Assessment of One- and Two-Equation Turbulence Models for Hypersonic Transitional Flows

Christopher J. Roy‡ and Frederick G. Blottner†
Sandia National Laboratories*
P. O. Box 5800
Albuquerque, NM 87185

Abstract

One- and two-equation turbulence models are examined for hypersonic perfect- and real-gas flows with laminar, transitional, and turbulent flow regions. These models were generally developed for incompressible flows, and the extension to the hypersonic flow regime is discussed. In particular, inconsistencies in the formulation of diffusion terms for one-equation models are examined. For the Spalart-Allmaras model, the standard method for forcing transition at a specified location is found to be inadequate for hypersonic flows. An alternative transition method is proposed and evaluated for a Mach 8 flat plate test case. This test case is also used to evaluate three different two-equation turbulence models: a low Reynolds number \(k - \varepsilon\) model, the Menter \(k - \omega\) formulation, and the Wilcox (1998) \(k - \omega\) model. These one- and two-equation models are then applied to the Mach 20 Reentry F flight vehicle. The Spalart-Allmaras model and both \(k - \omega\) for-


‡ Senior Member of Technical Staff, Thermal/Fluid Computational Engineering Sciences Department, Mail Stop 0835, E-mail: cjroy@sandia.gov, Member AIAA
† Distinguished Member of Technical Staff, Aerosciences and Compressible Fluid Mechanics Department, Mail Stop 0825, E-mail: fgblott@sandia.gov, AIAA Fellow
* Sandia is a multiprogram laboratory operated by Sandia Corporation, a Lockheed Martin Company, for the United States Department of Energy under Contract DE-AC04-94AL85000.

This paper is declared a work of the U. S. Government and is not subject to copyright protection in the United States.
DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, make any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.
DISCLAIMER

Portions of this document may be illegible in electronic image products. Images are produced from the best available original document.
mulations are found to provide reasonable agreement with the flight data for heat flux, while the Baldwin-Barth and low Reynolds number $k - \varepsilon$ models overpredict the turbulent heating rates by a factor of two. Careful attention is given to the verification of solution accuracy in the areas of both iterative and grid convergence.

**Nomenclature**

- $a$: speed of sound, $m/s$
- $D$: turbulence diffusion term
- $d$: distance to the wall, $m$
- $k$: specific turbulent kinetic energy, $m^2/s^2$
- $p$: pressure, $N/m^2$
- $q$: heat flux, $W/m^2$
- $R_N$: vehicle nose radius, $m$
- $R_T$: turbulence Reynolds number ($R_T = k^2/\varepsilon$)
- $r$: radial coordinate, $m$
- $S$: turbulence source term
- $S_D$: turbulence destruction source term
- $S_{ij}$: strain rate tensor, $1/s$ ($S_{ij} = \frac{1}{2}(\partial u_i/\partial x_j + \partial u_j/\partial x_i)$)
- $S_P$: turbulence production source term
- $S_t$: turbulence transition source term
- $T$: temperature, $K$
- $T_u$: freestream turbulence intensity, percent
- $t$: time, $s$
- $U$: conserved transport quantity
- $u_i$: velocity, $m/s$
- $V$: velocity magnitude, $m/s$
- $x$: axial coordinate, $m$
- $y$: wall normal direction, $m$
- $\alpha$: angle of attack, degrees
\( \gamma \)  
ratio of specific heats

\( \delta_{ij} \)  
Kronecker delta function \((\delta_{ij} = 1 \text{ when } i=j)\)

\( \varepsilon \)  
specific dissipation rate, \(m^2/s^3\)

\( \theta_{cone} \)  
cone half-angle, degrees

\( \mu \)  
absolute viscosity, \(N\cdot s/m^2\)

\( \nu \)  
kinematic viscosity, \(m^2/s\)

\( \dot{\nu} \)  
Spalart-Allmaras working variable, \(m^2/s\)

\( \rho \)  
density, \(kg/m^3\)

\( \tau_{ij} \)  
turbulent stress tensor, \(m^2/s^2\)

\( \varphi \)  
non-conserved transport quantity

\( \Omega_{ij} \)  
rotation tensor, \(1/s\), \(\Omega_{ij} = \frac{1}{2}(\partial \tilde{u}_i / \partial x_j - \partial \tilde{u}_j / \partial x_i)\)

\( \omega \)  
specific turbulent frequency, \(1/s\)

Subscripts

\( E \)  
extact value

\( eff \)  
effective value (turbulent + laminar)

\( i, j, k \)  
indices for tensor notation

\( RE \)  
Richardson Extrapolation value

\( ref \)  
reference value

\( T \)  
turbulent quantity

\( t \)  
transitional quantity

\( w \)  
wall value

\( \infty \)  
freestream value

Superscripts

\( + \)  
quantity in wall coordinates

\( \sim \)  
denotes Favre (density-weighted) averaging

\( - \)  
denotes Reynolds (time-based) averaging

\( ' \)  
denotes Favre fluctuating quantity
Introduction

This work is concerned with developing a capability to model high-speed compressible boundary layers with laminar, transitional, and turbulent flow regions. The approach uses one- and two-equation eddy viscosity models to predict the turbulent flow. The same governing equations are presently being used to predict the transitional flow region where the onset to turbulent flow is specified and assumed to be known. The prediction of where onset to turbulent flow occurs is a research area that depends on an analysis of the flow stability, understanding of the flow disturbances outside the boundary layer, and a capability to predict the boundary layer receptivity. The process of entraining disturbances into the boundary layer and producing perturbations that can be amplified is called "receptivity."

The modeling of compressible transitional and turbulent flows is still an active area of research. A discussion of the physical mechanisms for transition to turbulence in supersonic and hypersonic boundary layers is given by Masad and Abid.1 Singer2 presents a review of modeling procedures for the transitional flow region, while Wilcox3 gives a discussion of turbulence modeling for compressible flows. Huang et al.4 discuss modifications for the standard turbulence models which are required to reproduce the compressible law of the wall for high-speed boundary layers.

The experimental data utilized in the current work is from the Reentry F flight vehicle,5 a 5 degree half-angle cone with a small spherical nosetip and flight data available at a wide range of hypersonic Mach numbers. In recent years, this experimental data set has been reevaluated with modern computational codes and is documented in Refs. 6-8. Aerothermal predictions have also been presented in these papers. Most of these solutions are for axisymmetric flow with the vehicle at zero degree angle of attack, but full three-dimensional solutions have been obtained with the actual flight angle of attack of 0.14 degrees. While there are many details of this flight experiment...
that are not well defined, the overall heat transfer predictions are in reasonable agreement with the flight measurements.

The application of turbulence models to compressible flows is not always clear, as most models were originally developed for incompressible flows. Formulations for incompressible flow are not applicable to compressible flow because some variables (e.g., density, viscosity) have been assumed constant in the development. The turbulent transport equations are often written in substantial differential form, while conservation form is generally required in compressible Navier-Stokes codes. Problems with the formulation of the governing equations for compressible turbulence models in conservation form are discussed. For example, the form of the diffusion term in the Spalart-Allmaras\textsuperscript{9,10} model is rewritten and justification for the new form is given.

The SACCARA (Sandia Advanced Code for Compressible Aerothermodynamics Research and Analysis) code\textsuperscript{11-14} is used for the results presented in this paper. For one-equation turbulence models, the SACCARA code has options for both the Baldwin-Barth\textsuperscript{15} and Spalart-Allmaras eddy viscosity models. There is evidence that the use of the Baldwin-Barth model does not constitute a well-posed system of governing equations.\textsuperscript{16} For boundary layer and shear layer flows, the solutions do not appear to converge to a unique solution as the mesh is refined. Therefore, there is more interest in using the Spalart-Allmaras model, as it has proven to be numerically robust. Part of the present work is concerned with the evaluation of the Spalart-Allmaras model for high-speed flows and the simulation of the transition region with the Spalart-Allmaras model.

