A Simple Two-Equation Turbulence Model For Transition-Sensitive Cfd Simulations Of Missile Nose-Cone Geometries

Joseph Matthew Jones

Follow this and additional works at: https://scholarsjunction.msstate.edu/td

Recommended Citation
https://scholarsjunction.msstate.edu/td/251

This Graduate Thesis is brought to you for free and open access by the Theses and Dissertations at Scholars Junction. It has been accepted for inclusion in Theses and Dissertations by an authorized administrator of Scholars Junction. For more information, please contact scholcomm@msstate.libanswers.com.
A SIMPLE TWO-EQUATION TURBULENCE MODEL FOR TRANSITION-SENSITIVE CFD SIMULATIONS OF MISSILE NOSE-CONE GEOMETRIES

By

Joseph Matthew Jones

A Thesis
Submitted to the Faculty of
Mississippi State University
in Partial Fulfillment of the Requirements
for the Degree of Master of Science
in Mechanical Engineering
in the Department of Mechanical Engineering

Mississippi State University

December 2007
A SIMPLE TWO-EQUATION TURBULENCE MODEL FOR TRANSITION-SENSITIVE CFD SIMULATIONS OF MISSILE NOSE-CONE GEOMETRIES

By

Joseph Matthew Jones

Approved:

D. Keith Walters
Assistant Professor of Mechanical Engineering
(Major Professor)

B. Keith Hodge
Professor of Mechanical Engineering
(Committee Member)

Montgomery C. Hughson
Associate Research Professor of Computational Engineering, and Deputy Director of the HPC² SimCenter
(Committee Member)

Steven R. Daniewicz
Professor of and Graduate Coordinator in the Department of Mechanical Engineering

Roger King
Associate Dean for Research and Graduate Studies of the Bagley College of Engineering
This study reports the development and validation of a modified two-equation eddy-viscosity turbulence model for computational fluid dynamics prediction of transitional and turbulent flows. The existing terms of the standard k-ω model have been modified to include transitional flow effects, within the framework of Reynolds-averaged, eddy-viscosity turbulence modeling. The new model has been implemented into the commercially available flow solver FLUENT and the Mississippi State University SimCenter developed flow solver U^2NCLE. Test cases included flow over a flat plate, a 2-D circular cylinder in a crossflow, a 3-D cylindrical body and three conical geometries, which represent the nose-cones of aerodynamic vehicles such as missiles. The results illustrate the ability of the model to yield reasonable predictions of transitional flow behavior using a simple modeling framework, including an appropriate response to freestream turbulence quantities, boundary-layer separation, and angle of attack.
ACKNOWLEDGEMENTS

I would like to thank my research advisor, Dr. Keith Walters. Throughout my graduate career, he was always available to offer his guidance and aid. Without his vast knowledge of the subject material and support on the project, the completion of this thesis and research would not have been possible. I would also like to thank the members of my graduate committee, Dr B. Keith Hodge and Dr. Montgomery C. Hughson, for their support in the completion of this thesis. In addition, I would like to acknowledge the MSU HPC² for the use of their resources and the Department of Defense for providing the research funding. I would especially like to give thanks to my family for all their love and support over the many years. Without their guidance, I would not be as well prepared for the future as I am today.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>I.</td>
<td>INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>II.</td>
<td>TURBULENCE AND TRANSITION MODELING</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>2.1 Turbulence Modeling</td>
<td>5</td>
</tr>
<tr>
<td></td>
<td>2.2 Modes of Transition</td>
<td>7</td>
</tr>
<tr>
<td></td>
<td>2.3 Transition Models</td>
<td>9</td>
</tr>
<tr>
<td></td>
<td>2.3.1 Direct Numerical Simulation and Large-Eddy Simulation</td>
<td>9</td>
</tr>
<tr>
<td></td>
<td>2.3.2 ed Modeling</td>
<td>11</td>
</tr>
<tr>
<td></td>
<td>2.3.3 Correlation-based Modeling</td>
<td>12</td>
</tr>
<tr>
<td></td>
<td>2.3.4 Intermittency Modeling for the Duration of Transition</td>
<td>15</td>
</tr>
<tr>
<td></td>
<td>2.3.5 Laminar Kinetic Energy</td>
<td>18</td>
</tr>
<tr>
<td>III.</td>
<td>NEW MODEL DEVELOPMENT</td>
<td>20</td>
</tr>
<tr>
<td></td>
<td>3.1 Objective</td>
<td>20</td>
</tr>
<tr>
<td></td>
<td>3.2 Derivation</td>
<td>21</td>
</tr>
<tr>
<td></td>
<td>3.2.1 Fully-turbulent Region</td>
<td>22</td>
</tr>
<tr>
<td></td>
<td>3.2.2 Growth of Pretransitional Disturbances</td>
<td>25</td>
</tr>
<tr>
<td></td>
<td>3.2.3 Transition Inception</td>
<td>27</td>
</tr>
<tr>
<td>IV.</td>
<td>SOLVER IMPLEMENTATION</td>
<td>30</td>
</tr>
<tr>
<td></td>
<td>4.1 FLUENT</td>
<td>30</td>
</tr>
</tbody>
</table>
LIST OF TABLES

<table>
<thead>
<tr>
<th>TABLE</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.1</td>
<td>Flat Plate Leading-edge Turbulence Quantities</td>
</tr>
<tr>
<td>5.2</td>
<td>ERCOFTAC Leading-edge Turbulence Quantities</td>
</tr>
<tr>
<td>6.1</td>
<td>10° Cone Test Matrix</td>
</tr>
<tr>
<td>6.2</td>
<td>5° Cone Test Matrix</td>
</tr>
<tr>
<td>A.1</td>
<td>FLUENT Model Constants</td>
</tr>
<tr>
<td>A.2</td>
<td>U^2NCLE Model Constants</td>
</tr>
<tr>
<td>B.1</td>
<td>AoA Angle Calculations</td>
</tr>
</tbody>
</table>
### LIST OF FIGURES

<table>
<thead>
<tr>
<th>FIGURE</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.1</td>
<td>Schematic for flat-plate flow conditions</td>
</tr>
<tr>
<td>5.2</td>
<td>$C_f$ vs. $Re_x$ for FLUENT validation test cases</td>
</tr>
<tr>
<td>5.3</td>
<td>$C_f$ vs. $Re_x$ for FLUENT ERCOFTAC test cases</td>
</tr>
<tr>
<td>5.4</td>
<td>Schematic for cylinder in cross-flow simulations</td>
</tr>
<tr>
<td>5.5</td>
<td>Turbulence-intensity contours in the cylinder subcritical regime</td>
</tr>
<tr>
<td>5.6</td>
<td>Turbulence-intensity contours in the cylinder supercritical regime</td>
</tr>
<tr>
<td>6.1</td>
<td>Grid (flow domain) used for flat-plate simulations</td>
</tr>
<tr>
<td>6.2</td>
<td>Grid (plate) used for flat-plate simulations</td>
</tr>
<tr>
<td>6.3</td>
<td>$C_f$ vs. $Re_x$ for U$^2$NCLE validation test cases</td>
</tr>
<tr>
<td>6.4</td>
<td>$C_f$ vs. $Re_x$ for U$^2$NCLE test cases with $Re_T = 100$; Natural transition calibration</td>
</tr>
<tr>
<td>6.5</td>
<td>$C_f$ vs. $Re_x$ for U$^2$NCLE ERCOFTAC test cases</td>
</tr>
<tr>
<td>6.6</td>
<td>Grid (flow domain) used for circular cylinder axial-flow simulations</td>
</tr>
<tr>
<td>6.7</td>
<td>Grid (cylinder) used for circular cylinder axial-flow simulations</td>
</tr>
<tr>
<td>6.8</td>
<td>$C_f$ vs. $Re_x$ for the cylinder axial-flow simulations; $Re_T = 100$ and $Ma = 1.0$</td>
</tr>
<tr>
<td>6.9</td>
<td>Contours of $C_f$ and $\mu_t$; $Tu = 0.2%$, $Re_T = 100$ and $Ma = 1.0$</td>
</tr>
<tr>
<td>6.10</td>
<td>Contours of $C_f$ and $\mu_t$; $Tu = 2.0%$, $Re_T = 100$ and $Ma = 1.0$</td>
</tr>
<tr>
<td>6.11</td>
<td>Schematic for sharp-nosed cone flow conditions</td>
</tr>
<tr>
<td>6.12</td>
<td>Grid (flow domain) used for conical geometry simulations</td>
</tr>
<tr>
<td>FIGURE</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
</tr>
<tr>
<td>6.13</td>
<td>Grid (cone tip) used for conical geometry simulations</td>
</tr>
<tr>
<td>6.14</td>
<td>Sample contour plot of $C_f$ for U²NCLE cone cases</td>
</tr>
<tr>
<td>6.15</td>
<td>$C_f$ vs. $Re_x$ for the 10° cone, with varied $Tu$, $Re_T = 100$ and $Ma = 1.05$</td>
</tr>
<tr>
<td>6.16</td>
<td>$C_f$ vs. $Re_x$ for the 10° cone, with $Tu = 0.01%$, varied $Re_T$ and $Ma = 1.05$</td>
</tr>
<tr>
<td>6.17</td>
<td>$C_f$ vs. $Re_x$ for the 10° cone, with $Tu = 0.1%$, varied $Re_T$ and $Ma = 1.05$</td>
</tr>
<tr>
<td>6.18</td>
<td>$C_f$ vs. $Re_x$ for the 10° cone, with $Tu = 0.5%$, varied $Re_T$ and $Ma = 1.05$</td>
</tr>
<tr>
<td>6.19</td>
<td>$C_f$ vs. $Re_x$ for the 10° cone, with $Tu = 0.5%$, $Re_T = 100$ and varied $Ma$ number</td>
</tr>
<tr>
<td>6.20</td>
<td>$Re_l$ vs. $Ma$ with $Re_T = 100$ for the 5° cone of Fisher and Dougherty [13]</td>
</tr>
<tr>
<td>6.21</td>
<td>$Re_l$ vs. $Ma$ with $Re_T = 100$ for the 5° cone of Chen et al. [5]</td>
</tr>
<tr>
<td>6.22</td>
<td>$C_f$ contours for the 10° cone at $AoA = 1°$</td>
</tr>
<tr>
<td>6.23</td>
<td>$Re_l$ vs. $AoA$ with $Tu = 0.5%$ and $Re_T = 100$ for the 10° cone</td>
</tr>
<tr>
<td>7.1</td>
<td>$C_f$ vs. $Re_x$ for a 5° cone, with $Tu = 0.7817%$, $Re_T = 100$ and $0.4 \leq Ma \leq 1.0$</td>
</tr>
<tr>
<td>7.2</td>
<td>$C_f$ vs. $Re_x$ for a 5° cone, with $Tu = 0.7817%$, $Re_T = 100$ and $1.0 \leq Ma \leq 1.8$</td>
</tr>
<tr>
<td>7.3</td>
<td>$C_f$ vs. $Re_x$ for a 5° cone, with varied $Tu$, $Re_T = 100$ and $Ma = 1.0$</td>
</tr>
<tr>
<td>7.4</td>
<td>Close-up view of transition locations from Figure 7.3</td>
</tr>
<tr>
<td>7.5</td>
<td>$C_f$ vs. $Re_x$ for the T3A ERCOFTAC test case; Freestream turbulence limiter validation</td>
</tr>
<tr>
<td>7.6</td>
<td>$Re_l$ vs. $Ma$ with $Re_T = 100$ for the 5° cone; Freestream turb. limiter validation</td>
</tr>
<tr>
<td>C.1</td>
<td>Method for calculating leading edge turbulence quantities</td>
</tr>
</tbody>
</table>
LIST OF SYMBOLS, ABBREVIATIONS, AND NOMENCLATURE

