Workshop on Computational Turbulence Modeling

Proceedings of a workshop sponsored by the Institute for Computational Mechanics in Propulsion and Center for Modeling of Turbulence and Transition
NASA Lewis Research Center
Cleveland, Ohio
September 15–16, 1993
Workshop on Computational Turbulence Modeling

Proceedings of a workshop sponsored by the Institute for Computational Mechanics in Propulsion and Center for Modeling of Turbulence and Transition
NASA Lewis Research Center
Cleveland, Ohio
September 15–16, 1993
Page intentionally left blank
1. Objectives

The purpose of this meeting was to discuss the current status and future development of turbulence modeling in computational fluid dynamics for aerospace propulsion systems. Various turbulence models have been developed and applied to different turbulent flows over the past several decades and it is becoming more and more urgent to assess their performance in various complex situations. In order to help users in selecting and implementing appropriate models in their engineering calculations, it is important to identify the capabilities as well as the deficiencies of these models. This also benefits turbulence modelers by permitting them to further improve upon the existing models.

This workshop was designed for exchanging ideas and enhancing collaboration between different groups in the Lewis community who are using turbulence models in propulsion related CFD. In this respect this workshop will help the Lewis goal of excelling in propulsion related research.

This meeting had seven sessions for presentations and one panel discussion over a period of 2 days. Each presentation session was assigned to one or two branches (or groups) to present their turbulence related research work. Each group was asked to address at least the following points: current status of turbulence model applications and developments in the research; progress and existing problems; requests about turbulence modeling. The panel discussion session was designed for organizing committee members to answer management and technical questions from the audience and to make concluding remarks.

Organizing Committee

L.A. Povinelli (Chairman) T. Keith R.J. Shaw
J.M. Abbott R. Mankbadi R.J. Simoneau
P.G. Batterton E.J. Mularz R.M. Stubbs
R.E. Gaugler D.R. Reddy A. Shabbir
T.-H. Shih
Page intentionally left blank
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Session I</th>
<th>Turbulence Modeling Activities at CMOTT</th>
</tr>
</thead>
<tbody>
<tr>
<td>CMOTT Research Activities</td>
<td>T.-H. Shih ............................................................... 5</td>
</tr>
<tr>
<td>Research on two equation models</td>
<td>Z. Yang ................................................................. 27</td>
</tr>
<tr>
<td>A Reynolds stress algebraic equation model</td>
<td>J. Zhu ................................................................. 37</td>
</tr>
<tr>
<td>Assessment and development of second order turbulence models</td>
<td>A. Shabbir ............................................................. 55</td>
</tr>
<tr>
<td>A multiple-scale model for compressible turbulent flows</td>
<td>W. Liou and B. Duncan (CFD Branch) ................................. 69</td>
</tr>
<tr>
<td>PDF models</td>
<td>A. Hsu (CFD Branch) and A. Norris ........................................ 79</td>
</tr>
<tr>
<td>Analytical theories of turbulence applied to turbulence modeling</td>
<td>R. Rubenstein ............................................................ 93</td>
</tr>
<tr>
<td>DNS of turbulence, transition and effect of rotation</td>
<td>A. Hsu (CFD Branch) and J. Van der Vegt ................................ 99</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Session II</th>
<th>Heat Transfer and Turbomachinery Flow Physics Branches</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modeling of by-pass transition</td>
<td>F. Simon ............................................................... 107</td>
</tr>
<tr>
<td>Algebraic models for turbine blade heat transfer</td>
<td>R. Boyle ............................................................... 119</td>
</tr>
<tr>
<td>Two-equation models for turbine blade heat transfer</td>
<td>A. Ameri ............................................................... 129</td>
</tr>
<tr>
<td>Thermal turbulence models for turbine blade heat transfer</td>
<td>J. Schwab ............................................................... 141</td>
</tr>
</tbody>
</table>
Session III
Aerothermochemistry and Computational Methods for Space Branches