The SACCARA code also has options for three popular two-equation eddy viscosity turbulence models: a low Reynolds number $k - \varepsilon$ formulation and two $k - \omega$ models. The $k - \varepsilon$ model employs the low Reynolds number modification of Nagano and Hishida\textsuperscript{17} to allow integration to solid walls. The first $k - \omega$ formulation is the hybrid model of Menter\textsuperscript{18} which is a blending between
a \( k-\omega \) formulation (near solid walls) and a \( k-\varepsilon \) formulation (in shear layers and freestream flow). Menter proposed this hybrid model to take advantage of the accuracy of the \( k-\omega \) model for wall-bounded flows and the \( k-\varepsilon \) model for free shear layers. The final model is the Wilcox \( k-\omega \) model\(^3\) which was modified in 1998 to improve the predictive accuracy for shear flows. This model is referred to as the Wilcox (1998) model in the current work. The appropriate form of the two-equation eddy viscosity equations is important because the one-equation formulation can be developed from the two-equation transport relations. This approach may be used to determine the appropriate form of the transport equation for the one-equation models.

Two flow cases have been used to investigate the performance of the one- and two-equation eddy viscosity models. The first case is the flow over a flat plate at Mach 8 and an altitude of 15 km where the perfect gas model is appropriate. The skin friction along the flat plate is used to judge the accuracy of the predictions through comparisons with the accurate laminar and turbulent results of Van Driest.\(^{19,20}\) If the standard turbulence models (without modifications for transition) are employed over the whole domain, the transition location often depends on turbulent intensity in the freestream. This behavior is similar to the bypass transition problem. When a transition plane is specified in which the turbulent eddy viscosity is neglected upstream, the transition locations for the Spalart-Allmaras and low Reynolds number \( k-\varepsilon \) models still show sensitivity to the freestream turbulence quantities. The control of the transition location with the Spalart-Allmaras model has been investigated.

The second case investigated is the flow over the Reentry F flight vehicle at Mach 20 and at an altitude of 24.4 km (80,000 ft) where real gas effects are significant. The measured heat transfer along the vehicle is used to judge the accuracy of the model predictions. The transition location is specified to give a reasonable match of the wall heat flux with the flight data. The solutions have
been obtained on three meshes with the number of cells in each coordinate direction doubled for each mesh refinement. In addition, the solutions on each mesh are marched in time until the wall heat flux has obtained a steady-state value. The accuracy of the iterative solution relative to the steady-state solution has been estimated for each model. The various uncertainties and assumptions in the flight experiment and prediction are discussed. Real gas effects have been taken into account with the use of an equilibrium air model.

**Favre-Averaged Transport Equation for Turbulence Models**

The generic form of the turbulent transport equation in substantial derivative form\(^3\) is

\[
\bar{\rho} \frac{D\phi}{Dt} = D + S_P - S_D
\]

where

\[
D = D_1 - \bar{D}, \quad D_1 = \frac{\partial}{\partial x_j} \left( \mu_{eff} \frac{\partial \phi}{\partial x_j} \right).
\]

For example for a one-equation eddy viscosity model, the dependent variable \(\phi\) is the kinematic eddy viscosity \(\nu_T\) and the effective diffusion coefficient is \(\mu_{eff}\). In some models there are two parts to the diffusion term on the right-hand side of Eq. (1); \(D_1\) is the first part of the diffusion term which can be put in conservation form, and \(\overline{D}\) is the remaining part. When \(\overline{D}\) is included, it can take on several forms. The source term \(S = S_P - S_D\) has a production part \(S_P\) and a dissipation part \(S_D\). If the continuity equation is multiplied by \(\phi\) and added to Eq. (1), the resulting equation is the generic transport equation in conservation form:

\[
\frac{\partial U}{\partial t} + \frac{\partial}{\partial x_j} \left( \bar{\rho} \bar{u}_j \phi - \mu_{eff} \frac{\partial \phi}{\partial x_j} \right) = \bar{D} + S_P - S_D
\]
The dependent variable is $U = \bar{\rho} \varphi$. This development utilizes Favre (overtilde) and Reynolds (overbar) averaging. See Ref. 3 for notation and for details on the averaging procedures. For all results presented herein, a value of unity is assumed for the turbulent Prandtl number.

**One-Equation Turbulence Model**

There have been a number of one-equation turbulence models developed which use a transport equation to solve for the eddy viscosity directly. The present work is focused on the Spalart-Allmaras model and a brief description is presented below.

**Spalart-Allmaras Model**

The transport equation for determining the eddy viscosity with near-wall effects included has been developed by Spalart and Allmaras.\textsuperscript{9,10} The governing equation form is slightly different than Eq. (1) and is

$$\frac{\partial D}{\partial t} = D + S_p - S_D + S_t,$$

$$\varphi = \frac{\mu_t}{f_{v1}} = \bar{\rho} \varphi$$  \hspace{1cm} (3)

The dependent variable $\varphi = \hat{\nu} = v_T/f_{v1}$, where $f_{v1}$ is a damping function used in the near-wall region and mainly in the viscous sublayer. This function and the right hand side terms will be defined below. The continuity equation is multiplied by $\varphi$ and added to Eq. (3) which gives a transport equation in conservation form for the Spalart-Allmaras model in the form of Eq. (2)

$$\frac{\partial U}{\partial t} + \frac{\partial}{\partial x_j} \left\{ \bar{\rho} \bar{u}_j \varphi - \mu_{eff} \frac{\partial \varphi}{\partial x_j} \right\} = \bar{D} + S_p - S_D + S_t$$

$$U = \bar{\rho} \hat{\nu} = \bar{\rho} \varphi, \quad \mu_{eff} = \frac{\mu + \hat{\mu}}{\sigma}$$  \hspace{1cm} (4)
The right-hand side has contributions from a diffusion term as well as production, destruction, and trip terms. The four terms in the model are written as follows:

**Diffusion-Original Form**

\[
D = \frac{\partial}{\partial t} \left[ \frac{\mu_{\text{eff}}}{\rho} \frac{\partial \Phi}{\partial x_j} \right] + D_2 = D_1 + \overline{D}
\]

\[
\mu_{\text{eff}} = \frac{\mu + \mu}{\sigma} = \frac{\rho (\nu + \Phi)}{\sigma}, \quad \overline{D} = D_2 - D_3
\]

\[
D_2 = \frac{c_{b2} \tilde{D}}{\sigma} \left( \frac{\partial \phi}{\partial x_j} \right) \left( \phi \frac{\partial \phi}{\partial x_j} \right), \quad D_3 = \left( \frac{\mu_{\text{eff}}}{\rho} \right) \frac{\partial \rho}{\partial x_j} \frac{\partial \phi}{\partial x_j}
\]

**Diffusion-Modified for Compressible Flow**

\[
D = D_1 + D_2, \quad \overline{D} = D_2
\]

**Production**

\[
S_P = c_{b1} (1 - f_{t2}) \tilde{S} \tilde{\phi}, \quad \tilde{S} = \sqrt{2} \Omega_{ij} \Omega_{ij} + \frac{\Phi}{\kappa^2 d^2} f_{v2}
\]

**Destruction**

\[
S_D = \left( c_{w1} f_w - \frac{c_{b1}}{\kappa^2} f_{t2} \right) \phi \left( \frac{\partial \phi}{\partial d} \right)^2
\]

**Trip Term**

\[
S_t = f_{t1} \rho (\Delta U)^2
\]