$bl_{var}$ ratio of turbulent length scale to wall-normal distance

$C_\mu$ eddy-viscosity coefficient

$d$ wall-normal distance

$E_{\omega\omega}$ destruction term

$f_{bl}$ damping function on $\omega$ for freestream limiter

$f_p$ production damping function

$f_{Re_T}$ $Re_T$ damping function

$f_{TS}$ T-S damping function

$f_{\omega\omega}$ inviscid wall damping function

$f_\mu$ eddy-viscosity damping function

$k$ turbulent kinetic energy

$k_{FS}$ freestream turbulent kinetic energy

$Ma$ freestream Mach number

$P_k$ turbulent kinetic energy production term

$P_{\omega\omega}$ production term

$R_\omega$ transitional source contribution for $\omega$

$Re_D$ Reynolds number based on diameter
$Re_L$ Reynolds number based on length

$Re_T$ turbulent Reynolds number

$Re_t$ transition Reynolds number

$Re_\varepsilon$ ratio of wall distance to Kolmogorov eddy scale

$Re_\theta$ momentum-thickness Reynolds number

$Re_{\theta t}$ momentum-thickness Reynolds number at transition inception

$S$ magnitude of strain-rate tensor

$S_{ij}$ strain-rate tensor

$Tu$ freestream turbulence intensity

$U_\infty$ freestream velocity

$U_i$ mean velocity component

$u_i$ instantaneous velocity component

$\beta_{TS}$ natural transition model term

$\delta_{ij}$ Kronecker delta function

$\varepsilon_w$ turbulent kinetic energy wall destruction term

$\gamma$ time-scale parameter

$\eta_k$ Kolmogorov length scale

$\lambda_{eff}$ effective (wall-limited) length scale

$\lambda_T$ turbulent length scale

$\mu$ dynamic viscosity

$\mu_T$ turbulent viscosity
\( \nu \)  kinematic viscosity

\( \rho \)  fluid density

\( \Omega \)  magnitude of rotation-rate tensor

\( \Omega_{ij} \)  rotation-rate tensor

\( \omega \)  specific dissipation rate (inverse turbulent time scale)

**Subscripts**

\( i,j,k \)  computational indices for the x, y and z directions

**Abbreviations**

AoA  Angle of Attack

CFD  Computational Fluid Dynamics

DNS  Direct Numerical Simulation

HPC\(^2\)  High Performance Computing Collaboratory

LES  Large Eddy Simulation

MSU  Mississippi State University

N-S  Navier-Stokes

RANS  Reynolds-Averaged Navier-Stokes

**SIMCENTER**  Computational Simulation and Design Center

T-S  Tollmien-Schlichting (disturbances)

U\(^2\)NCLE  Unstructured Unsteady Computation of Field Equations

UDF  User-Defined Function
CHAPTER I
INTRODUCTION

Computational Fluid Dynamics (CFD) has developed into an important tool for predicting flow behavior in a wide variety of applications, including aerospace, automotive, biomedical, chemical processing, heating and cooling, and power generation. Typical CFD simulations are performed by characterizing the flowfield as either laminar or fully turbulent. However, modeling the transition of a boundary layer from laminar to turbulent flow is critically important in many engineering design and production simulations. For example, the ability to accurately model the variation of skin friction due to laminar-turbulent transition is critical in aerospace and automotive drag reduction studies. In heat transfer applications, such as the cooling of turbomachinery and the design of heat exchangers, the transition from laminar to turbulent flow is characterized by an increase in the heat transfer coefficient due to the enhanced mixing of the turbulent flow. Similar dependencies on the transition of a boundary layer can be found in other applications as well.

Therefore, the need for some method to fully resolve a transitional boundary layer or to simply estimate the point of transition inception is recognized. Although Direct Numerical Simulation (DNS) will most likely be the method of choice in the future, the vast complexity of turbulence dynamics and deficiencies in computing power are the current limiting factors. Over the past few decades, many models have been developed
and presented in the literature that address the resolution of transitional flowfields. The types of transitional models developed include models based upon empirical correlations, models developed from the stability analysis of the Orr-Sommerfeld equation, Large Eddy Simulation (LES) methods, and models developed within the Reynolds Averaged Navier-Stokes (RANS) framework. The transition of a laminar boundary layer to fully-turbulent flow has been shown to be dependent on many different parameters [1], including, but not limited to, the following:

- Freestream turbulence quantities
- Adverse and favorable pressure gradients
- Reynolds number
- Mach number
- Surface roughness
- Surface temperature
- Surface curvature

Since the transition process is influenced by so many different parameters, developing an accurate model for transition prediction that includes the affects of each is very difficult. Experiments that investigate the transition process typically provide data for variations in only a sampling these parameters [18].

The current study focuses on the development of a simple, robust transition sensitive two-equation, eddy-viscosity turbulence model developed within the framework of Reynolds Averaged Navier-Stokes (RANS) based CFD. The model is based on the standard k-ω turbulence model and can be implemented into existing CFD solvers.
Appropriate model response for laminar, transitional and fully-turbulent boundary layers is controlled by the manner in which the terms within the equations for $k$ and $\omega$ are defined. Inviscid and viscous damping functions are included in the formulation of the eddy viscosity and the production term for turbulent kinetic energy to reproduce the appropriate behavior in the laminar, pretransitional and fully-turbulent flow regimes. Within the chapters to follow, the details of model development, validation, and application to experimental test cases are summarized.
The majority of fluid flows encountered by scientists and engineers are turbulent. Before approaching any fluid flow problem, an understanding of the nature of turbulent flows is important. The definition of turbulent flow is given as “a spatially varying mean flow with superimposed 3-D random fluctuations that are self-sustaining and enhance mixing, diffusion, entrainment and dissipation” [54].

Turbulent flows are characterized by packets of fluid known as “eddies.” Turbulent eddies generate random fluctuations in velocity that are superimposed onto the mean (averaged) flow quantities. In turbulent flows, a continuous distribution of eddy sizes exists, forming a cascade. Energy is removed from the bulk flow by the largest eddies and passed down the cascade to the smallest eddies, where it is dissipated by viscous forces. Turbulent flows are generally self-sustaining; hence, new eddies are continuously formed to replace those lost to viscous dissipation. Since eddies are continuously fluctuating in all three dimensions, the mixing of mass, momentum and energy is much greater than that of laminar flow, yielding enhanced heat transfer and increased skin friction.

Direct numerical simulation (DNS) provides a means to simulate the effects of randomly fluctuating eddies. However, fully resolving the turbulent flow characteristics at the smallest scales requires an excessively fine mesh, leading to a large computational
load. A more economical approach is to model the effects of the turbulent fluctuations imposed on the mean flow.

2.1 Turbulence Modeling

In order to model the turbulent effects in the flow, the flow is assumed to be in an unsteady fluctuating state. Any of the fluctuating flow variables can be decomposed into a mean value and a fluctuating component. This is accomplished using the following relation [54], known as Reynolds Averaging:

\[ \overline{Q} = \frac{1}{T} \int_{t_0}^{t_0+T} Q dt \]  

(2.1)

where \( Q \) represents any random variable and \( T \) is much larger than the characteristic fluctuation period. For statistically stationary flow, the averaged variable, \( \overline{Q} \), represents the limiting case as \( T \to \infty \). The Navier-Stokes equations of fluid motion can also be Reynolds-averaged in this manner, where the unsteady equations become formally steady. The Reynolds-Averaged Navier-Stokes (RANS) equations for incompressible conservation of mass and momentum are [32]:

\[ \frac{\partial}{\partial x_j} (\rho U_j) = 0 \]  

(2.2)

\[ \frac{\partial}{\partial x_j} (\rho U_i U_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( 2\mu S_{ij} - \rho \overline{u_i u_j} \right) \]  

(2.3)
where the instantaneous time dependent variables are decomposed into mean \((P_i, U_i)\) and fluctuating \((p_i, u_i)\) components. The \(S_{ij}\) term in Equation (2.3) is the deviatoric rate of strain and is defined by:

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{1}{3} \frac{\partial U_k}{\partial x_k} \partial_{ij} \tag{2.4}
\]

As a result of the Reynolds Averaging process, a new term \((\rho u_i u_j)\) is introduced into the momentum equation. This new term is often referred to as the Reynolds Stress Tensor or the “turbulent stress.” Note that the Reynolds Stress is simply a mathematical artifact of the Reynolds Averaging process and represents the transport of mean momentum by the fluctuating velocity components. Since the Reynolds Stress is not defined as a known function of the mean flow variables, \(U_i\) and \(P\), the system of equations is “unclosed.” The development of methods to “close” the equation set is the main focus in the field of turbulence modeling.

The most common approach is to model the anisotropic part of the Reynolds Stress Tensor by employing the eddy-viscosity hypothesis [32]. By assuming that the Reynolds Stress is analogous to molecular shear (Boussinesq hypothesis), the Reynolds Stress is proportional to the mean strainrate, as defined in Equation (2.5):

\[
\rho \overline{u_i u_j} - \frac{2}{3} \rho \overline{u_k u_k} \delta_{ij} = -2\mu_T S_{ij} \tag{2.5}
\]

where all of the fluctuating velocity effects are represented by the single scalar variable, \(\mu_T\). This new variable is referred to as the eddy or turbulent viscosity and is generally defined as a function of a characteristic turbulent velocity scale and turbulent length scale:
\( \mu_T \sim \rho u \hat{\lambda}_T \)  

(2.6)

RANS-based, eddy-viscosity turbulence modeling focuses on the specification of values for \( u \) and \( \lambda_T \). In the present study, a two-equation turbulence model is used, in which transport equations are solved for turbulent kinetic energy, \( k \), and the inverse turbulent time scale, \( \omega \). The turbulent scale values, \( u \) and \( \lambda_T \), are related to \( k \) and \( \omega \). The standard forms of these transport equations are given as:

\[
\frac{\partial}{\partial x_j} \left( \rho U_j k \right) = P_k - \rho \omega k - \varepsilon_w + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \tag{2.7}
\]

\[
\frac{\partial}{\partial x_j} \left( \rho U_j \omega \right) = P_\omega - E_\omega + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] \tag{2.8}
\]

The turbulent viscosity is then defined as:

\[
\mu_T = \rho C_\mu \frac{k}{\omega} \tag{2.9}
\]

where \( C_\mu \) can be treated as either a constant or a variable. For the present study, \( C_\mu \) is treated as a variable to be defined later. The \( k-\omega \) turbulence model was selected for this study because it has shown superior performance over other two-equation models, such as the \( k-\varepsilon \) model, in the viscous near-wall region and in the presence of streamwise pressure gradients [55].

2.2 Modes of Transition

In order to obtain accurate predictions for the transitional flows, an understanding of the modes of transition is important. The three primary modes of transition are given
as: natural transition, “bypass” transition, and separated flow transition [28]. Schlichting [38] provides a detailed description of the natural transition process. Linear stability theory suggests that transition occurs as result of disturbance waves present in the pretransitional boundary layer, referred to as Tollmien-Schlichting (T-S) waves. When the momentum thickness Reynolds number reaches a critical value, the Tollmien-Schlichting waves begin to break down and the disturbances are amplified exponentially. In regions of the highest fluctuations, “turbulent spots” develop in the surrounding laminar flow. As these turbulent spots moves downstream, they increase in size, until a fully-turbulent boundary layer develops. Natural transition is typically defined to occur when the freestream value of turbulence intensity is less than one percent (Tu < 1.0%) [28].

For flows where the freestream turbulence is high, the development and convection of turbulent spots are influenced directly by the freestream disturbances. The disturbances associated with natural transition are said to be “bypassed.” Linear stability theory is, therefore, omitted, since the presence of T-S waves in bypass transition has not been verified by experiments [28]. The influence of relatively high freestream turbulence levels on the pre-transitional boundary layer was first described in detail by Klebanoff [22]. The disturbances are characterized by streamwise velocity streaks of velocity fluctuations in the pretransitional boundary layer. In a similar manner as the T-S waves, the Klebanoff disturbances are amplified as they move downstream and break down into turbulent spots. The turbulent spots later evolve into a fully-turbulent boundary layer.
For flows in which a laminar boundary layer detaches from the surface, transition can occur within the shear layer and away from the surface. This mode is often called “separated flow transition” [28]. After transition occurs, the turbulent boundary layer may reattach to the surface, forming a separation-transition-reattachment “bubble.” The size of these bubbles is characterized by the transition mode present in the shear layer. In cases of low freestream turbulence, T-S disturbance waves have been observed within the shear layer, indicating that natural transition is occurring within the bubble. Since the natural transition process occurs much slower than other modes, the length of these bubbles can be significantly longer than desired [28]. Under conditions of increased freestream turbulence, transition may occur much more rapidly, leading to relatively short bubble lengths.

2.3 Transition Models

This section provides a brief sampling of models that have been developed for predicting transitional flows.

2.3.1 Direct Numerical Simulation and Large-eddy Simulation

Although computationally expensive, DNS does offer a means for fully-resolving turbulent boundary layers using the Navier-Stokes equations of fluid flow. DNS is most commonly applied to relatively simple flows to better understand turbulence dynamics. Kalitzin et al. [14] have recently used DNS to simulate the fully-turbulent flow around a low-pressure turbine (LPT) blade. The objective was to develop a set of reference data
for use in the validation of RANS based models. With turbulent free inlet conditions, boundary-layer transition was well predicted on the turbine blade surface. Rist and Fasel [36] have reported the development of a numerical method for solving a vorticity-velocity formulation of the Navier-Stokes equations, in which transport equations are applied for vorticity and Poisson-type equations are solved for the components of velocity. The controlled transition experiments of Klebanoff et al. [23] were simulated and shown to be in agreement with the experimental data as well as linear stability theory.