A coupled implicit solution for turbulent spray combustion in propulsion systems
K.-H. Chen and J.-S. Shuen ........................................................ 147

On turbulent transport of chemical species in compressible reacting flows and Unsteady transitional
flows over forced oscillatory surfaces
S.-W. Kim ....................................................................163

On the accuracy of compact differencing schemes for DNS
S.-T. Yu .....................................................................179

Session IV
Computational Fluid Dynamics Branch and Lewis Research Academy

Turbulent back-facing step flow and the k – ε model: A critical comparison
C. Steffen ....................................................................195

Analysis of supersonic flows using the k – ε models and the RPLUS code
J. Lee .......................................................................203

Validation of a k – ε model in RPLUS2D code for non-reacting/reacting subsonic shear layers
H. Lai .......................................................................215

Computational aeroacoustics as a branch of turbulence research
R. Mankbadi ..................................................................223

Jet noise prediction using k – ε turbulence model
A. Khavaran ..................................................................231

Numerical simulation of supersonic flow using k – ε model
S.H. Shih .....................................................................243

Implementation of a k – ε model in spectral element code
W.M. To .....................................................................255

Session V
Propulsion Systems Division

Subsonic inlet flows with transition
D. Hwang and K. Ahn ...........................................................261

A comparative study of turbulence models in predicting hypersonic inlet flows
K. Kapoor .....................................................................269

Turbomachinery flows
R. Chima .....................................................................279
Session VI
Propulsion Systems Division

Low emission combustors
J. Deur ................................................................. 293

Development of a reliable algebraic turbulence model giving engineering accuracy at reasonable cost
B.P. Leonard and J.E. Drummond (University of Akron) ......................................................... 305

Application of algebraic and two-equation turbulence models to HSR nozzle flow calculations
J. DeBonis and N. Georgiadis ................................................................. 321

Aircraft icing
M. Potapczuk ................................................................. 343

Applied RNG turbulence model for 3-D turbomachinery flows
K. Kirtley ................................................................. 355

Applied $k - \varepsilon$ and Baldwin-Lomax turbulence models for S-ducts
G. Harloff ................................................................. 367

Session VII
Inlet, Duct, and Nozzle Flow Physics Branch

PROTEUS experience with the modified MML turbulence model
J. Conley ................................................................. 377

Turbulence model experiences for a round-to-rectangular transition duct
C. Towne ................................................................. 389

PROTEUS experience with three different turbulence models
T. Bui ................................................................. 403

Several examples where turbulence models fail in inlet flow field analysis
B.H. Anderson ................................................................. 413
Main Issues Raised in the Workshop

(summarized by workshop committee)

A total of thirty five papers were presented at the workshop covering a wide spectrum of work related to propulsion systems as is evident from the contents of this proceedings. Several problems and issues related to the turbulence and transition modeling were raised in these presentations as well as in the panel discussion. These issues are briefly summarized below.

1. Transition

Flow over a turbine blade very often undergoes a transition from laminar to turbulent and accurate modeling of this phenomenon is very important. The transition in propulsion systems (named as bypass transition) has its own features and is caused by elevated free stream turbulence, surface roughness, etc. Current work at Lewis shows that turbulence models are not being modified for their use in transitional flows. Current work at Lewis using turbulence models to capture the transition have only limited success. Bypass transition models developed at CMOTT, which take into account the effect of intermittency, account for the effect of free-stream turbulence. The development of transition models which account for both the free stream turbulence and surface roughness effects is a pacing item for transitional flow calculations for propulsion systems. Much work still needs to be done in this area.

2. Heat transfer

Currently, most turbulent heat transfer calculations at Lewis use Reynolds analogy (i.e. constant turbulent Prandtl number) coupled with the zero-equation or two-equation models for the velocity field. The turbulent Prandtl number, in general, is not constant for complex flows. Therefore, improvement in the heat transfer calculations requires a more accurate approach for calculating the turbulent thermal diffusivity. One such approach is to use the transport equations for the temperature fluctuations and its dissipation rate. Some work in this direction is in progress at CMOTT.