The quantity \( \Delta U \) is simply the local velocity magnitude for fixed wall flows. In the formal transform of the transport equation into conservation form, the diffusion term includes a density gradient term \( (D_3 \text{ above}) \). This term is zero when the transport equation (Eq. (4)) is developed from the compressible form of the \( k - \varepsilon \) transport equations, and is shown above in the term **Diffu-**
Compression for Compressible Flow. This form of the diffusion term is used in the present work. Including this density gradient term has been found to cause stability problems for high-speed flows, while having negligible effect on the predictions. The model controls transition from laminar to turbulent flow with the use of the trip term. With this additional physics, the foregoing governing equation requires some additional terms and definitions for \( f_{v1} \) and \( f_{v2} \) which involves the coefficients \( c_{i1} \) to \( c_{i4} \). Except where noted, the standard values for the model constants and functions are used in the current work and are given by:

\[
\begin{align*}
 f_{v1} &= \frac{\chi^3}{\chi^3 + c_{v1}}, \\
 f_{v2} &= 1 - \frac{\chi}{1 + \chi f_{v1}}, \\
 f_w &= g \left( \frac{1 + c_w^6}{g^6 + c_w^6} \right)^{1/6} \\
 \chi &= \frac{\hat{v}}{v}, \\
 g &= r + c_w(r^6 - r), \\
 r &= \frac{\hat{v}}{S^2 \kappa d^2} \\
 c_{b1} &= 0.1355, \\
 c_{b2} &= 0.622, \\
 \kappa &= 0.41, \\
 c_{w1} &= c_{b1} \kappa^2 + \frac{1 + c_{b2}}{\sigma} \\
 c_{w2} &= 0.3, \\
 c_{w3} &= 2.0, \\
 c_{v1} &= 7.1 \\
 c_{i1} &= 1.0, \\
 c_{i2} &= 2.0, \\
 c_{i3} &= 1.2, \\
 c_{i4} &= 1.2
\end{align*}
\]

**Boundary Conditions for Spalart-Allmaras Model**

At the wall \( \mu_T = 0 \) or \( \hat{v} = 0 \). The freestream boundary condition for this model is the specification of the turbulent eddy viscosity \( \mu_T \). In the freestream there should be no production of the eddy viscosity, which requires that

\[
f_{i2} = c_{i3}e^{-c_{i4}\chi^2} > 1
\]

in order to turn off the production term in Eq. (6). The restriction on \( \chi \) is

\[
\chi = \frac{\hat{v}}{v} < \sqrt{\frac{\ln c_{i3}}{c_{i4}}} = 0.604
\]

The restriction on the freestream eddy viscosity becomes
\[
\frac{\mu_T}{\mu} < \chi f_{v1} = \chi / [1 + (c_{v1}/\chi)^3] = 3.713 \times 10^{-4}
\]  
(5)

The freestream eddy viscosity as suggested by Spalart-Allmaras is \( \tilde{\nu} < 1/2\nu \), which gives

\[
\frac{\mu_T}{\mu} < \tilde{\nu} f_{v1}/\nu = 1.746 \times 10^{-4}.
\]

Control of Laminar and Turbulent Flow with the Spalart-Allmaras Model

The governing equation has three terms that are influenced by the transition model. The complete source term (for the conservative formulation) is

\[
S = \frac{c_{b2}}{Pr} \left( \frac{\partial \bar{\nu}}{\partial x_j} \right)^2 + c_{b1} [1 - f_{t2}] \bar{S} U
\]

\[
-(c_w f_w - c_{b1} f_{t2}/\kappa^2) \bar{d} \frac{\partial \bar{\nu}}{\partial d} + f_{t1} \bar{d}(\Delta U)^2
\]

where the trip terms are underlined in the above equation. The first term is part of the diffusion term and is included in the source term as it is evaluated numerically in an explicit manner. The second term is the production term and it will produce or increase the eddy viscosity if \( f_{t2} < 1 \). The third term is the destruction term and it will decrease the eddy viscosity if \( c_w f_w > c_{b1} f_{t2}/\kappa^2 \). The fourth term is the trip term and it will increase the eddy viscosity as \( f_{t1} > 0 \).

The model generally predicts turbulent flow everywhere when the trip terms are zero

\[
f_{t1} = 0 \text{ or } c_{t1} = 0, \quad f_{t2} = 0 \text{ or } c_{t3} = 0
\]

The flow can be made laminar everywhere with the following values of the trip terms:

\[
f_{t1} = 0 \text{ and } f_{t2} \geq 1.0 \text{ or } f_{t2} = c_{t3} \Lambda, \quad \Lambda = e^{-c_{t1} \kappa^2}
\]

or \( f_{t1} = 0, \quad f_{t2} = 0, \quad c_{b1} = 0 \)

Several different approaches have been investigated to control transition and to replace the origi-
nal trip model approach of Spalart-Allmaras.

**Method 1 \((f_{t2} \text{ is modified})\)**

In this approach the value of \(f_{t1} = 0\), and \(f_{t2} = c_{t3}(1 - \lambda)\) where \(\lambda\) varies from zero in the laminar flow region to one in the turbulent flow region. The parameter is increased smoothly and defines the transitional flow region. This method requires specification of the location and length of the transitional region.

**Method 2 \((c_{b1} \text{ is modified})\)**

In this method the trip terms \(f_{t1}\) and \(f_{t2}\) are set to zero. The coefficient \(c_{b1}\) is modified from the laminar flow region to the turbulent flow region as follows:

\[
x < x_s: \quad c_{b1} = 0, \quad x > x_e: \quad c_{b1} = 0.1355
\]

\[
x_s \leq x \leq x_e: \quad c_{b1} = 0.1355\lambda^p, \quad \lambda = (x - x_s)/(x_e - x_s)
\]

In this method, the location of the start of transitional flow \(x_s\) and end of transitional flow \(x_e\) are specified.

**Method 3 \((c_{b1}(1 - f_{t2}) \text{ is modified})\)**

In this method the production term coefficient is modified by writing this term as \(\alpha c_{b1}(1 - f_{t2})\). The parameter \(\alpha\) increases from zero to one in the transitional flow region. From the definition of \(f_{t2}\), the following is obtained:

\[
\chi = \frac{\hat{\nu}}{\nu} = \sqrt{-\ln[f_{t2}/c_{t3}]/c_{t4}} \quad (7)
\]

The production term switches sign when \(f_{t2} = 1\) which gives a critical value of \(\chi\) which is \(\chi^* = 0.604\). When \(x < x_t\), set \(\alpha = 0\) and there is no production of eddy viscosity upstream of
the transition location \(x_t\). When \(x > x_t\), \(\alpha\) is increased downstream towards one. This increase is controlled by setting

\[
\alpha = 1 - f_{t2} = 1 - c_t e^{-c_x x^2}
\]

When \(x > x_t\) and \(\chi \leq \chi^*\), then \(\chi = \chi^*\). When \(x > x_t\) and \(\chi > \chi^*\), then \(\chi\) is obtained from Eq. (7). In this method, only the single parameter \(x_t\) must be specified.

**Two-Equation Turbulence Models**

The standard method for specifying transition to turbulence is through analogy with the turbulence intermittency approach. The turbulence transport equations are solved over the entire domain, with a user-defined transition plane specified. Upstream of this plane, the effective viscosity is simply set to the laminar value, while downstream the effective viscosity is the sum of the laminar and turbulent viscosities.