In an effort to reduce the computational load of a direct simulation, Large Eddy Simulation (LES) techniques are currently being aggressively researched. LES methods fully resolve the lower frequencies of turbulent fluctuations in the energy spectrum and model the higher frequencies. Yang and Voke [57] have reported on LES for a flat plate with a semicircular leading edge, causing the flow to initially separate. Transition to turbulence was numerically resolved in the shear layer. The Kelvin-Hemholtz instability mechanism initiated minuscule two-dimensional disturbances in the flow field. Eventually, these small disturbances developed into three-dimensional fluctuations. Development of turbulent flow was achieved just prior to reattachment and was followed by the rapid development of a fully-turbulent boundary layer upon reattachment. However, development of the log law velocity profile was significantly delayed.

Ducros et al. [10] have applied LES to simulate the complete transition of a flat plate boundary layer. Results have shown the presence and breakdown of T-S instability waves prior to transition. Although the results correspond qualitatively to the findings of
Klebanoff et al. [23], the skin friction coefficient is under predicted in the simulations. The authors attribute this to low grid resolution in the near-wall region and the high dissipation of the sub-grid turbulence model used. Despite these discrepancies, LES offers a valuable alternative to the high computational costs associated with DNS.

### 2.3.2 \( e^n \) Modeling

The \( e^n \) method was developed based on the ideas of linear stability theory and breakdown of instability waves from the start of the boundary layer to the point of transition. When the T-S waves in the pretransitional boundary layer have been amplified by a factor \( e^n \), transition is said to initiate. The value of \( n \) is typically within the range of 7-11 [15]. Malik [26] has evaluated the performance of this method for hypersonic flow over a sharp-nosed cone. The location of transition was well predicted with \( n = 10 \), and the \( e^n \) method was recommended for cases with low freestream disturbances.

An empirical correlation-based model and \( e^n \) model based on linear stability theory have been compared by Johansen and Sorensen [17]. The transition models are coupled with a two-equation \( k-\omega \), fully-turbulent model. The empirical model simply determines the transition location based on a data correlation for transition and momentum thickness. The \( e^n \) model begins transition when the perturbation amplitude of the dominant instability has grown by a factor of \( e^n \), where \( n \) was determined to be 9 based on empirical correlations to experimental data. The transition region for each of the models is addressed through the use of an intermittency (or scaling) factor on the eddy viscosity. Each of the models was compared to experimental data for drag and lift on two
airfoil geometries. Results using only the fully-turbulent $k$-$\omega$ model were also provided for comparison. The authors found the two transition models to be on the same level of accuracy, correlating well with the experimental data. However, the transition location derived from the empirical based model exhibited minor fluctuations that delayed convergence of the solution. Although the accuracy of the two models was reasonably close, the performance of the fully-turbulent $k$-$\omega$ was relatively inaccurate, indicating the importance of modeling laminar-turbulent transition.

Crouch and Ng [7] have developed and presented another example of an $e^n$ factor transitional flow model. However, the model was defined as a variable $n$-factor method. The value for $n$ typically varies between applications, introducing the need for repeated calibrations. In contrast, Crouch and Ng elected to allow $n$ to vary as a function of surface roughness and freestream turbulence intensity, extending the applicability of the model and eliminating the need for repeated calibrations between applications. The model focuses on stationary and traveling crossflow instabilities due to the surface roughness characteristics and higher values of freestream turbulence intensity. The resulting values for $n$ over a range of surface roughness levels were compared to experimental data and found to be in good agreement.

### 2.3.3 Correlation-based Modeling

Models based on correlations to experimental data have become quite popular and have been successfully applied in the prediction of transitional flows. Abu-Ghannam and Shaw [1] have developed an empirical correlation for transition based on flat plate
experimental data with no attempt to model the physics of the transition process. The initiation and end of transition is correlated by the momentum-thickness Reynolds number and includes pressure gradient effects. The authors suggest that the behavior of the pressure gradient upstream is more influential on transition than the local value. However, introducing a need for non-local information increases the complexity of the model. The correlation has been shown to perform well in the presence of high freestream turbulence. Mayle [28] has proposed a simple correlation for transition Reynolds number relating the momentum thickness Reynolds number to freestream turbulence:

$$Re_{\theta} = 400Tu^{5/8}$$  \hspace{1cm} (2.10)

A transition model for separated and attached flows has been presented by Praisner et al. [34, 35]. The authors sought to incorporate a more accurate empirical based transition model into a RANS flow solver. In their efforts, the authors compiled databases of experimental data for separated and attached flow cases. For the transition of attached flow, the authors found the onset of transition to be correlated to freestream turbulence quantities (i.e. turbulence intensity and length scale) and the momentum thickness Reynolds number, where the boundary layer momentum thickness is defined as a function of “local” flow variables. From these variables, the criterion for bypass transition is obtained, which is the ratio of turbulent-eddy time scale to the laminar-diffusion time scale. In a separated flow situation, the relevant correlation is based on the boundary-layer pre-separation state; important quantities include the length of the separation bubble as a function of the pre-separation momentum thickness and the
surface distance from stagnation to separation. Since the databases for separated and attached flows are based exclusively on two-dimensional data, the authors state that the applying the models to three-dimensional flow problems may prove insufficient. The correlation models for attached and separated flow transition were coupled with a fully-turbulent, two-equation $k$-$\omega$ turbulence model and were shown to increase the accuracy in predicted LPT airfoil losses over fully-turbulent models.

Roberts and Yaras [37] have proposed a model that correlates the development of turbulent spots with the boundary-layer shape factor. With the shape factor as the key parameter for the correlation, the authors state that the model is able to accurately predict turbulent spot production for separated and attached flows. In addition, the model takes pressure gradients and freestream turbulence into consideration. The model includes modifications that account for the effects of surface roughness on transition start location and on transition length. The proposed model is reported as the first to include the effects of surface roughness on transition. The pretransition and fully-turbulent regions are bridged by an intermittency factor similar to the functions applied in the transition models reviewed above. The initial validation tests of the model indicate proper model response and an improvement over the empirical model of Abu-Ghannam and Shaw.

In order to implement transition correlations into fully-turbulent models, the local $Re_\theta$ must be calculated along the surface of the body. These values are then compared to the correlated value of $Re_1$ and transition initiates when the local $Re_\theta$ is greater than $Re_1$. At this point, a turbulence model is triggered to resolve the fully-turbulent boundary layer. Transition can either be assumed to occur instantaneously or momentum thickness
Reynolds numbers can be defined for the start and end of transition [1]. However, the duration of transition can also be simulated with an intermittency function, ramping up the eddy viscosity to that of fully-turbulent flow.

Menter et al. [30, 24] developed a correlation-based transition model for prediction of transitional flows. In their model, transport equations are solved for the intermittency factor and momentum-thickness Reynolds number. The transport equations do not attempt to model the physics of transition, as in other transition models [51, 52]. However, the intermittency function is used to bridge the gap between the start and end of the transition process. The transport equation for the momentum thickness Reynolds number can be coupled with popular experimental correlations to initiate the transition process. The results are in good agreement with turbine blade and flat plate experiments.

2.3.4 Intermittency Modeling for the Duration of Transition

Edwards et al. [12] have recently documented efforts in coupling a previously developed enstrophy based transition model with the Spalart-Allmaras one-equation turbulence model. The enstrophy-based model addresses the intensity of the T-S, crossflow and bypass transition mechanisms. The transition model is a two-equation transition model, with the first equation representing the development of non-turbulent fluctuating kinetic energy and the second the evolution of non-turbulent eddy viscosity. The applicable time scale is defined as a function of boundary layer displacement thickness and/or surface distance. For determining the transition start location, a turbulent Reynolds number is calculated and transition initiates when the maximum value of this
parameter surpasses unity. The transition model equations are coupled to the Spalart-
Allmaras fully-turbulent model through an intermittency function, which yields fully-
turbulent flow when equal to unity. In order to more accurately resolve transition in
complex geometries (where both shear layers and boundary layers may exist), the
intermittency function is divided into two parts. The first part is a function of surface
distance and the second is a function of spatial coordinates in the boundary layer. For
validation, the model was employed by two N-S flow solvers and simulations were
performed for flat plate, airfoil, cylinder and cone configurations. The model results
agreed reasonably well with the experimental data for skin friction, recovery factor, and
Nusselt numbers. In reference to future work, the authors state that they would like to
include terms that resolve the effects of pressure gradients on transition.

Baek et al. [3] have proposed a modified two-equation $k-\varepsilon$ model for representing
transitional flowfields. In an effort to more completely include the physics of transition,
the flowfield is divided into 3 regions: the pretransition region, transitional region, and
the fully-turbulent region. In the pretransition region, the viscous sublayer dominates the
boundary layer and the relevant velocity and length scales are the square-root of kinetic
energy and the cube of the wall distance. An intermittency function, defined by a
streamwise variation and a wall-normal variation, links the pretransitional region to the
transitional region. This multidimensional definition of the intermittency function is used
to represent the streamwise growth of the developing turbulent spots. The intermittency
function is equal to unity for the fully-turbulent flow region. The model was validated
against flat plate experimental data for freestream turbulence intensities of 1-6%. The
model showed excellent agreement with the reference data for mean velocity profiles, skin friction variation, and shape factor variation. However, the performance of the model was least accurate at the lower values of freestream turbulence intensity due to the dominance of natural transition versus bypass transition.

Another model that employs an intermittency function has been proposed by Schobeiri and Chakka [44]. Their work focuses on the prediction of heat transfer and flow characteristics on turbine blades, taking into account an impinging unsteady wake flow. The intermittency is ensemble-averaged and used to define a relative intermittency. Specifically, the relative intermittency is defined as the local ensemble-averaged intermittency minus the ensemble-averaged intermittency value outside of the wake vortical core (minimum intermittency), normalized by the difference in the ensemble-averaged intermittency value inside the wake vortical core (maximum intermittency) and the minimum intermittency. Using the relative intermittency, the model is able to calculate the local ensemble-averaged intermittency. The model was implemented into the boundary-layer code, TEXSTAN. Validation tests of the model accurately predicted aerodynamic and heat transfer quantities for a cascade of turbine airfoils.

Steelant and Dick [47] have provided a transport equation for a turbulence weighting factor, $\tau$. This turbulence weighting factor is defined as the sum of the multidimensional intermittency factor for turbulent spot growth and another variable representing the diffusion of freestream turbulent eddies into the boundary layer. Unlike other models that implement the intermittency factor, this approach allows for determination of the correct freestream turbulent flow quantities before transition occurs.
The conditional averaged Navier-Stokes equations are separated into laminar and turbulent flow equations that are functions of the $\tau$ term. A transport equation is defined for $\tau$ and includes terms for convection, diffusion, and dissipation of $\tau$. In the determination of the transition onset location, the authors define a new correlation based on freestream turbulence quantities and a turbulent spot growth parameter. The model was validated using zero pressure gradient flat plate test cases and flat plate cases with a pressure distribution similar to that of an aft loaded turbine blade. The model showed reasonable agreement with the reference data for skin friction and shape factor variations. In the future, the authors suggest that compressibility effects must be considered to obtain more accurate results.

### 2.3.5 Laminar Kinetic Energy

Based on the idea of “laminar-kinetic-energy” proposed by Mayle and Schulz [29], Walters and Leylek [51] have recently proposed a single-point, RANS based model for transition. The development of the new model was based on the evolution of this laminar-kinetic-energy. Streamwise fluctuations begin developing in the laminar boundary layer due to the freestream turbulence. An additional transport equation was developed to model the development of laminar-kinetic-energy and must be solved in conjunction with the two other transport equations of the two-equation turbulence model. In order to include the effects of natural transition, a separate “production” term is included in the laminar-kinetic-energy equation. Although the flow is fully-turbulent after the prediction of transition onset, the model still predicts minor amounts of laminar-
kinetic-energy in the viscous sublayer, which agrees with experimental observations. The model was implemented into the commercially-available flow solver, FLUENT. Test simulations were performed for a channel flow, flow over a flat plate and a highly-loaded turbine airfoil. The model predicted transition and demonstrated the proper response to freestream turbulence quantities. Lardeau et al. [25] have also developed a model for transitional flows that employs an additional transport equation for laminar-kinetic-energy. The model addresses the transition length with an intermittency factor based on the correlation of Dhawan and Narasimha [8].
CHAPTER III
NEW MODEL DEVELOPMENT

3.1 Objective

Several approaches have been presented in the literature for prediction of transition onset. Currently, the two most common approaches resolve transitional flow effects by: 1) coupling empirical correlations for transition onset to fully-turbulent eddy-viscosity turbulence models or 2) prescribing additional transport equations in an attempt to model the physics of the transition to turbulent flow. The first employs commonly used engineering correlations from experimental data to predict the location of transition inception. The correlations are coupled to the fully-turbulent models by assuming either “instantaneous” transition or using some form of an intermittency function to “ramp” the value of eddy viscosity. The experimental data used to derive the empirical relationships are often based on relatively simple flow problems, increasing the difficulty of implementation to 3-D solvers. Furthermore, the location of transition onset is usually based upon non-local parameters, such as boundary-layer momentum thickness and the distance downstream from the leading edge. Determination of these parameters increases the computational load, especially within massively parallel-computing environments. The latter method solves additional transport equations for the physics of transition in parallel with the transport equations of the fully-turbulent model. Although this methodology has been reasonably successful, the complexity of the model is increased,
making the transition-sensitive model difficult to implement into existing commercially
available flow solvers. The total computational expenses are also increased with the
addition of these equations.

