative. The stagnation pressure ahead of the VLS configuration is better predicted by the [20] turbulence model. Hence, the best choice corresponds to the [20] turbulence model for this study.

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 find the best one.

Acknowledgment

The authors would like to thank the infrastructure of ITA that allowed the realization of this work.

References