**High Turbulent Reynolds Number \(k - \varepsilon\) Model**

The high Reynolds number formulation\(^3\) is appropriate for turbulent flows but is not appropriate in the near-wall region. It can be applied in the outer part of boundary layers and combined with an inner boundary layer approach near the wall to obtain a complete formulation. For the standard \(k - \varepsilon\) model, the turbulent kinetic energy equation for a compressible fluid can be rearranged into the form of Eq. (2) where the variables have the following values:

\[
\begin{align*}
U &= \bar{\rho}k = \bar{\rho}\Phi, \quad \mu_{eff} = \mu_k, \quad \bar{D} = 0 \\
S_{pk} &= \bar{\rho}P, \quad S_{Dk} = \bar{\rho}\varepsilon
\end{align*}
\]

The standard form of the production term \(P\) for compressible flows\(^3\) is
where

\[ \tau_{ij} = -u_i u_j^" = 2v \left( S_{ij} - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} k \delta_{ij} \]  

However, for the k-\( \varepsilon \) model, the compressible production term is approximated by the incompressible contribution only, i.e.

\[ P = \sqrt{\frac{1}{T} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j}} \]  

The effective viscosities are:

\[ \mu_k = \mu + \mu_T / \sigma_k, \quad \mu_\varepsilon = \mu + \mu_T / \sigma_\varepsilon \]  

where \( \mu_T = c_\mu f_\mu \bar{p} k^2 / \varepsilon \) (12)

The transport equation for dissipation of turbulent kinetic energy can also be put in the form of Eq. (2), where the variables have the following values:

\[ U = \bar{p} \varepsilon = \bar{p} \varphi, \quad \mu_{eff} = \mu_\varepsilon \]  

\[ \bar{D} = 0, \quad S_{P\varepsilon} = c_{\varepsilon 1} f_1 \bar{p}^\varepsilon_k P, \quad S_{D\varepsilon} = c_{\varepsilon 2} f_2 \bar{p}^\varepsilon^2_k \]  

The constants in the foregoing equations use the standard values:

\[ c_{\varepsilon 1} = 1.44, \quad c_{\varepsilon 2} = 1.92 \]
\[ c_\mu = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3 \]

Low Turbulent Reynolds Number k – \( \varepsilon \) Model
The Nagano and Hishida model\textsuperscript{17} was developed for incompressible flow and is included in the current formulation. The model uses the following damping function in the eddy viscosity relation given in Eq. (12):

\[ f_\mu = [1 - \exp(-y^+ / 26.5)]^2 \]

The source term for the turbulent kinetic energy Eq. (8) is

\[ S = \bar{\rho} P - \bar{\rho} \epsilon + \bar{D} \]

Again, the production term has been approximated with by the incompressible form given in Eq. (11). The source term for the dissipation rate equation is

\[ S = \frac{\epsilon}{k} (c_{\epsilon 1} f_1 \bar{\rho} P - c_{\epsilon 2} f_2 \bar{\rho} \epsilon) + \bar{E} \]

The parameters in these source terms are

\[ f_1 = 1, \quad f_2 = 1 - 0.3e^{-R_T^2}, \quad R_T = k^2 / \nu \epsilon \]

\[ \bar{D} = -2\mu \left( \frac{\partial \sqrt{K}}{\partial y} \right)^2, \quad \bar{E} = \mu \nu_T (1 - f_\mu) \left( \frac{\partial^2 \bar{u}_t}{\partial y^2} \right)^2 \]

where the variables $\bar{D}$ and $\bar{E}$ use boundary layer type derivatives normal to the wall, and $\bar{E}$ requires the tangential velocity component $\bar{u}_t$.

The Nagano-Hishida and the Launder-Sharma\textsuperscript{22} low Reynolds $k – \epsilon$ turbulence models have been used by Theodoridis, Prinos, and Goulas\textsuperscript{23} to predict transitional flow. They have investigated a flat plate flow experiment where the freestream turbulent intensity was approximately 3 percent and 6 percent. The two turbulence models were used to model the laminar to turbulent bypass transition in which the freestream turbulence determines where the transition to turbulent flow occurs. For this bypass transition case, the Nagano-Hishida model predicts transition to turbulent flow too near the leading edge while the Launder-Sharma model predictions are in reason-
able agreement with the experimental data. Of course, neither of these turbulence models were
developed to predict where transition will occur in a flow; the performance of the Launder-Shar-
ma model in predicting the location of transition is fortuitous. The failure of the Nagano-Hishida
for transitional flow requires caution in the application of this model and a procedure is required
to have the model turned on at the appropriate location.

**Menter k-ω Model**

Two different two-equation turbulence models are described which solve equations for the tur-
bulent kinetic energy, $k$, and the frequency of turbulent fluctuations, $\omega$. The Menter k-ω model\textsuperscript{18} is a hybrid model which uses a blending function to combine the best aspects of both the k-ω and
the k-ε turbulence models. Near solid walls, a k-ω formulation is used which allows integration
to the wall without any special damping or wall functions. Near the outer edge of the boundary
layer and in shear layers, the model blends into a transformed version of the k-ε formulation, thus
providing good predictions for free shear flows.

For the Menter k-ω model, the terms in the turbulent kinetic energy and turbulent frequency
equations take the form of Eq. (2):

\begin{align}
U &= \tilde{\rho}k = \tilde{\rho}\varphi, \quad \mu_{\text{eff}} = \mu_k, \quad \bar{D} = 0 \\
S_{Pk} &= \tilde{\rho}P, \quad S_{Dk} = \beta^*\tilde{\rho}k\omega \\
U &= \tilde{\rho}\varphi = \tilde{\rho}\varphi, \quad \mu_{\text{eff}} = \mu_\omega \\
\bar{D} &= 2\tilde{\rho}(1-F)\sigma_\omega^2\omega^2 \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \\
S_{P\omega} &= \tilde{\rho}\frac{q_T}{n} P, \quad S_{D\omega} = \beta\tilde{\rho}\omega^2
\end{align}

(14) (15)

The cross-diffusion term ($\bar{D}$) in Eq. (15) arises due to the transformation of the $\varepsilon$-equation into
an equation for $\omega$. The compressible form of the production term from Eq. (9) is employed. The effective viscosities are given by

$$\mu_k = \mu + \sigma_k \mu_T, \quad \mu_\omega = \mu + \sigma_\omega \mu_T$$

where $\mu_T = \bar{p} k / \omega$ (16)

and the model constants above are blended values of the $k-\omega$ and $k-\varepsilon$ parameters. For example, for the constant $\beta$,

$$\beta = F \beta_1 + (1 - F) \beta_2$$

where $F$ varies from unity at the wall to zero outside wall boundary layers, and a subscript “1” denotes $k-\omega$ constants and “2” denotes $k-\varepsilon$ constants. The values for these constants are:

$$\sigma_{k1} = \frac{1}{2}, \quad \sigma_{\omega1} = \frac{1}{2}, \quad \beta_1 = \frac{3}{40},$$

$$\beta^* = 0.09, \quad \kappa = 0.41, \quad \gamma_1 = \frac{\beta_1}{\beta^*} - \frac{\sigma_{\omega1} \kappa^2}{\sqrt{\beta^*}}$$

and

$$\sigma_{k2} = 1, \quad \sigma_{\omega2} = 0.856, \quad \beta_2 = 0.0828,$$

$$\beta^* = 0.09, \quad \kappa = 0.41, \quad \gamma_2 = \frac{\beta_2}{\beta^*} - \frac{\sigma_{\omega2} \kappa^2}{\sqrt{\beta^*}}$$

Wilcox (1998) k-\omega Model

For the Wilcox (1998) k-\omega model, the terms in Eq. (2) for the turbulent kinetic energy equation are

$$U = \bar{p} k = \bar{p} \varphi, \quad \mu_{\text{eff}} = \mu_k, \quad \bar{D} = 0$$

$$S_{pk} = \bar{p} P, \quad S_{Dk} = \beta^* \bar{p} k \omega$$

(17)
and for the turbulent frequency equation are

\[ U = \bar{\rho} \omega = \bar{\rho} \phi, \quad \mu_{\text{eff}} = \mu_{\omega}, \quad \overline{D} = 0 \]

\[ S_{P\omega} = \frac{\bar{\rho}}{\nu_T} \overline{P}, \quad S_{D\omega} = \beta \bar{\rho} \omega^2 \]  

(18)

where

\[ \alpha = \frac{13}{25}, \quad \beta = \beta_0 f_\beta, \quad \beta^* = \beta_0^* f_\beta^* \]

\[ \sigma_k = \sigma_\omega = \frac{1}{2}, \quad \beta_0^* = \frac{9}{100} \]

\[ f_\beta^* = \begin{cases} 
1, & \chi_k \leq 0 \\
1 + \frac{680 \chi_k^2}{1 + 400 \chi_k^2}, & \chi_k > 0
\end{cases} \quad \chi_k = \frac{1}{\omega^3} \frac{\partial k}{\partial x_j} \frac{\partial k}{\partial x_j} \]

\[ \beta_0 = \frac{9}{125}, \quad f_\beta = \frac{1 + 70 \chi_\omega}{1 + 80 \chi_\omega}, \quad \chi_\omega = \left| \frac{\Omega_{ij} \Omega_{jk} S_{ki}}{(\beta_0^* \omega)^3} \right| \]

The production term, \( P \), and the eddy viscosity definitions are given in Eqs. (9) and (16), respectively. This formulation is a modification to an earlier Wilcox \( k - \omega \) model\(^ {24} \) and is designed to improve model predictions for free shear layers and to reduce the solution sensitivity to freestream \( \omega \) values.