The objective of the current study is to develop a transition sensitive turbulence
model that can be easily implemented into currently available flow solvers. The new
model is developed within the framework of the RANS based, two-equation turbulence
models available within many commercially used flow solvers. By developing the new
model without the use of empirical correlations or additional transport equations for
transition physics, the model maintains the same level of complexity as the fully-
turbulent model and implementation is relatively simple. In addition, the new model
should demonstrate the proper response to changes in the freestream turbulence
quantities, such as freestream turbulence intensity and turbulent length-scale.

### 3.2 Derivation

The new model is developed within the framework of Reynolds Averaging. The
RANS equation for incompressible conservation of momentum and the Boussinesq form
of the eddy-viscosity are given in Equations (2.3) and (2.5), respectively:

\[
\frac{\partial}{\partial x_j} \left( \rho U_i U_j \right) = -\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left( 2\mu S_{ij} - \rho u_i u_j \right)
\]  

(2.3)

\[
\rho u_i u_j - \frac{2}{3} \rho u_k u_k \delta_{ij} = -2\mu_t S_{ij}
\]

(2.5)

The new model has been developed within and implemented into the standard
two-equation k-\omega fully-turbulent model used by many commercially available flow
solvers. Transport equations are solved for turbulent kinetic energy, $k$, and the inverse time scale, $\omega$. The forms of these equations adopted here are given as:

$$\frac{\partial}{\partial x_j} \left( \rho U_j k \right) = P_{\text{total}} - \rho \omega k - \varepsilon_{\omega} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$

(3.1)

$$\frac{\partial}{\partial x_j} \left( \rho U_j \omega \right) = P_{\omega} + R_{\omega} - E_{\omega} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_{\omega}} \right) \frac{\partial \omega}{\partial x_j} \right]$$

(3.2)

Transitional flow effects are incorporated into the new model by redefining the mathematical forms for the following terms in Equation (3.1) and Equation (3.2): $\mu_T$, $P_k$, $E_k$, $\sigma_k$, $P_\omega$, $R_\omega$, $E_\omega$ and $\sigma_\omega$. The goal is to redefine these terms to reproduce the appropriate behavior for both transitional and fully-turbulent flows. Since only the forms of these terms are modified, the new model maintains the same level of complexity as a standard $k$-$\omega$ model.

3.2.1 Fully-turbulent Region

In the region of fully-turbulent flow, the new model uses the standard $k$-$\omega$ turbulence model employed by most commercially available CFD flow solvers. However, the eddy-viscosity is redefined in the new model as:

$$\mu_T = \mu_C f_{w}^{2} \frac{4}{\omega} \rho \frac{k}{\omega}$$

(3.3)

Schumann [45] has shown that modeling the Reynolds shear stress with constant-coefficient eddy-viscosity models yields non-physical results in regions of rapid strain rate. Therefore, Shih et al. [46] developed a new formulation of the eddy-viscosity
coefficient, $C_\mu$, to impose the realizability constraint. In the present study, the coefficient $C_\mu$ has a functional form similar to that of Shih et al. [46]:

$$C_\mu = \frac{1}{A_0 + A_s f_w^3 \frac{\Omega}{\omega}} \tag{3.4}$$

The new model also contains modifications to address the kinematic-wall and viscous-wall effects present in the near-wall region of the flow. The kinematic-wall effect has been shown to scale with the large-eddy length scale [31] and its influence on turbulence structure results in varying production and dissipation dynamics of the turbulent flow [49]. The new model addresses this kinematic wall effect through the use of an “effective” turbulence length scale near the wall. This effective length scale is limited such that:

$$\lambda_{\text{eff}} = \text{MIN}[C_L d, \lambda_T] \tag{3.5}$$

where $d$ is the normal distance to the wall. $\lambda_T$ is the turbulent length scale and is defined as:

$$\lambda_T = \frac{\sqrt{k}}{\omega} \tag{3.6}$$

The kinematic wall effect is imposed on the eddy-viscosity (Equation (3.3)) via an inviscid wall damping function:

$$f_w = \frac{\lambda_{\text{eff}}}{\lambda_T} \tag{3.7}$$

Therefore, the value of eddy viscosity is reduced in the near-wall region when $\lambda_T$ is significantly greater than the wall distance.
The viscous-wall effect is imposed on the eddy-viscosity (Equation (3.3)) through the following damping function, where simple exponential damping is defined:

$$f_{\mu} = 1 - \exp\left(\frac{Re_e}{A_{\mu}}\right)$$  \hspace{1cm} (3.8)

The relevant scaling parameter for the viscous wall effect is assumed to be proportional to the length scale of the smallest (Kolmogorov) turbulent eddies. The Kolmogorov length scale is approximated as:

$$\eta_k = \left(\frac{\nu^3}{k \omega}\right)^{\frac{1}{4}}$$  \hspace{1cm} (3.9)

$Re_e$ represents the ratio of the wall distance, $d$, to the Kolmogorov scales:

$$Re_e = \frac{\frac{4}{3} \frac{1}{k^\frac{1}{3}} \frac{1}{\omega^\frac{1}{3}}}{\nu}$$  \hspace{1cm} (3.10)

In an effort to eliminate any over-prediction in turbulent production for areas of large strain, a limiting function is imposed on the production term in Equation (3.1) similar to that of Kato and Launder [21]. The production limiter is included to account for the stagnation point anomaly, as reported by Durbin [11]. Without this modification, large amounts of turbulent kinetic energy would be initially captured within the boundary layer at the stagnation point, increasing the difficulty to accurately predict the location of transition. Kato and Launder [21] proposed that the mean strain rate magnitude in the production term should be replaced by the mean rotation rate. However, this implies the possibility of turbulent production in strain-free regions of the flowfield, which is non-physical. The modified form used here addresses this by replacing the mean-rotation rate
in the Kato and Launder limiter with an effective mean strain rate. The effective mean-strain rate is defined as the minimum of the mean-rotation rate and mean-strain rate. The exact full form of the production term in Equation (3.1) is given below, with additional modifications that incorporate transitional effects.

The $\omega$ production term in Equation (3.2) maintains the standard form:

$$P_\omega = C_{\omega f} \frac{\omega}{k} \rho_p$$  \hspace{1cm} (3.11)

The $k$ destruction term in Equation (3.1) is formulated to yield the correct wall-limiting behavior for $k$:

$$\varepsilon_w = \frac{2\mu_k}{d^2}$$  \hspace{1cm} (3.12)

To improve accuracy in the near-wall region, the destruction term in the $\omega$ equation (Equation (3.2)) has been modified to include the inviscid wall effect:

$$E_\omega = f_w \frac{4}{3} C_{\omega f} \omega^2$$  \hspace{1cm} (3.13)

3.2.2 Growth of Pretransitional Disturbances

Section 2.2 presented the mechanisms that eventually lead to the development of fully-turbulent boundary layers. When the flow reaches a critical point along the surface, fluctuating disturbances begin to develop within the laminar boundary layer. Further downstream, these disturbances are amplified and turbulent spots form. Once large enough, the turbulent spots develop into fully-turbulent flow. The model presented here includes the effects of Klebanoff [23] and Tollmien-Schlichting fluctuating disturbances.
Klebanoff disturbance modes are associated with bypass transition and are characterized by algebraic growth. However, the algebraic growth of the Klebanoff modes is not explicitly included in the new model. Klebanoff disturbance modes are implicitly incorporated in the new model by assuming that the eddy viscosity in the momentum equations is equal to the fully-turbulent value, but damped in the turbulence production term as discussed below. This incorporates the effect of freestream turbulence in on the pretransitional boundary layer in the same manner as Praisner and Clark [34], and has been shown to yield accurate results in the pre-transitional region.

Tollmien-Schlichting disturbance waves are associated with natural transition and are characterized by exponential growth. The exponential growth of T-S disturbances is modeled by modifying the production term in Equation (3.1) using the following damping function:

\[ f_{TS} = 1 - \exp \left[ - \left( \frac{\beta_{TS}}{C_{TS,1}} \right)^2 \right] \]  
\[ \beta_{TS} = \text{MAX} \left( \frac{d^2 \Omega}{\nu} - C_{TS,2,0} \right) \]

where the constants \( C_{TS1} \) and \( C_{TS2} \) can be calibrated to experimental data for natural transition. It is apparent that the damping function controlling growth of T-S disturbances is non-zero only for values of the so-called vorticity Reynolds number \( (d^2 \Omega/\nu) \) greater than a threshold value. For flat-plate boundary layers, it has been shown that the vorticity Reynolds number is proportional to the momentum-thickness Reynolds number \( (Re_\theta) \) in the pretranstional region [24, 30]. The influence of the exponential growth of Tollmien-Schlichting disturbances on production is modeled as:

\[ 26 \]
\[ P_{TS} = 0.15 \rho k \Omega (1 - f_{p1}) f_{TS} \]  

where \( f_{p1} \) is defined in the next section. Equation (3.16) will be combined with the complete production damping function in the next section as well.

### 3.2.3 Transition Inception

Recent studies have identified the phenomenon known as “shear sheltering.” This effect is characterized by the dampening of turbulence dynamics in regions of high vorticity. The shear layer hinders the ability of disturbances to penetrate the sheared region [16]. The effect of shear sheltering is included in the current model to yield transition behavior. Essentially, turbulence production in the viscous sublayer is suppressed because the ratio of the turbulent eddy time-scale to the molecular diffusion time-scale has not reached the critical value for non-linear disturbance amplification. The effects of shear sheltering on disturbance breakdown are incorporated into the new model with the following damping function:

\[
f_{p1} = \exp \left[ - \left( \frac{C_\beta \Omega}{f_{Re_T} k} \right)^2 \right]
\]

\[
f_{Re_T} = 1 - \exp \left[ - \left( \frac{k}{C_{Re_T} \nu \Omega} \right)^{3/2} \right]
\]

The turbulent eddy time-scale is defined to be the production time scale and, within the boundary layer, is proportional to the mean vorticity. The molecular diffusion time scale is approximated by considering the length scale of fluctuations associated with
turbulence in the boundary layer. The streamwise energy associated with wall-normal fluctuations, as a function of this length scale, can be approximated as:

\[ \bar{u}_i u_i \approx \left( \ell_v \cdot \frac{\partial U_1}{\partial x_2} \right)^2 \]  \hspace{1cm} (3.19)

where \( \ell_v \) is a length scale associated with the perturbations. The streamwise energy is proportional to the turbulent kinetic energy, \( k \). Therefore, the molecular diffusion time-scale is equal to \( \ell_v^2 / \nu \), or \( k/(\nu \Omega^2) \). The production time scale is defined to be the relevant time scale associated with the non-linear break down and is proportional to the mean vorticity. As a result, this critical time-scale ratio is modeled in Equation (3.16) as \( k/(\nu \Omega) \). When this ratio is low, shear sheltering prevents the development of turbulence dynamics and transition occurs when this time-scale reaches a critical value.

For natural transition, the ratio of the Tollmien-Schlichting time-scale and the molecular diffusion time-scale is used to derive the Tollmien-Schlichting damping function given in the previous section as Equation (3.14). The T-S waves are defined to be proportional to the wall normal distance and the relevant time-scale for molecular diffusion is \( (d^2)/\nu \). Therefore, the critical time-scale ratio for natural transition is \( (d^2 \Omega)/\nu \), as shown in Equation (3.15). As mentioned above, this time-scale ratio has also been used in the correlation-based transition model of Menter et al. [24, 30].