**Flow Predictions for Flat Plate**

Flow over a flat plate has been chosen as a high speed test case to illustrate the behavior of the laminar/turbulent flow results obtained with the one- and two-equation turbulence models. The test case is Mach 8 flow over a flat plate with a wall temperature of \( T_w = 1000 \) K and freestream conditions corresponding to an altitude of 15 km. For this case, the temperature in the flow is sufficiently low that perfect gas assumption with \( \gamma = 1.4 \) is reasonable.
**Freestream Flow Conditions**

The freestream conditions\(^{25}\) for the flat plate case are

\[
p_{\infty} = 1.21114 \times 10^4 \text{ N/m}^2, \quad T_{\infty} = 216.65 \text{ K}
\]
\[
\rho_{\infty} = 0.19475 \text{ kg/m}^3, \quad a_{\infty} = \sqrt{\frac{\gamma p_{\infty}}{\rho_{\infty}}} = 295.07 \text{ m/s}
\]
\[
V_{\infty} = a_{\infty} M_{\infty} = 2360.54 \text{ m/s}
\]
\[
\mu_{\infty} = \frac{1.458 \times 10^{-6} T_{\infty}^{3/2}}{(T_{\infty} + 110.4)} = 1.4216 \times 10^{-5} \text{ N s/m}^2
\]

where Sutherland law is used for the absolute viscosity. For the Spalart-Allmaras model, the restriction on the freestream eddy viscosity is determined by Eq. (5), which gives

\[
\mu_{T_{\infty}} < 5.27 \times 10^{-9} \text{ N s/m}^2
\]

for the Mach 8 flat plate flow case. The freestream eddy viscosity for all models was thus chosen as

\[
\mu_{T_{\infty}} < 1.0 \times 10^{-9} \text{ N s/m}^2
\]

unless indicated otherwise. For the two-equation models, the further specification of a freestream turbulence intensity of 0.01% was used to determine the turbulent kinetic energy in the freestream from the following relationship:\(^3\)

\[
k = \frac{1.2}{2} \left( \frac{T u}{100} V_{\infty} \right)^2
\]

**Computational Mesh for the Flat Plate**

A parabolic mesh has been used around the flat plate with the \((x, y)\) Cartesian coordinate system fixed at the leading edge. The parabolic mesh topology was chosen to mitigate the effects of
the leading edge singularity. The computational coordinates \( \xi, \eta \) are related to the Cartesian coordinates as follows:

\[
\begin{align*}
    x &= \alpha (\xi^2 - \eta^2) \\
    y &= 2\alpha \xi \eta \\
    0 &\leq \xi \leq \xi_{max} \\
    0 &\leq \eta \leq 1 \\
\end{align*}
\]

\[ \xi_{max} = \sqrt{1 + 1/\alpha} \]

The value of \( \xi_{max} \) has been determined by setting \( x = 1 \) at \( \eta = 1 \). This gives a mesh that is slightly longer than one meter along the flat plate. A uniform mesh is used in the \( \xi \) coordinate direction while a non-uniform mesh spacing is used in the \( \eta \) coordinate direction. The mesh spacing has been determined with the lower boundary stretching transformation of Roberts\(^2\) (see also Ref. 27). Most of the results have been obtained with 80x160 cells. A coarser mesh of 40x80 and a finer mesh of 160x320 have been used to show that the 80x160 mesh provides results sufficiently accurate for the figures presented. A stretching parameter of \( \beta = 1.001 \) has been used for the one-equation models. This choice for \( \beta \) gives maximum \( y^+ \) values of approximately 2.3 for the coarse 40x80 mesh. As expected, the maximum allowable \( y^+ \) values for the two-equation models were found to be much smaller than for the one-equation models, with the larger values resulting in convergence problems. Thus for the two-equation models, a stretching parameter of \( \beta = 1.00007 \) was used giving \( y^+ \leq 0.2 \) for the coarse mesh.

**Flat Plate Results with Standard Transition Method**

For the freestream conditions and meshes specified above, the laminar/turbulent flow has been calculated with the SACCARA code and compared to the accurate laminar and turbulent results obtained for this case by Van Driest.\(^{19,20}\) The standard transition method is used where the laminar viscosity is the sole contributor to the effective viscosity upstream of the transition plane. For this case, the Spalart-Allmaras model has the trip terms \( f_{t1} \) and \( f_{t2} \) set to zero. The transition location was specified at \( Re_{x_t} = 3.85 \times 10^6 \) (\( x_t = 0.1196 \text{ m} \)). The choice of the transition location
is arbitrary since the results will be compared to both laminar and turbulent flow theory. The $L_2$ norms of the residuals for both the momentum equations and the turbulence equations were reduced at least eight orders of magnitude in each case, suggesting that the results for the flat plate problem are not influenced by iterative convergence error.

Skin friction profiles have been obtained using all five turbulence models for the Mach 8 flat plate case. The Baldwin-Barth and both $k-\omega$ models give transition at the specified transition plane for the given freestream turbulence levels as shown in Fig. 1. In order to move the transition point to the desired location, the freestream eddy viscosity had to be increased to $1 \times 10^{-6} \, N\cdot s/m^2$ for the Spalart-Allmaras model, and the turbulence intensity had to be increased to 0.1% for the low Reynolds number $k-\varepsilon$ model (see Fig. 2). All of the models which correctly predict turbulent flow downstream of the transition point also predict skin friction in this region in agreement with the theory.

**Modified Transition Results for Spalart-Allmaras Model**

Solutions have been obtained with the Spalart-Allmaras trip functions $f_{1i}, f_{22}$ set to zero. The flow is turbulent along the flat plate for this case with the freestream eddy viscosity $\mu_T$ varying from $10^{-9}$ to $10^{-5} \, N\cdot s/m^2$. Solutions have been obtained with the trip function $f_{i2}$ included and $f_{i1}$ set to zero. For this case the flow transition location is dependent on the freestream eddy viscosity. When the eddy viscosity is $10^{-9} \, N\cdot s/m^2$, the flow remains laminar over the length of the flat plate. As previously discussed, numerical solutions show that the flow can be maintained laminar by making the production term $S_p$ zero by setting $c_{b1} = 0$ with the trip functions $f_{i1}, f_{i2}$ set to zero.

The complete Spalart-Allmaras model has trip terms included to control the transition location, but the formulation is not intended to model the transition flow region. The behavior of this model
has been investigated with the results for the local skin friction given in Fig. 3 where the trip location $x_t$ is specified ($x_t = 0.11964 \text{ m}$). The numerical predictions show that the transition location varies as the freestream eddy viscosity is increased above a value of approximately $10^{-9} \text{ N} \cdot \text{s/m}^2$. For these high speed flows, it is difficult to control the transition location with the suggested trip model. In addition, there is no control of the length of the transition region. Because of these experiences with the behavior of the Spalart-Allmaras trip model, different approaches have been investigated.

Three methods have been investigated to control the transition location and the length of transition as previously described. There are two parameters $x_t$ and $x_l$ introduced to control the transition behavior. The parameter $x_t$ is at the middle of the transition region while $x_x = x_t - x_l$ is the location upstream where transition starts and the location downstream $x_x = x_t + x_l$ where the flow becomes fully turbulent. The values for these parameters are chosen as

$$x_t = 0.1196 \text{ m}, \quad x_l = 0.1 \text{ m}, \quad Re_{x_t} = 3.84 \times 10^6$$

and these locations are also indicated in Fig. 4.

The results for the skin friction with the three proposed approaches for modeling transition have been investigated. All of the methods remain laminar a significant distance after the specified start of transition. With Method 1, where the trip function $f_{t2}$ is modified, transition occurs downstream of the desired location with very rapid transition onset. With Method 2, where the production coefficient $c_{b1}$ is modified, transition occurs near the desired location with a reasonable variation of the skin friction in the transition region. With Method 3, where the production term coefficient $c_{b1}(1 - f_{t2})$ is modified, transition occurs downstream of the desired transition location with very rapid transition onset. From this investigation it is concluded that Method 2 provides a reasonable technique to specify the transition location with limited control over the
transition region length. The results for Method 2 are presented in Fig. 4. When \( \lambda \) varies linearly over the transition region, there is better control. The transition control Method 2 appears to be insensitive to the freestream eddy viscosity.

**Flow Predictions for Reentry F Vehicle**

**Reentry F Description and Experimental Results**

The Reentry F flight experiment\(^5\) was performed in 1968 to provide measurements of wall heat transfer rates at reentry flow conditions that cannot be obtained in ground-based experimental facilities. The data is for the flow over a slender conical vehicle where there is only a small amount of surface ablation localized at the nosetip. The boundary layer flow is laminar, transitional, or turbulent depending on the altitude and location along the body surface. The Reentry F vehicle was a 5 degree sphere-cone with an initial nose radius of 0.00254 m (0.1 in) and the vehicle length is 4.0 m (13 ft). A graphite nosetip extended for the first 0.1915 m (7.54 in) followed by a conical beryllium frustum. The heat transfer measurements were obtained at altitudes between 36.6 and 18.3 km (120,000 and 60,000 ft). The data at a flight time of 456.0 seconds or an altitude of 24.4 km (80,000 ft) is used to validate the turbulence model predictions. Although this flight experiment provides exceptional data, there are many aspects of the flow conditions, body orientation, body shape, and wall surface temperature that are not completely or precisely known. Additional details of the flight experiment are given by Wright and Zoby.\(^5\)

The flow conditions at an altitude of 24.4 km (80,000 ft) are analyzed most often and are chosen for the present investigation. The freestream conditions used herein are based on the U. S. Standard Atmosphere, 1976,\(^{25}\) and are given below in SI units:

---

23
\[ M_\infty = 19.97, \quad \alpha = 0^\circ, \quad \rho_\infty = 0.043523 \text{ kg/m}^3 \]
\[ T_\infty = 221.034 \text{ K}, \quad T_w = 500 \text{ K}, \quad p_\infty = 2761.41 \text{ N/m}^2 \]
\[ V_\infty = 5951.858 \text{ m/s}, \quad a_\infty = 298.04 \text{ m/s} \]
\[ \mu_\infty = 1.445 \times 10^{-5} \text{ N} \cdot \text{s/m}^2, \quad \mu_T = 3.3227 \times 10^{-14} \text{ N} \cdot \text{s/m}^2 \]
\[ Tu = 0.01\%, \quad x_{\text{body}} = 4.0 \text{ m} \]

An assumed turbulence intensity \( Tu \) is used in the determination of the turbulent kinetic energy for the two-equation models. It should be noted that there is some amount of uncertainty in the specification of these properties. In addition, the experimentally reported angle of attack was 0.14 degrees, while zero degrees is assumed herein so that the axisymmetric flow assumption can be used.

Due to ablation, the nose radius increases to 0.00343 m at an altitude of 24.4 km. This result is an estimated value from an ablation analysis of the nosetip. For the present analysis, it is assumed that the nosetip shape remains a sphere-cone after ablation with the same cone half angle as the conical vehicle, which is \( \theta_{\text{cone}} = 5^\circ \). The nosetip is illustrated in Fig. 5. The origin in this figure is located at the virtual tip of the conical vehicle. For the approximated sphere-cone configuration in the Reentry F vehicle simulation, the location of the original nosetip and ablated nosetip 456 s into the flight trajectory is specified as

\[ x_0 = 0.012752 \text{ m}, \quad x_{\text{tip}}(0 \text{ s}) = 0.026603 \text{ m} \]
\[ x_{\text{tip}}(456 \text{ s}) = 0.035925 \text{ m} \]

In previous analyses of this vehicle, the coordinate \( x \) is defined as the axial distance without a clear definition of the origin location given in many cases. Some figures indicate that the origin is located at the ablated nose of the body. The axial location \( x \) in this paper is measured from the nosetip of the un-ablated vehicle. However, due to the small amount of ablation, the uncertainty
in the location of the axial heat flux measurements has a negligible impact on the results presented.

Due to the high velocities, the gas temperature is more than 6000 \textit{K} in the nosetip region with dissociation of the oxygen and nitrogen occurring. Downstream of the nose, on the conical portion of the vehicle, the temperature immediately behind the oblique shock is 420 \textit{K}, and perfect gas flow occurs. However, in the boundary layer the viscous dissipation increases the gas temperature to approximately 3000 \textit{K} and dissociation of oxygen occurs. At 24.4 \textit{km}, the chemical reactions are sufficiently fast that the air is assumed to be in local thermochemical equilibrium. There is some ablation of the nosetip which introduces chemical species from the ablation products into the boundary layer flow. As the amount of ablation is small, this influence has been neglected.

**Predictions of Wall Heat Flux for Reentry F Vehicle**

*Simulation Code and Model Approach*

The flow around the Reentry F vehicle has been calculated with the SACCARA\textsuperscript{11-14} Navier-Stokes code. This investigation is concerned with obtaining accurate numerical solutions of the wall heat flux based upon the input conditions to the code and models used in the simulation. The wall heat flux predictions are then compared with the flight measurements at an altitude of 24.4 \textit{km} (80,000 \textit{ft}). The solution is for the flow over the ablated vehicle. The small angle of attack of the vehicle (0.14\degree) is neglected and the flow is assumed to be axisymmetric. The solutions use a gas model of air in local thermochemical equilibrium and the flow is laminar over the front part of the body. The flow transitions to turbulent flow at a specified location. The turbulent flow has been modeled with the Baldwin-Barth and Spalart-Allmaras one-equation eddy viscosity approaches and the low Reynolds number \textit{k} – \varepsilon, Menter \textit{k} – \omega, and Wilcox (1998) \textit{k} – \omega two-equation turbulence models. The iterative convergence has been examined in order to assess the
accuracy of the steady-state solutions. Various levels of grid refinement were used to assess the spatial convergence errors of the numerical solutions. For the Spalart-Allmaras turbulence model, solutions have been obtained on four mesh levels: 100x40 cells (Mesh 0-f), 200x80 cells (Mesh 1-f), 400x160 cells (Mesh 2-f), and 800x320 cells (Mesh 3-f). The number of gridpoints are given in the axial and radial directions, respectively. For the two-equation models, solutions have been obtained on three mesh levels: 130x40 cells (Mesh 0-2eq), 260x80 cells (Mesh 1-2eq), and 520x160 cells (Mesh 2-2eq).