The effects of shear sheltering are incorporated into the turbulent kinetic energy production term as:

\[ P_k = MIN\left( f_{p1} \mu_r \Omega^2, 0.16 \rho d^2 S^3 \right) \]  \hspace{1cm} (3.20)
This production term is combined with the Tollmien-Schlichting production term to yield the total production term:

\[ P_{\text{total}} = P_k + P_{TS} \]  \hspace{1cm} (3.21)
CHAPTER IV
SOLVER IMPLEMENTATION

The new model has been developed for easy implementation into existing CFD flow solvers that employ two-equation eddy-viscosity turbulence models. In this study, the model has been implemented into the commercially-available flow solver, FLUENT, and the Mississippi State University (MSU) Computational Simulation and Design Center (SimCenter) developed flow solver, U²NCLE.

4.1 FLUENT

For initial validation of model performance, the model was implemented into FLUENT version 6.1.22 (Fluent, Inc.). Incorporating the modified two-equation \( k-\omega \) turbulence model into the solver was accomplished through the use of user-defined subroutines. The principal variables of the standard two-equation model, \( k \) and \( \omega \), were redefined in the form of user-defined scalars. User-defined function (UDF) subroutines were defined to provide the needed source terms, effective diffusivities, and the effective turbulent viscosity. The use of UDF’s for model implementation has previously been validated by SimCenter personnel. UDF’s were developed for the standard \( k-\varepsilon \) model and the realizable \( k-\varepsilon \) model and compared to the solutions given by FLUENT internal models. The UDF versions were found to produce results identical to those generated by
the available models in FLUENT. The values for the model constants are given in Table A.1. The primary variables, $k$ and $\omega$, were equal to zero at the wall.

4.2 U$^2$NCLE

In addition to the FLUENT implementation, the new model was implemented into the MSU in-house flow solver, U$^2$NCLE. Since the new model is based on a fully-turbulent, low-Reynolds number $k$-$\omega$ model, the functional form of the source terms was modified in the existing U$^2$NCLE $k$-$\omega$ turbulence model. The U$^2$NCLE implementation is applicable for both incompressible and compressible flow regimes. The values for the model constants are given in Table A.2. The primary variables, $k$ and $\omega$, were equal to zero at the wall.
CHAPTER V

FLUENT TEST CASES AND RESULTS

The model developed within Chapter III was initially implemented and validated by Walters (Mississippi State HPC²), using the commercially-available flow solver, FLUENT. In transition studies, the flat plate is the canonical geometry for model validation and is used in this study to demonstrate the sensitivity of the model to freestream-turbulence quantities. Results are provided for the distribution of the skin friction coefficient over a sampling of freestream-turbulence intensities. In addition, results are compared to an experimental data set that is recognized as the standard for transition-model validation. Simulations were also performed for a two-dimensional circular cylinder in a cross-flow to evaluate the capability of the model to predict separation-induced transition. The resulting drag coefficients are compared to experimental data provided in the literature.

5.1 Incompressible Flat-plate Validation

Model performance was evaluated for a zero-pressure gradient boundary-layer flow, over an array of freestream turbulence intensities and length scales. The simulations were performed to illustrate proper model response to different freestream quantities. It is important to note that for these simulations, the computational inlet was placed upstream
from the leading edge of the plate, at a position corresponding to $Re_x = -10^5$. The boundary layer is allowed to develop naturally, as opposed to starting the simulation within the boundary layer. Other studies presented in the literature have suggested that transition prediction is sensitive to the inlet profile of turbulence quantities, when simulations are “started” within the boundary layer itself [39, 40, 53, 56]. Figure 5.1 is a schematic illustrating the flow conditions for the flat plate simulations.

![Figure 5.1](image)

Schematic for flat-plate flow conditions

Figure 5.2 shows the skin friction coefficient versus streamwise Reynolds number, for seven different values of freestream-turbulence intensity. Freestream-turbulence conditions are given for each case in Table 5.1. Limiting curves for laminar and turbulent flows are provided on the plot, which are defined by:

$$C_f = 0.64 Re_x^{-1/2} \quad \text{[Laminar]} \quad (5.1)$$

$$C_f = 0.05924 Re_x^{-1/5} \quad \text{[Turbulent]} \quad (5.2)$$

As expected, the transition location moves upstream as turbulence intensity is increased, and the skin friction coefficient is well predicted for both the fully-laminar
[Equation (5.1)] and fully-turbulent [Equation (5.2)] regions. The model demonstrates the proper response to increasing freestream turbulence intensity.

Table 5.1

Flat Plate Leading-edge Turbulence Quantities

<table>
<thead>
<tr>
<th>Tu (%)</th>
<th>Re_T</th>
<th>Re_L</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.02</td>
<td>100</td>
<td>10^7</td>
</tr>
<tr>
<td>0.2</td>
<td>100</td>
<td>10^7</td>
</tr>
<tr>
<td>0.5</td>
<td>100</td>
<td>10^7</td>
</tr>
<tr>
<td>1.0</td>
<td>1000</td>
<td>10^7</td>
</tr>
<tr>
<td>2.0</td>
<td>1000</td>
<td>10^6</td>
</tr>
<tr>
<td>4.0</td>
<td>1000</td>
<td>10^6</td>
</tr>
<tr>
<td>8.0</td>
<td>1000</td>
<td>10^6</td>
</tr>
</tbody>
</table>

The FLUENT model was further validated using the experimental data available from the ERCOFTAC database [6]. These data represent the influence of turbulence quantities on the zero pressure gradient flat-plate flows. Table 5.2 lists the freestream-turbulence quantities for cases T3A-, T3A, and T3B, representing nominal freestream-turbulence intensity values of 1%, 3% and 6%, respectively. The ERCOFTAC database provides information regarding the decay of freestream turbulence for the nominal intensity values. Therefore, combinations of freestream turbulence intensity and turbulent...
Reynolds numbers have been selected that satisfy this rate of decay. Figure 5.3 illustrates the ability of the model to duplicate the trends provided in the experimental data for the distribution of skin friction.

![Graph showing skin friction coefficient (Cf) vs. Reynolds number (Rex) for different Tu values.](image)

**Figure 5.2**

$C_f$ vs. $Re_x$ for FLUENT validation test cases

**Table 5.2**

ERCOFTAC Leading-edge Turbulence Quantities

<table>
<thead>
<tr>
<th>Case</th>
<th>$Tu$ (%)</th>
<th>$Re_T$</th>
</tr>
</thead>
<tbody>
<tr>
<td>T3A-</td>
<td>0.858</td>
<td>113.29</td>
</tr>
<tr>
<td>T3A</td>
<td>2.956</td>
<td>138.95</td>
</tr>
<tr>
<td>T3B</td>
<td>5.962</td>
<td>1097.59</td>
</tr>
</tbody>
</table>
5.2 Incompressible Cylinder Test Case

According to the flat-plate simulations in FLUENT, the new model responds appropriately to the influence of freestream turbulence parameters in attached boundary layers. However, the ability of the model to demonstrate the proper behavior for separated flows is equally important. The circular cylinder in a cross-flow test case not only includes the effects of flow separation, but the effects of streamwise pressure gradients due to surface curvature. Figure 5.4 is a schematic illustrating the flow conditions for the cylinder in a cross-flow.

Research has shown that there exists a critical value of $Re_D$ where the separation-attachment behavior changes under the influence of transition, also known as the “drag crisis.” This value has been found to be within the range of $Re_D = 10^5$ and $Re_D = 10^6$. 

Figure 5.3

$C_f$ vs. $Re_x$ for FLUENT ERCOFTAC test cases
When $Re_D$ is determined to be less than the critical value $Re_{D,\text{crit}}$, the boundary layer separates at a location approximately $85^\circ$ from the stagnation point and transition occurs within the shear layer. The boundary layer remains unattached from the cylinder surface. When $Re_D$ is greater than the critical value, the boundary layer separates at a location approximately $90^\circ$ from the stagnation point, quickly transitions and reattaches to the surface. At approximately $120^\circ$, the turbulent boundary layer re-separates from the surface and remains unattached.

Figure 5.5 shows contours of turbulence intensity for the sub-critical regime ($Re_D < Re_{D,\text{crit}}$), illustrating the characteristic single separation behavior. For the supercritical case ($Re_D > Re_{D,\text{crit}}$), the correct behavior is confirmed in Figure 5.6. The boundary initially separates and transition occurs shortly thereafter within the shear layer. The figure also illustrates the reattachment of the boundary layer, followed by the final separation at the correct location. A ratio is calculated between the drag coefficients for each flow regime and the supercritical drag was determined to be $18\%$ of the subcritical value. This value correlates well with a value of $20\%$ reported from experiments [2].
The initial results from the implementation of the model in the FLUENT flow solver demonstrate the ability of the model to properly resolve transitional flow effects. It is apparent from the flat-plate results that the model appropriately reproduces the
transition process and the proper trends in the presence of natural and bypasses transition mechanisms. Additionally, the model performed well in the presence of separation-induced transition, as shown in the results for the circular cylinder flow cases. Results simulated with the new model show excellent agreement with the distribution of shear stress in the ERCOFTAC transition experiments and in the prediction of the “drag crisis” of the circular cylinder. Further results from the initial validation of the model can be located in Ref. [19], including information regarding varying turbulent length scale and comparisons of transition onset locations with models based upon experimental correlations.
CHAPTER VI
U2NCLE TEST CASES AND RESULTS

The new model was implemented into the standard $k-\omega$ turbulence model provided within U2NCLE. Similar test cases to those performed in FLUENT were used in the validation and calibration of the model constants in the U2NCLE version. In the current study, the compressible version of the flow solver was calibrated and applied in the remaining test cases.

6.1 Flat-plate Validation

The zero-pressure gradient flat plate represents the canonical case for transition prediction and was used to validate the response of the model to freestream-turbulence quantities. Freestream-turbulence intensity was varied from 0.02% to 8%. Information regarding turbulent length scale is not often provided with experimental data. However, a length-scale value equivalent to $Re_T = 100$ has been selected based upon previous experience of model behavior in U2NCLE. This value of length scale will be applied for calibrations and applications to experimental data provided later in this report. Figure 6.1 and Figure 6.2 present the grid used for the flat plate computations. Grid independent solutions were found for values of $y^+ = 1$ and wall-normal stretching ratios in the structured near-wall grid layer less than 1.2.
Figure 6.1

Grid (flow domain) used for flat-plate simulations

Figure 6.2

Grid (plate) used for flat-plate simulations

Figure 6.3 shows the skin friction coefficient versus streamwise Reynolds number. Leading edge conditions were the same as those defined in Table 5.1. The model
demonstrates the appropriate behavior across the given range of turbulence intensities. As the turbulence intensity is increased, the location of transition onset shifts upstream. Furthermore, the skin friction coefficient was well predicted for both the fully-laminar and fully-turbulent regions.

![Graph showing CF vs. Rex for U^2 NCLE validation test cases](image)

**Figure 6.3**

\( C_f \) vs. \( Re_x \) for U^2 NCLE validation test cases

In the current study, natural transition is defined to occur for values of freestream turbulence intensity below 1%. Values of turbulence intensity greater than 1% are associated with bypass transition. The model has constants that govern the behavior of each. The results from the two lowest values of intensity, 0.02% and 0.2%, were used to calibrate the model constant, \( C_{TS} \), that primarily influences natural transition, with
experimental data [38]. These experimental data provide information regarding transition initiation and completion Reynolds numbers for turbulence intensities ranging from 0.02% to 0.36%. Although the new model does not have the capability to control the length of the transition zone, the predicted location of instantaneous transition within this zone is deemed sufficient. The resulting skin friction plots are given in Figure 6.4. The 0.02% case is predicted to transition at a location corresponding to $Re_x = 3.6 \times 10^6$, within the zone defined by $Re_x = 2.8 \times 10^6$ and $3.9 \times 10^6$. The 0.2% case was predicted to transition at a location corresponding to $Re_x = 2.4 \times 10^6$, within the zone defined by $2.2 \times 10^6$ and $3.6 \times 10^6$.

![Figure 6.4](image)

$C_f$ vs. $Re_x$ for $U^2$NCLE test cases with $Re_T = 100$; Natural transition calibration

For calibration of the model constant that primarily influences bypass transition, $C_{\beta}$, simulations were performed to reproduce the results provided in the ERCOFTAC
database. Test conditions were previously outlined in Table 5.2. Test cases T3A and T3B were evaluated for a plate Reynolds number \((Re_L)\) of \(10^6\); \(Re_L = 10^7\) for the T3A simulation. Although experimental conditions indicate incompressible flow, the compressible flow solver was applied at the lower limit of compressible flow (i.e., \(Ma = 0.3\)) to approximate incompressible conditions. Figure 6.5 illustrates the ability of the model to predict the skin friction trends given in the experimental data. It is apparent from the plot that the onset of transition is well-predicted for the ERCOFTAC experimental data.