**Transition Model**

As previously discussed, the basic SACCARA code treats the transition process by setting the effective viscosity to the laminar value upstream of a specified transition plane, while downstream of this plane the effective viscosity is the sum of both the laminar and turbulent viscosities. This approach has been used with the Baldwin-Barth one-equation eddy viscosity model and all two-equation models. The transition plane is specified to be perpendicular to the vehicle axis and located at \( x = 2.6 \, m \). With the Spalart-Allmaras one-equation eddy viscosity model, a different approach has been implemented as described previously, with \( x_s = 1.8844 \, m \) and \( x_e = 2.8844 \, m \). From the results of the investigation of the flat plate flow case, it was concluded that Method 2 (coefficient \( c_{b1} \) is varied) is the best approach to control the transition process with the Spalart-Allmaras model at this time.

**Iterative Convergence of the Numerical Solutions**

The \( L_2 \) norms of the momentum and turbulence transport equations exhibited oscillatory behavior after only a two or three order of magnitude drop, thus another method was needed to monitor convergence. The iterative convergence has been initially determined by plotting the wall heat flux at various number of time steps and assuming convergence has been obtained when there is
no noticeable change in the results. This method is illustrated in Fig. 6 for the 400x160 cell mesh (Mesh 2-f) with the Spalart-Allmaras turbulence model. The laminar flow region takes the longest time to converge as there is a very fine mesh in the wall region. With Mesh 2-f, the wall heat flux appears to have no significant changes after 25,000 time steps. However, these results are misleading! A more careful analysis has been performed to estimate the iterative convergence error.

The accuracy of the wall heat flux $q^n$ relative to the steady-state value is determined by expressing the numerical solution at time $t^n$ as

$$q^n = q(t^n) = q_E + \varepsilon^n.$$  \hspace{1cm} (19)$$

The exact steady-state value of the wall heat flux is $q_E$ and the convergence error at time $t^n$ is $\varepsilon^n$.

The convergence error of the SACCARA code has been observed to have an exponential decrease in time which gives the following variation as the solution approaches a steady state

$$\varepsilon^n = \alpha e^{-\beta t^n}.$$  \hspace{1cm} (20)$$

where $\alpha$ and $\beta$ are constants. Eq. (19) and Eq. (20) may be combined and rewritten as

$$\beta t^n = \ln \alpha - \ln (q^n - q_E).$$  \hspace{1cm} (21)$$

Eq. (21) is evaluated at three time levels, $(n - 1)$, $n$, and $(n + 1)$, and the three relations are used to eliminate $\alpha$ and obtain

$$\beta (t^n - t^{n-1}) = \ln [(q^n - q_E)/(q^{n-1} - q_E)]$$

$$\beta (t^{n+1} - t^n) = \ln [(q^n - q_E)/(q^{n+1} - q_E)]$$

If the time increments are equal, then $(t^n - t^{n-1}) = (t^{n+1} - t^n)$ and the above becomes

$$(q^{n-1} - q_E)(q^{n+1} - q_E) = (q^n - q_E)^2$$
The exact steady-state value of the wall heat flux is solved for in the above equation which gives

\[ q_E = \frac{q^n - \Lambda^n q^{-1}}{1 - \Lambda^n} \]

where \( \Lambda^n = \frac{(q^{n+1} - q^n)}{(q^n - q^{n-1})} \).

(22)

The iterative convergence error becomes

\[ \varepsilon^n = -\frac{(q^{n+1} - q^n)}{(1 - \Lambda^n)} \]

and the percent convergence error relative to the exact steady-state value becomes

\[ \% \text{ Error of } q^n = -100 \left[ \frac{q^{n+1} - q^n}{q^n - \Lambda^n q^{n-1}} \right] \]

(23)

The foregoing results are based on the works of Ferziger and Peric\textsuperscript{28,29} for determining the convergence error of the numerical iterative solution of difference equations, but their results have been obtained with a different approach. In their work, the parameter \( \Lambda^n \) is the spectral radius (or the magnitude of the largest eigenvalue) of the iteration matrix. If the eigenvalues are complex, then the present approach is not appropriate. The complex eigenvalue case has been considered by Ferziger and Peric.

The above procedure is illustrated for the wall heat flux solution at \( x = 2.15 \text{ m} \) (where the flow is laminar) using the Spalart-Allmaras model. The percent error is shown in Fig. 7 for the four mesh levels. The local percent errors obtained from Eq. (23) based on time levels \( (n-1), n, \) and \( (n+1) \) are indicated by the symbols. The lines in Fig. 7 represent the percent error obtained from the best estimate of the exact solution given by Eq. (22). This best estimate is determined from the final three iteration levels of the solution. Due to the expense of the fine grid Mesh 3-f calculation (800x320 cells), the converged Mesh 2-f results were used to provide an initial starting solution for this case. The initial solution results on Mesh 2-f (shown in Fig. 6) appeared converged at
25,000 iterations; however, the above error analysis indicates that the local iterative errors are on the order of 4%. An additional 15,000 iterations were needed to reduce the error down to 0.2%. The iterative solution errors are much smaller than the spatial solution errors, as will be demonstrated.

The iterative convergence for the two-equation turbulence models was also examined for the three mesh levels. Results of the iterative error analysis at \( x = 2.12 \, m \) are presented for the low Reynolds number \( k - \varepsilon \) model (Fig. 8), the Menter \( k - \omega \) model (Fig. 9), and the Wilcox (1998) \( k - \omega \) model (Fig. 10). A larger number of iterations were required due to the finer mesh requirements for the two-equation turbulence models (Mesh 2eq) versus the one-equation models (Mesh f). The two-equation results were also converged to less than 0.1% error.

**Spatial Convergence of the Numerical Solutions**

Spatial convergence has been assessed from the steady-state solutions with the Spalart-Allmaras turbulence model on the four meshes. The wall heat flux obtained from mesh 0, 1, 2, and 3 (from coarsest to finest) is given in Fig. 11 with the variable spacing given by Mesh f. The Richardson Extrapolation procedure\(^{30}\) has been used to obtain a more accurate result from the relation

\[
q_{RE} = q_{2,3} = q_3 + (q_3 - q_2)/3. \tag{24}
\]

The above relation assumes that the numerical scheme is second-order both within the domain and at the boundaries. While recent findings\(^{31,32}\) have shown that flows with captured shock waves will tend towards first order as the mesh is refined, analyses which account for mixed first and second order behavior are beyond the scope of the current work. The results from Ref. 32 indicate that the application of standard second order Richardson Extrapolation to mixed order problems does provide good estimates of the exact solution. The Richardson Extrapolation result
and the solution on Mesh 2-f and 3-f are nearly the same (also shown in Fig. 11). The accuracy of the solutions on the four meshes has been estimated with the exact solution approximated with $q_{RE}$ which gives the solution error as

$$\% \text{ Error of } q_M = 100\left(q_M - q_{RE}\right)/q_{RE}$$

where $M = 0, 1, 2, \text{ or } 3$ refers to the mesh level.

If the mesh has been refined sufficiently where the solution error has second-order behavior, then the errors on the four meshes have the following relationship

$$\% \text{ Error of } q_3 = \frac{\% \text{ Error of } q_2}{4} = \frac{\% \text{ Error of } q_1}{16} = \frac{\% \text{ Error of } q_0}{64}$$

In the above equation, the first equality will always be satisfied when Eq. (24) has been used. The other equalities will only be satisfied if the mesh has been sufficiently refined to be in the asymptotic range. The normalized percent error of the wall heat flux along the vehicle is presented in Fig. 11. The laminar and turbulent flow regions are in the asymptotic range, while the transitional flow region is not in the asymptotic range. This result is not surprising since the Mesh f grid does not use axial clustering at the transition region. The wall heat flux prediction in the laminar and fully turbulent regions have fine grid errors (Mesh 3-f) of less than 0.5%. Only the solutions on Mesh 2-f and 3-f are considered sufficiently accurate for comparison with the flight measurements. The Richardson extrapolated results provide even a more accurate numerical prediction.

Spatial convergence has also been examined for the two-equation turbulence models using three mesh levels. The spatial error of the heat flux is given in Figs. 12-14 for the low Reynolds number $k - \varepsilon$, the Menter $k - \omega$, and the Wilcox (1998) $k - \omega$ models, respectively. The spatial error in the laminar regions is under 2%, while in the transitional and turbulent regions the errors are under 5%. The results for both $k - \omega$ models indicate that the heat flux does not show a fully
second order grid convergence behavior, even in the fully turbulent region. The spike in the error for the two-equation models is due to movement of the transition location on the different size meshes and is more dramatic for the two-equation models due to the fine axial spacing around the transition point.

Wall Heat Flux

The predictions of the wall heat flux on the Reentry F vehicle at an altitude of 24.4 km (80,000 ft) with the one-equation turbulence models are given in Fig. 15 along with the flight data. The Spalart-Allmaras prediction uses the numerical solution with Mesh 2-f and the Richardson Extrapolation results for this case. The Spalart-Allmaras model overpredicts the laminar wall heat flux by roughly 10 percent while the turbulent wall heat flux is overpredicted by approximately 15 percent. At this altitude the vehicle has a 0.14 degree angle of attack and the heat transfer measurements were made on the leeward side of the conical body. A full three-dimensional solution, with the vehicle at angle of attack, would bring the prediction and flight data into closer agreement. The prediction with Mesh 2-f is believed to be a sufficiently accurate steady-state solution that it can be used to validate the turbulence model, but there is some uncertainty in these results due to uncertainties in the freestream conditions and the flight measurements as discussed previously.

The simulation with the Baldwin-Barth turbulence model (also shown in Fig. 15) overpredicts the laminar wall heat flux by roughly 10 percent and is in agreement with the simulation with the Spalart-Allmaras model. Of course, in the laminar flow region the turbulence models should have no impact on the flow solution. The turbulent wall heat flux is overpredicted by roughly 100 percent with the Baldwin-Barth turbulence model. It is recommended that the Spalart-Allmaras model should be used rather than the Baldwin-Barth turbulence model for reentry flows.
Results with the Nagano and Hishida $k-\varepsilon$, the Menter $k-\omega$, and the Wilcox (1998) $k-\omega$ models are presented in Fig. 16. Fine grid results with Mesh 2eq are shown along with the results from Richardson Extrapolation. Again, the surface heat flux is overpredicted by approximately 10% in the laminar region. The $k-\varepsilon$ results show an overprediction of the turbulent heating rates by approximately 100%, possibly due to the use of the incompressible form of the turbulent kinetic energy production term. The two $k-\omega$ models show better agreement with the flight data, with the Menter model within 40% and the Wilcox (1998) model within 30% of the data. All three models display a peak in the turbulent heating just downstream of the specified transition plane, which is possibly due to crude behavior of the standard transition method.

**Conclusions**

For the Mach 8 flat plate boundary layer flow with the standard transition method, the Baldwin-Barth and both $k-\omega$ models gave transition at the specified location. The Spalart-Allmaras and low Reynolds number $k-\varepsilon$ models required an increase in the freestream turbulence levels in order to give transition at the desired location. All models predicted the correct skin friction levels in both the laminar and turbulent flow regions.

For Mach 8 flat plate case, the transition location could not be controlled with the trip terms as given in the Spalart-Allmaras model. Several other approaches have been investigated to allow the specification of the transition location. The approach that appears most appropriate is to vary the coefficient that multiplies the turbulent production term in the governing partial differential equation for the eddy viscosity (Method 2). When this coefficient is zero, the flow remains laminar. The coefficient is increased to its normal value over a specified distance to crudely model the transition region and obtain fully turbulent flow.
Predictions have been obtained for the Reentry F flight vehicle with both one- and two-equation turbulence models where the transition location is specified \textit{a priori}. Care has been taken to quantify the errors in surface heat flux distributions due to both iterative and grid convergence. The $L_2$ norms of the residuals exhibited oscillatory behavior after a three order of magnitude drop, thus requiring alternative methods for monitoring iterative convergence. A new method for iterative convergence error, based on the works of Ferziger and Peric,\textsuperscript{28,29} was used to reduce the iterative convergence errors below approximately 0.1\% for all cases. Simulations were performed on three grid levels for the two-equation turbulence models and four grid levels for the Spalart-Allmaras model to assess the grid convergence errors. The errors in the laminar and turbulent regions were reduced to 2\% and 5\%, respectively, with the two-equation models and to below 0.5\% for the Spalart-Allmaras model. Richardson Extrapolation was employed with the two finest grid solutions (assuming second order spatial accuracy) to get even more accurate surface heat flux solutions.

For the Reentry F flight simulations, the axisymmetric turbulent predictions for wall heat flux with the Spalart-Allmaras, Menter $k-\omega$, and Wilcox (1998) $k-\omega$ models are in reasonable agreement with the flight measurements. The wall heat flux in the turbulent region is overpredicted by 15\% with the Spalart-Allmaras model, 30\% with the Wilcox (1998) $k-\omega$ model, and 40\% with the Menter $k-\omega$ model. These axisymmetric simulations assume the vehicle is at zero degree angle-of-attack; thus, the agreement with the leeward-side data will improve if three-dimensional are performed at the reported 0.14 degree angle-of-attack. The Spalart-Allmaras model predictions for this case are much better than the results from the Baldwin-Barth model. The low Reynolds number $k-\varepsilon$ two-equation model greatly overpredicts the heating in the fully turbulent region.
Acknowledgments

This work was supported by Sandia National Laboratories and the Department of Energy’s Accelerated Strategic Computing Initiative.

References


Fig. 1: Transition location with one- and two-equation turbulence models for Mach 8 flat plate flow with the standard transition model.
Fig. 2: Transition location for various freestream turbulence levels with the Spalart-Allmaras and $k-\varepsilon$ turbulence models for Mach 8 flat plate flow (standard transition model).
Fig. 3: Transition location for different freestream eddy viscosities with the Spalart-Allmaras turbulence model ($f_{l1}$ and $f_{l2}$ terms included) for Mach 8 flat plate flow.
Fig. 4: Transitional flow behavior for Spalart-Allmaras (Method 2) with various freestream eddy viscosities for Mach 8 flat plate flow.
Fig. 5: Diagram of the Reentry F flight vehicle with the nosetip location at times $t = 0 \, s$ and $t = 456 \, s$ in the flight trajectory indicated.
Fig. 6: Iterative convergence of surface heat flux on Mesh 2-f with the Spalart-Allmaras turbulence model.
Fig. 7: Iterative convergence error of the surface heat flux for the Spalart-Allmaras model at \( x = 2.14892 \, m \) with Mesh f.
Fig. 8: Iterative convergence error of the surface heat flux for the $k-\varepsilon$ model at $x = 2.11697$ m with Mesh 2eq.
Fig. 9: Iterative convergence error of the surface heat flux for the Menter $k-\omega$ model at $x = 2.11697\, m$ with Mesh 2eq.
Fig. 10: Iterative convergence error of the surface heat flux for the Wilcox (1998) $k-\omega$ model at $x = 2.11697 \, m$ with Mesh 2eq.
Fig. 11: Spatial convergence of heat flux as the Mesh f is refined with the Spalart-Allmaras turbulence model.
Fig. 12: Error in heat flux along the vehicle with Mesh 2eq refinement using the low Reynolds number $k-\varepsilon$ model.
Fig. 13: Error in heat flux along the vehicle with Mesh 2eq refinement using the Menter $k-\omega$ model.
Fig. 14: Error in heat flux along the vehicle with Mesh 2eq refinement using the Wilcox (1998) $k-\omega$ model.
Fig. 15: Comparison of flight data for wall heat flux along Reentry F vehicle at an altitude of 24.4 km (80,000 ft) with predictions of the one-equation turbulence models.
Fig. 16: Comparison of flight data for wall heat flux along Reentry F vehicle at an altitude of 24.4 km (80,000 ft) with predictions of the two-equation turbulence models.