![Figure 6.5](image)

*Cf vs. Rex for U^2NCLE ERCOFTAC test cases*

### 6.2 Cylinder Test Cases
The U$^2$NCLE version of the new model has been shown to perform well in predicting the skin friction distributions for the flat-plate simulations detailed in the previous section. Similar to the circular cylinder in a cross-flow simulation performed in FLUENT, the ability of the model to resolve separation-induced transition was evaluated. The geometry used for this case is a circular cylinder with a rounded tip. Flow direction is parallel to the axis of the cylinder. Figure 6.6 and Figure 6.7 present the grid used for the cylinder axial-flow computations. Grid independent solutions were found for values of $y^+ = 1$ and wall-normal stretching ratios less than 1.2.

![Grid (flow domain) used for circular cylinder axial-flow simulations](image)

Figure 6.6

Grid (flow domain) used for circular cylinder axial-flow simulations
Two simulations were completed and evaluated for the cylinder test case. For each simulation, $Ma = 1.0$ and a turbulent length scale corresponding to $Re_T = 100$ was chosen. The unit Reynolds number with respect to the cylinder length was $Re_L = 10^6$. The cylinder test case was evaluated for two values of freestream turbulence intensity, 0.2% and 2.0%. This is a qualitative study, not a quantitative study; no experimental data are provided for comparison to the computational results. The objective of this test case is only to show that the model responds in a physically realistic manner to boundary-layer separation.

Figure 6.8 provides the skin friction coefficient distribution along the length of the cylinder for each value of $Tu$. Figure 6.9 and Figure 6.10 show the contours of skin friction coefficient and eddy-viscosity for each value of $Tu$, 0.2% and 2.0%, respectively. Skin friction contours are plotted on the cylinder surface, and the variation in eddy-viscosity is given in the flowfield. The figures demonstrate proper model behavior in the presence of a separated boundary layer. Once the flow separates over the leading edge of the cylinder, transition occurs within the shear layer and the flow reattaches to the cylinder surface as a turbulent boundary layer.
Figure 6.8

$C_f$ vs. $Re_x$ for the cylinder axial-flow simulations; $Re_T = 100$ and $Ma = 1.0$

Figure 6.9

Contours of $C_f$ and $\mu_t$; $Tu = 0.2\%$, $Re_T = 100$ and $Ma = 1.0$
Figure 6.10

Contours of $C_f$ and $\mu_t$; $Tu = 2.0\%$, $Re_T = 100$ and $Ma = 1.0$

6.3 Cone Test Cases

Transitional flow over missile nose cone geometries is a problem of considerable interest to the aerospace design community. The accurate resolution of boundary layer transition for these geometries with CFD tools has become increasingly important and desired for aerodynamic and aerothermal engineering design. In an effort to provide practical predictive capability using general purpose CFD tools, the new model has been applied to the case of transitional flow on sharp-nosed cones. As mentioned above, turbulence models for transitional flow have been previously developed based on correlations of experimental data for transition onset location with parameters such as Reynolds number and Mach number. However, the transition process begins with the amplification and breakdown of freestream disturbances. These disturbances are
influenced by many factors such as freestream turbulence, Mach number, freestream pressure gradients, surface curvature and roughness. As a result, a considerable assortment of parameters influences the transition location of the boundary layer on the conical surface. The accuracy will be restricted with the use of these transitional flow models that correlate the location of transition onset with only a sampling of these parameters [41].

Freestream turbulence or “noise” appears to be one of the dominating factors in predicting boundary layer transition. Several references [5, 42, 43] can be found in the literature that report the sensitivity of transition to freestream turbulence quantities and the noise associated with ground-test facilities. Even though much experimental data have been collected using low-noise wind tunnels, the disturbances influencing transition originating in ground-test facilities can be quite dissimilar from the disturbances present in flight tests. As a result, these disturbances tend to produce premature and unreliable data for onset of transition. Despite the difficulties associated with reproducing real-flight disturbance environments using wind tunnels, the new model exhibits the proper behavior when influenced by freestream turbulence quantities.

The calibration of the model constants was based solely on the experimental data for the zero pressure gradient flat plate cases detailed above. However, one of the objectives for this study is to evaluate the ability of the model to reproduce the correct transitional flow behavior for conical flow applications. Three sets of experimental data for transitional flows involving conical geometries have been identified for simulation. Figure 6.11 is a schematic illustrating the flow conditions for the cone simulations.
Figure 6.12 and Figure 6.13 present the grid used for the conical flow geometry computations. Grid independent solutions were found for values of $y^+ = 1$ and wall-normal stretching ratios less than 1.2. An example of the skin friction contours predicted for a conical flow simulation is provided in Figure 6.14. Analysis of the computational results was performed using the flow visualization tools available in the post-processing software EnSight Gold version 8.2 (CEI, Inc.).

![Figure 6.11](image)

**Figure 6.11**

Schematic for sharp-nosed cone flow conditions

### 6.3.1 10° Cone Simulations

The first conical-flow dataset serves to demonstrate model response for varying freestream-turbulence quantities. The geometry is a 10° half-angle sharp-nosed cone. Mach number is varied from 1.05 to 1.44. However, information detailing freestream turbulence quantities is not provided in the reference [33].
Figure 6.12
Grid (flow domain) used for conical geometry simulations

Figure 6.13
Grid (cone tip) used for conical geometry simulations
Values for freestream turbulence intensity and length scale have been selected based on similar experimental datasets provided in the literature [5, 13]. The test matrix for the first set of cone cases is outlined in Table 6.1.

Figure 6.15 illustrates the model response to increasing freestream turbulence intensity. As the turbulence intensity is increased, the location of transition moves upstream. This behavior agrees with the results provided for increasing turbulence intensity in the flat plate calibrations.

Figure 6.16, Figure 6.17, and Figure 6.18 show the models response to increasing turbulent length scale. As the turbulent Reynolds numbers are increased, the location of transition onset moves downstream. As the length scale becomes quite large, the location of transition moves downstream. This behavior is in agreement with observations reported in the literature [34].
Table 6.1

10° Cone Test Matrix

<table>
<thead>
<tr>
<th>Ma</th>
<th>Tu (%)</th>
<th>Re₁</th>
<th>Re₁ (experimental)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.05</td>
<td>0.01</td>
<td>10</td>
<td>4.7 x 10⁶</td>
</tr>
<tr>
<td></td>
<td>0.10</td>
<td>10</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.50</td>
<td>10</td>
<td></td>
</tr>
<tr>
<td>1.30</td>
<td>0.50</td>
<td>100</td>
<td>3.2 x 10⁶</td>
</tr>
<tr>
<td>1.44</td>
<td>0.50</td>
<td>100</td>
<td>4.0 x 10⁶</td>
</tr>
</tbody>
</table>

Figure 6.15

Cₚ vs. Reₓ for the 10° cone, with varied Tu, Reᵦ = 100 and Ma = 1.05
However, the case of $Tu = 0.5\%$ and $Re_T = 10$ in Figure 6.18 does not agree. According to the observations in the literature, the $Re_T = 10$ case should transition prior to the other two cases, and the reason for this discrepancy is unknown at this time. The consistent behavior when $Re_T = 100$ further supports the assumption of this value for model calibration and all remaining test cases.

![Figure 6.16](image)

Figure 6.16

$C_f$ vs. $Re_x$ for the 10° cone, with $Tu = 0.01\%$, varied $Re_T$ and $Ma = 1.05$

Figure 6.19 shows the response of the model to increasing Mach number. Although the results do not exactly predict the transition location for each of the Mach numbers given by the experimental data, the predictions for transition location are
$C_f$ vs. $Re_x$ for the 10° cone, with $Tu = 0.1\%$, varied $Re_T$ and $Ma = 1.05$

Figure 6.17

$C_f$ vs. $Re_x$ for the 10° cone, with $Tu = 0.5\%$, varied $Re_T$ and $Ma = 1.05$

Figure 6.18

55
promising, considering the fact the no information was provided for freestream inlet turbulence quantities. Furthermore, the capability of the model to predict any location of transition is expected to be a significant improvement over traditional two-equation turbulence models, where a fully-turbulent boundary layer is calculated for the entire body. Turbulence quantities were chosen to be $Tu_{inlet} = 0.5\%$ and $Re_{T,inlet} = 100$, which yielded the closest match to experimental data.

Figure 6.19

$C_f$ vs. $Re_x$ for the 10° cone, with $Tu = 0.5\%$, $Re_T = 100$ and varied $Ma$ number
6.3.2 5° Cone Simulations from Fisher and Dougherty

The second cone case corresponds to the work of Fisher and Dougherty [13]. In-flight transition data were recorded for a 5° half-angle sharp-nosed cone. Tests were performed at various unit Reynolds numbers, and Mach numbers ranged from 0.4 to 2. Coefficient of pressure variations were recorded from 0.16% at the lowest Mach number and were taken to decrease linearly to 0.017% near Mach 2. These data were recorded in flight, as opposed to a wind tunnel. The freestream-turbulence levels present during the experiments represent the actual quantities experienced in real flight. Simulations were completed for the \( Re_L = 1.0 \times 10^7 \) recordings. The given pressure fluctuations were converted to freestream turbulence intensity values using Equation (6.1) [50]:

\[
p' = C \rho \left( |U'u'| + |V'v'| + |W'w'| + u'^2 + v'^2 + w'^2 \right)
\]

(6.1)

where \( C = 0.1 \) and the freestream turbulence is taken to be isotropic. The value for turbulent Reynolds number was chosen as \( Re_{T,inlet} = 100 \). The test matrix for the 5° cone of Fisher and Dougherty is outlined in Table 6.2.

Figure 6.20 plots the predicted and experimentally determined transition locations versus Mach number. The predicted values of transition Reynolds numbers are within the range of the experimental data. However, the appropriate trend of the data is not reproduced. It is apparent from the experimental data that transition onset progresses downstream as the Mach number increases. This is the expected behavior, since the values of turbulence intensity are decreasing as Mach number increases. At Mach numbers less than 1.2, the predicted transition locations do not exhibit the same behavior. The cause of this disagreement is investigated in Chapter 7. Note that the model was only
calibrated using experimental data for zero-pressure gradient flat-plate boundary-layer flows. Furthermore, the prediction of transition at any point along the length of the body is a significant improvement over other fully-turbulent two-equation models.

Table 6.2

5° Cone Test Matrix

<table>
<thead>
<tr>
<th>$Ma$</th>
<th>$C_{p_{rms}}$</th>
<th>$Tu$ (%)</th>
<th>$Re_t$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4</td>
<td>0.00160</td>
<td>0.7817</td>
<td>100</td>
</tr>
<tr>
<td>0.5</td>
<td>0.00151</td>
<td>0.7389</td>
<td>100</td>
</tr>
<tr>
<td>0.6</td>
<td>0.00142</td>
<td>0.6981</td>
<td>100</td>
</tr>
<tr>
<td>0.7</td>
<td>0.00133</td>
<td>0.6531</td>
<td>100</td>
</tr>
<tr>
<td>0.8</td>
<td>0.00124</td>
<td>0.6101</td>
<td>100</td>
</tr>
<tr>
<td>0.9</td>
<td>0.00115</td>
<td>0.5669</td>
<td>100</td>
</tr>
<tr>
<td>1.0</td>
<td>0.00106</td>
<td>0.5236</td>
<td>100</td>
</tr>
<tr>
<td>1.1</td>
<td>0.00097</td>
<td>0.4803</td>
<td>100</td>
</tr>
<tr>
<td>1.2</td>
<td>0.00089</td>
<td>0.4368</td>
<td>100</td>
</tr>
<tr>
<td>1.3</td>
<td>0.00080</td>
<td>0.3982</td>
<td>100</td>
</tr>
<tr>
<td>1.4</td>
<td>0.00071</td>
<td>0.3495</td>
<td>100</td>
</tr>
<tr>
<td>1.5</td>
<td>0.00062</td>
<td>0.3050</td>
<td>100</td>
</tr>
<tr>
<td>1.6</td>
<td>0.00053</td>
<td>0.2617</td>
<td>100</td>
</tr>
<tr>
<td>1.7</td>
<td>0.00044</td>
<td>0.2176</td>
<td>100</td>
</tr>
<tr>
<td>1.8</td>
<td>0.00035</td>
<td>0.1735</td>
<td>100</td>
</tr>
<tr>
<td>1.9</td>
<td>0.00026</td>
<td>0.1292</td>
<td>100</td>
</tr>
<tr>
<td>2.0</td>
<td>0.00017</td>
<td>0.0848</td>
<td>100</td>
</tr>
</tbody>
</table>
6.3.3 5° Cone Simulations from Chen et al.

The third and final cone case is based on the experiments of Chen et al. [5]. Transition data were recorded in a quiet wind-tunnel for a 5° half-angle sharp-nosed cone. For zero angle of attack and $Ma = 3.5$, transition was experimentally determined to occur at a location corresponding to $Re_t = 6.97 \times 10^6$. Using Equation (6.1), the freestream turbulence intensity was calculated to be approximately 0.05%. Although measurements are recorded for a Mach number of 3.5, the highest achievable Mach number for the U²NCLE compressible flow solver was 2.2. Figure 6.21 plots the predicted location of transition as Mach number increases from 1.0 to 2.2. The values of freestream turbulence quantities used in the simulations were $Tu = 0.05\%$ and $Re_T = 100$. Since a Mach number
of 3.5 was unattainable, the results are only qualitatively compared to the experimental data, not quantitatively. The transitional Reynolds numbers obtained from the simulations over the Mach number range are relatively close to that determined in the experiments for $Ma = 3.5$. This suggests that transition model is demonstrating the proper behavior, within the limits of the flow solver. Furthermore, the Mach number influence shown in Figure 6.21 is similar to that of Figure 6.20.

![Figure 6.21](image)

$Re_T$ vs. $Ma$ with $Re_T = 100$ for the $5^\circ$ cone of Chen et al. [5]
6.3.4 Angle of Attack

The effect of angle of attack (AoA) on the transition Reynolds number was investigated for the 10° semi-angle cone. According to experimental data [9, 20, 27, 48], the location of transition inception moves upstream on the leeward ray and shifts downstream on the windward ray for angles of attack less than the cone semi-angle. Since no experimental data are available for this particular 10° cone, this is a qualitative study, not quantitative. For the present investigation, the angle of attack was varied from 0° to the cone semi-angle of 10°. The derived values for flow and lift direction due to angle of attack are provided in Appendix B. Similar to the previous simulations for the 10° cone, inlet conditions for the AoA tests were equal to $Tu = 0.5\%$, $Re_T = 100$ and $Ma = 1.05$.

As an example, Figure 6.22 shows the distribution of skin friction on the surface of the cone at an AoA of 1°. As expected, the location of transition onset shifts upstream on the leeward side of the cone and downstream on the windward side. The variation of the transition Reynolds numbers on the leeward and windward rays due to increasing angle of attack is illustrated in Figure 6.23. The model demonstrates the appropriate response to changes in the attack angle. The transition Reynolds numbers for the windward ray become greater than the cone unit Reynolds number ($10^7$) for angles of attack larger than 2°, and transition occurs downstream of the cone. The transition location on the leeward ray begins to move back downstream as the angle of attack approaches the half-angle of the cone. This is because the leeward ray becomes parallel to the flow direction. The transition Reynolds number approaches the flat plate solution.
for similar inlet conditions, indicating that the model is less sensitive to cross-flow velocities and velocity gradients [27]. The results indicate that small changes in angle of attack influence the location of transition on the cone significantly, which is in agreement with observations from experiments [41].

Figure 6.22

$C_f$ contours for the 10° cone at $AoA = 1^\circ$
Figure 6.23

$Re_t$ vs. $AoA$ with $Tu = 0.5\%$ and $Re_T = 100$ for the $10^\circ$ cone
CHAPTER VII
FREESTREAM TURBULENCE DECAY AND LIMITER

7.1 Freestream Turbulence Decay

Although the predicted transition Reynolds numbers from the 5° cone simulations are within the range of range of the experimental data, the predicted values did not reproduce the appropriate behavior. The experimental values of transition location move downstream with increasing Mach number. Since the leading edge values of turbulence intensity were shown to decrease with increasing Mach number in the experiments, this response of the transition location is also expected with the model. However, the simulations did not accurately reproduce this trend for Mach numbers less than one. Several new simulations have been performed to determine the source of this problem. The influence of Mach number was initially investigated. Figure 7.1 and Figure 7.2 present the skin friction distributions for subsonic and supersonic flows, respectively. The inlet values of freestream turbulence quantities are held constant at $Tu = 0.7817\%$ and $Re_T = 100$. As Mach number increases from 0.4 to 1.0, the value of $Re_T$ decreases by approximately 750,000. For increasing Mach numbers above one, $Re_T$ increases by approximately 400,000. The results demonstrate the same trend as the predicted values of $Re_T$ in Figure 6.20, indicating a Mach number influence that is possibly inherent in the U2NCLE compressible flow solver.
Figure 7.1

$C_f$ vs. $Re_x$ for a 5° cone, with $Tu = 0.7817\%$, $Re_T = 100$ and $0.4 \leq Ma \leq 1.0$

Additional simulations were completed to investigate model response to decreasing freestream turbulence intensity. Figure 7.3 provides the distribution of skin friction coefficients for a constant Mach number of $Ma = 1.0$. Freestream-turbulence intensity varies from 0.7817% to 0.6101%, and $Re_T$ is held at a constant value of 100. The change in $Re_T$ over the range of $Tu$ appears to be insignificant. Therefore, the Mach number influence is the largest contributing factor to the discrepancy with the experimental data in the 5° cone simulations. However, a closer look at the results given in Figure 7.3 reveals very interesting behavior.
Figure 7.2

$C_f$ vs. $Re_x$ for a 5° cone, with $Tu = 0.7817\%$, $Re_T = 100$ and $1.0 \leq Ma \leq 1.8$

Figure 7.4 presents the skin friction distributions at the location of transition onset in more detail. As the value of turbulence-intensity decreases, transition inception is moving upstream. Clearly, the model is demonstrating an inappropriate response to freestream turbulence quantities. This behavior is due to the decay of freestream turbulence.

The decay of freestream turbulence was only taken into account for the ERCOFTAC flat plate computations, where data for the rate of decay were provided. Using Euler’s method [4] and the inlet quantities applied in the 5° cone simulations, the actual leading edge conditions for the cone were calculated and found to be approximately equal for all of the inlet turbulence intensities investigated. By holding $Re_T$
Figure 7.3

$C_f$ vs. $Re_x$ for a $5^\circ$ cone, with varied $Tu$, $Re_T = 100$ and $Ma = 1.0$

Figure 7.4

Close-up view of transition locations from Figure 7.3
constant, the highest values of \( Tu \) have a slightly higher rate of decay than the lowest
values and slightly lower leading edge freestream turbulence conditions. This explains
the reverse response of the model to different magnitudes of Tu in Figure 7.4.

For future simulations, Euler’s method can be used to determine the inlet
conditions necessary to obtain the desired leading edge quantities. Instructions are
provided in Appendix C for specifying the desired leading edge conditions and
calculating the “reverse” decay of freestream turbulence across the inlet length of the
computational domain. This methodology was used for the flat plate simulations of
Figure 6.3.

### 7.2 Freestream Turbulence Limiter

The decay of freestream turbulence can also be addressed by using a simple
limiting function. Outside of the boundary layer, the limiter prevents the decay of
freestream turbulence below the desired leading edge quantities. The amount of turbulent
kinetic energy appropriate in the freestream is determined using the following equations:

\[
Tu = \frac{1}{2} \left( \frac{u^2 + v^2 + w^2}{U_{\infty}} \right)
\]  

(7.1)

\[
k_{FS} = \frac{3}{2} \left( \frac{Tu}{100} \right)^2
\]

(7.2)

The ratio between local turbulent kinetic energy and the minimum freestream value is:

\[
k_{ratio} = MIN \left( \frac{k_{FS}}{k}, 1.0 \right)
\]

(7.3)
where $Tu$ is the freestream turbulence intensity desired at the leading edge of the body. Equation (7.3) is the ratio of the desired freestream turbulent kinetic energy to the amount at any particular location with wall distance $d$. Equation (7.4) represents the ratio between the turbulent length scale and the wall-normal distance, $d$:

$$bl_{var} = \frac{\sqrt{k}}{d\omega}$$  \hspace{1cm} (7.4)

This ratio may be interpreted as a means of determining whether or not a particular location lies in the near-wall region ($bl_{var} > 1$) or in the freestream ($bl_{var} < 1$). In the near-wall region, the model should remain unmodified. A damping function used to modify the specific rate of dissipation, $\omega$, is provided in Equation (7.5):

$$f_{bi} = \exp\left[-(bl_{var}^a)\right]$$  \hspace{1cm} (7.5)

Note that $f_{bi} = 0$ in the near-wall region and $f_{bi} = 1$ in the farfield. The overall limiter is imposed on the modeled specific dissipation, $\omega$:

$$\omega_{mod} = \omega \cdot (1.0 - f_{bi}k_{ratio})$$  \hspace{1cm} (7.6)

From Equation (7.6), the unmodified dissipation of turbulent energy is allowed within the turbulent boundary layer. Also, when freestream turbulence is significantly higher than the desired value, $k_{FS}$, turbulence quantities will decay as in the unmodified model. However, the freestream-dissipation rate is reduced as the value of $k$ approaches the desired freestream value and is equal to zero when freestream $k$ equals $k_{FS}$, preventing further decay.

The freestream limiter has been implemented into the new model. Initial validation tests included the ERCOFTAC T3A flat-plate boundary layer and the lowest
Mach number case for the 5° cone. Figure 7.5 demonstrates that the distribution of skin friction is well-predicted for the T3A flat-plate boundary-layer, indicating that implementation of the freestream limiter requires no new calibrations of the model constants. As depicted in Figure 7.6, the new transition Reynolds number predicted for the lowest Mach number case is a significant improvement over the original model prediction. It is expected that the transition location for the remaining Mach numbers will exhibit the same shift toward the experimental data, significantly improving the model results. However, further research is necessary to assess the performance of the model for high Mach number cases.

Figure 7.5

$C_f$ vs. $Re_x$ for the T3A ERCOFTAC test case; Freestream turbulence limiter validation
Figure 7.6

$Re_t$ vs. $Ma$ with $Re_T = 100$ for the 5° cone; Freestream turb. limiter validation
The development and application of a new transition-sensitive k-ω turbulence model has been presented. The model resolves transitional flow effects through the use of damping functions on the eddy viscosity and turbulent kinetic energy production terms. Since the new model is developed entirely within a two-equation, eddy-viscosity framework, it can be easily implemented into any general purpose CFD flow solver with an available two-equation turbulence model. Only the mathematical forms of the present terms in the turbulence model transport equations are modified, maintaining the same level of complexity as commonly used two-equation, fully-turbulent models.

The new model has been implemented into the commercially available flow solver, FLUENT, and a flow solver developed by the MSU SimCenter, U^2NCLE. Results for the initial validation of the model in FLUENT have been previously presented and are reviewed in the present study. The zero-pressure gradient flat plate boundary layer represents the canonical case for validating turbulence model performance. The new model has demonstrated the proper response to increasing freestream turbulence quantities. The model constants associated with natural and bypass transition were then calibrated using flat-plate experimental data. The calibrated constants for each implementation were found to have slightly different values suggesting a dependency of
the model on solver details. However, the calibrations are straightforward and can be completed quite easily. Therefore, the model developed within this study is a valuable tool for production and design level transition-sensitive CFD simulations because: a) The model is robust and has been shown to be reasonably accurate, b) The model can be implemented into any CFD solver that employs a two-equation, eddy-viscosity prescription turbulence model, and c) The model can be calibrated with only a few flat-plate boundary-layer flows.

For model validation in separated flow cases, the FLUENT implementation was used to resolve separation-transition behavior for a cylinder in a cross-flow. The results for the subcritical regime ($Re_D < Re_{D,crit}$) showed separation of the boundary layer and transition to turbulent flow in the shear layer. The model accurately predicted the separation-transition-reattachment-separation behavior that is characteristic of the supercritical regime ($Re_D > Re_{D,crit}$). A comparison of the drag coefficients showed excellent agreement with experimental data. The U²NCLE version of the model was evaluated for a cylinder with a semi-circular leading edge. The flow direction was parallel to the cylinder axis. Results showed that the flow initially separates at the leading edge of the cylinder, transition occurs in the shear layer, and the flow reattaches as a turbulent boundary layer. Although there are no experimental data available for comparison, results using the U²NCLE version illustrate proper model behavior for separated flows.

The U²NCLE version of the model was evaluated for simulations of conical flow geometries, which represent the nose cone of a missile. The model was found to over-
predict the transition location in the 10° cone simulations. However, values for the freestream turbulence quantities are not indicated in the reference and were assumed based upon information given in similar experiments. The initial 10° cone simulations do demonstrate the appropriate trend of the transition location in response to changes in freestream turbulence intensity. In the 5° cone tests of Fisher and Dougherty [13], experimental data were provided for transition Reynolds numbers with increasing Mach number and freestream-turbulence intensity. The values of transition Reynolds numbers predicted by the new model were within the range of the experimental data. However, the model did not duplicate the trend of the experimental data, where the location of transition moves downstream as Mach number increases and values of turbulence intensity decrease. The new model results demonstrated the same type of Mach number influence for the 5° cone tests of Chen et al. [5]. Nevertheless, the results of each cone case offer significant improvement over the traditional fully-turbulent models, where a fully-turbulent boundary layer is assumed from the leading edge of the body. A qualitative investigation of model performance versus angle of attack was completed using the 10° cone. The location of transition onset was predicted to move upstream on the leeward side and downstream on the windward side of the cone. These findings are in agreement with experimental observations.

The decay of freestream turbulence was not considered for any of the cone test cases. Information was not provided in any of the cases for the freestream turbulent length scale, a necessary quantity for calculating the decay rate. However, the results have shown that the decay of freestream turbulence prior to the leading edge of the cone
must be addressed, especially since the distance from the inlet to the cone tip was much larger than the length of the cone. The freestream turbulence conditions at the leading edge of the cones were found to be much less than intended, yielding overpredicted transition Reynolds numbers. Based on desired freestream-turbulence quantities at the leading edge, a technique for determining the required inlet turbulence quantities has been presented. As a more promising alternative, a function for limiting the decay of freestream turbulence has also been provided. Initial testing of this freestream turbulence limiter has shown a significant improvement over the original results. Future work will focus on the validation of the freestream limiting function and its application to each of the simulations presented in this report.
REFERENCES


APPENDIX A

SUMMARY OF MODEL EQUATIONS
\[
\frac{\partial}{\partial x_j} (\rho U_j k) = P_k - \rho \omega k - \epsilon_w + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \tag{A.1}
\]
\[
\frac{\partial}{\partial x_j} (\rho U_j \omega) = P_\omega + R_\omega - E_\omega + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] \tag{A.2}
\]
\[
\rho \bar{u}_i \bar{u}_j - \frac{2}{3} \rho \bar{u}_k \bar{u}_k \delta_{ij} = -2 \mu_T S_{ij} \tag{A.3}
\]
\[
\mu_T = f_\mu C_\mu f_\omega \rho^4 \frac{\bar{k}}{\omega} \tag{A.4}
\]
\[
f_\mu = 1 - \exp \left( \frac{\sqrt{Re_e}}{A_\mu} \right) \tag{A.5}
\]
\[
Re_e = \frac{d_e^3 \frac{1}{2} \frac{1}{3} \frac{1}{2} k}{\nu} \tag{A.6}
\]
\[
C_\mu = \frac{1}{A_0 - A_s f_\omega \frac{\Omega}{\omega}} \tag{A.7}
\]
\[
f_\omega = \frac{\lambda_{eff}}{\lambda_T} \tag{A.8}
\]
\[
\lambda_{eff} = \text{MIN}(C_L d, \lambda_T) \tag{A.9}
\]
\[
\lambda_T = \frac{\sqrt{k}}{\omega} \tag{A.10}
\]
\[
P_k = f_p \mu_T \Omega^2 \tag{A.11}
\]
\[
f_p = f_{p1} + (1 - f_{p1}) f_{TS} \tag{A.12}
\]
\[
f_{p1} = \exp \left[ - \left( \frac{C_{\beta_\nu} \Omega}{f_{Ret} k} \right)^2 \right] \tag{A.13}
\]
\[
f_{Ret} = 1 - \exp \left[ - \left( \frac{k}{C_{Ret} \nu \omega} \right)^2 \right] \tag{A.14}
\]
\[
f_{TS} = 1 - \exp \left[ - \left( \frac{\beta_{TS}}{C_{TS,1}} \right)^2 \right] \tag{A.15}
\]
\[
\beta_{TS} = \text{MAX} \left( \frac{d^2 \Omega}{\nu} - C_{TS,2}, 0 \right) \tag{A.16}
\]
\[ \epsilon_w = \frac{2\mu k}{d^2} \]  \hspace{1cm} (A.17)

\[ R_w = \frac{C_w f_{P1\omega^2(1 - \gamma)}}{f_w} \]  \hspace{1cm} (A.18)

\[ \gamma = MIN \left( \frac{3.33\omega}{\Omega}, 1 \right) \]  \hspace{1cm} (A.19)

\[ P_\omega = C_{\omega1} \omega \frac{P_k}{k} \]  \hspace{1cm} (A.20)

\[ E_\omega = f_w^4 C_{\omega2} \omega^2 \]  \hspace{1cm} (A.21)

\[ S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{1}{3} \frac{\partial U_k}{\partial x_k} \]  \hspace{1cm} (A.22)

\[ \Omega_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right) \]  \hspace{1cm} (A.23)

\[ S = \sqrt{2S_{ij} S_{ij}} \]  \hspace{1cm} (A.24)

\[ \Omega = MIN \left( \sqrt{2\Omega_{ij} \Omega_{ij}}, S \right) \]  \hspace{1cm} (A.25)

\[ T_u = \frac{\sqrt{u^2 + v^2 + z^2}}{U_\infty} \]  \hspace{1cm} (A.26)

\[ k_{FS} = \frac{3}{2} \left( \frac{T_u}{100} \right)^2 \]  \hspace{1cm} (A.27)

\[ k_{ratio} = MIN \left( \frac{k_{FS}}{k}, 1.0 \right) \]  \hspace{1cm} (A.28)

\[ b_{var} = \frac{\sqrt{k}}{d\omega} \]  \hspace{1cm} (A.29)

\[ f_{bl} = exp \left[ - \left( b_{var}^4 \right) \right] \]  \hspace{1cm} (A.30)

\[ \omega_{mod} = \omega \cdot (2.0 - f_{bl} k_{ratio}) \]  \hspace{1cm} (A.31)
Table A.1

**FLUENT® Model Constants**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_\mu$</td>
<td>10</td>
</tr>
<tr>
<td>$A_0$</td>
<td>4.04</td>
</tr>
<tr>
<td>$A_S$</td>
<td>2.12</td>
</tr>
<tr>
<td>$C_L$</td>
<td>2.495</td>
</tr>
<tr>
<td>$C_\beta$</td>
<td>1.4</td>
</tr>
<tr>
<td>$C_{Re_T}$</td>
<td>300</td>
</tr>
<tr>
<td>$C_{TS,1}$</td>
<td>1250</td>
</tr>
<tr>
<td>$C_{TS,2}$</td>
<td>1250</td>
</tr>
<tr>
<td>$C_{\omega,R}$</td>
<td>0.12</td>
</tr>
<tr>
<td>$C_{\omega,1}$</td>
<td>0.44</td>
</tr>
<tr>
<td>$C_{\omega,2}$</td>
<td>0.92</td>
</tr>
<tr>
<td>$\sigma_\omega$</td>
<td>1.17</td>
</tr>
</tbody>
</table>

Table A.2

**U²NCLE Model Constants**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_\mu$</td>
<td>10</td>
</tr>
<tr>
<td>$A_0$</td>
<td>4.04</td>
</tr>
<tr>
<td>$A_S$</td>
<td>2.12</td>
</tr>
<tr>
<td>$C_L$</td>
<td>2.495</td>
</tr>
<tr>
<td>$C_\beta$</td>
<td>2.0</td>
</tr>
<tr>
<td>$C_{Re_T}$</td>
<td>300</td>
</tr>
<tr>
<td>$C_{TS,1}$</td>
<td>1100</td>
</tr>
<tr>
<td>$C_{TS,2}$</td>
<td>1100</td>
</tr>
<tr>
<td>$C_{\omega,R}$</td>
<td>0.12</td>
</tr>
<tr>
<td>$C_{\omega,1}$</td>
<td>0.44</td>
</tr>
<tr>
<td>$C_{\omega,2}$</td>
<td>0.92</td>
</tr>
<tr>
<td>$\sigma_\omega$</td>
<td>1.17</td>
</tr>
</tbody>
</table>
APPENDIX B

AoA ANGLE CALCULATIONS
Table B.1

AoA Angle Calculations

<table>
<thead>
<tr>
<th>Alpha</th>
<th>Radians</th>
<th>Flow Direction</th>
<th>Lift Vector</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>x</td>
<td>y</td>
</tr>
<tr>
<td>0.00</td>
<td>0.00000000</td>
<td>1.00000000</td>
<td>0.00000000</td>
</tr>
<tr>
<td>0.25</td>
<td>0.00436332</td>
<td>0.99999048</td>
<td>0.00436331</td>
</tr>
<tr>
<td>0.50</td>
<td>0.00972665</td>
<td>0.99996192</td>
<td>0.00872654</td>
</tr>
<tr>
<td>0.75</td>
<td>0.01308997</td>
<td>0.99991433</td>
<td>0.01308960</td>
</tr>
<tr>
<td>1.00</td>
<td>0.01745329</td>
<td>0.99984770</td>
<td>0.01745241</td>
</tr>
<tr>
<td>1.25</td>
<td>0.02181662</td>
<td>0.99976203</td>
<td>0.02181489</td>
</tr>
<tr>
<td>1.50</td>
<td>0.02617994</td>
<td>0.99965732</td>
<td>0.02617695</td>
</tr>
<tr>
<td>1.75</td>
<td>0.03054326</td>
<td>0.99953359</td>
<td>0.03053851</td>
</tr>
<tr>
<td>2.00</td>
<td>0.03490659</td>
<td>0.99939083</td>
<td>0.03489950</td>
</tr>
<tr>
<td>3.00</td>
<td>0.05235988</td>
<td>0.99862953</td>
<td>0.05233596</td>
</tr>
<tr>
<td>4.00</td>
<td>0.06981317</td>
<td>0.99756405</td>
<td>0.06975647</td>
</tr>
<tr>
<td>6.00</td>
<td>0.10471976</td>
<td>0.99452190</td>
<td>0.10452846</td>
</tr>
<tr>
<td>8.00</td>
<td>0.13962634</td>
<td>0.99026807</td>
<td>0.13917310</td>
</tr>
<tr>
<td>10.00</td>
<td>0.17453293</td>
<td>0.98480775</td>
<td>0.17364818</td>
</tr>
</tbody>
</table>
APPENDIX C

CALCULATING LEADING EDGE TURBULENCE QUANTITIES
Input: Desired leading edge conditions for freestream turbulence quantities, $Tu$ and $Re_T$.

Output: Inlet conditions that decay to desired leading edge quantities.

Assume only the destruction terms in the equations for $k$ and $\omega$ contribute to the decay of turbulence quantities in the freestream.

**Destruction term for $k$:**

\[
\frac{Dk}{Dt} = -k \omega \\
U_{ref} \frac{dk}{d(-x)} = -k \omega \\
U_{ref} \frac{dk}{d(-x)} = k \omega \\
\frac{dk}{d(-x)} = \frac{k \omega}{U_{ref}}
\]

**Destruction term for $\omega$:**

\[
\frac{D\omega}{Dt} = -C_\omega \omega^2 \\
U_{ref} \frac{d\omega}{dx} = -C_\omega \omega^2 \\
U_{ref} \frac{d\omega}{d(-x)} = C_\omega \omega^2 \\
\frac{d\omega}{d(-x)} = \frac{C_\omega \omega^2}{U_{ref}}
\]

**Define leading edge values of $k$ and $\omega$:**

\[
k_0 = \frac{3}{2} \left( \frac{Tu}{100} \right)^{1.5} \\
\omega_0 = \frac{k_0}{Re_T^{0.5}}
\]

**Employ Eulerian solver to for inverse decay of turbulence quantities:**

<table>
<thead>
<tr>
<th>-$(x)$</th>
<th>$k$</th>
<th>$\omega$</th>
<th>$\frac{dk}{d(-x)}$</th>
<th>$\frac{d\omega}{d(-x)}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>$k_0$</td>
<td>$\omega_0$</td>
<td>$\frac{k_0 - \omega_0}{U_{ref}}$</td>
<td>$\frac{C_\omega \omega_0^2}{U_{ref}}$</td>
</tr>
<tr>
<td>0.01</td>
<td>$k_0 + \frac{dk_0}{d(-x)} (0.01 - 0)$</td>
<td>$\omega_0 + \frac{d\omega_0}{d(-x)} (0.01 - 0)$</td>
<td>$\frac{k_0 + \omega_0 - 0.01}{U_{ref}}$</td>
<td>$\frac{C_\omega \omega_0^2}{U_{ref}}$</td>
</tr>
<tr>
<td>0.02</td>
<td>$k_0 + \frac{dk_0}{d(-x)} (0.02 - 0.01)$</td>
<td>$\omega_0 + \frac{d\omega_0}{d(-x)} (0.02 - 0.01)$</td>
<td>$\frac{k_0 + \omega_0 - 0.02}{U_{ref}}$</td>
<td>$\frac{C_\omega \omega_0^2}{U_{ref}}$</td>
</tr>
</tbody>
</table>

Continue until -$(x)$ = the inlet length of the flow domain.

Figure C.1

Method for calculating leading edge turbulence quantities