Since, as noted above, transition occurs on turbine blades quite often, part of the reason for the poor predictions of heat transfer is the inability of current models to capture this transition. There is a need to improve the modeling of this type of transitional flow.

3. Turbulent chemical reaction interactions

In this workshop, two approaches were presented for the calculation of turbulent flows with finite rate chemical reactions. One is the pdf method for species in which the chemical production rate of species can be exactly treated without modeling.
The other is the conventional moment closure scheme in which the averaged chemical production rate of species is estimated by the mean temperature of the flow field. Further studies are needed for clarifying and understanding the effect of chemical reaction on turbulence and vice versa. DNS of simple turbulent reacting flows could be very helpful to sort out this issue.

Studies on realistic chemical reaction models are also needed for simulating the interactions between turbulence and chemical reaction.

4. Wall bounded flows

Some presentations at this workshop showed that for high turbulent Reynolds number flows, the use of wall function as a boundary condition often produces reasonable results for even complex flows in spite of its theoretical in-validity. In addition, this approach can significantly reduce the number of numerical grid points which, sometimes, is critical for complex engineering calculations. It was also pointed out that sometimes a key ingredient of the flow, such as a separation bubble, is under y\textsuperscript{+} = 40. Such flows, therefore, cannot be well simulated with the wall function approach and in such cases turbulence model equations must be integrated to the wall. (This type of turbulence modeling is usually called low Reynolds number turbulence modeling.)

Most low Reynolds number turbulence models presented in this workshop are not tensorially invariant, i.e. the model equations contain the detailed coordinate parameters which require *ad hoc* treatments in applications to complex flow geometries. In addition, existing k-\(\varepsilon\) models do not respond to the effect of pressure gradient very well. To address these problems, two efforts have been made at CMOTT. For the wall function approach, a modified new wall function is proposed to better account for the effect of pressure gradient. For low Reynolds number k-\(\varepsilon\) models, a Galilean and tensorially invariant realizable k-\(\varepsilon\) model is developed which can also respond to the pressure gradient very well. Further testing of this model in complex flows is needed to find its potential as well as its deficiencies.

5. Inflow condition

Some calculations showed that results were very much dependent on the inflow conditions chosen for turbulence quantities (say, \(k\) and \(\varepsilon\)). To obtain a good comparison with experiments, the turbulence inflow conditions had to be adjusted. This raises the uncertainty about correctness of calculations as well as the performance of turbulence models. To resolve this problem, the determination of correct inflow conditions by either experiments or other methods is important. In situations lacking experimental inflow turbulence quantities, methods to establish reasonable inflow conditions are critical and must be further studied. Turbulence model testing should not be based on the flows with unknown inflow conditions except for the situations in which the flows are self preserving.

6. Numerical method

It was noticed that different turbulence models were built in different codes. It is difficult to compare the performance of different models based on the results
produced by different codes. It is agreed that the validation and comparison of different turbulence models must be based on the grid independent converged solutions produced by one code with the same/similar initial and boundary conditions.

Turbulence equations are much more stiff and complicated than the Navier-Stokes equations. The study of numerical methods to effectively obtain the accurate solution of turbulence equations is also very critical for turbulent CFD calculations. The collaboration between CFD and turbulence model developers will help to resolve this problem.

7. Hierarchy of turbulence models and standards

Several different types of turbulence models have been developed over the past several decades. Each of them was proposed for certain type of flows. In CFD calculations, all models are one-point closure schemes which include zero-equation models (or mixing length models), one- and two-equation models, algebraic Reynolds stress models and second order closure models. At each level of these turbulence models, there are several versions (as many as 10 or more) of same type of models. Each later version was developed to improve the performance over its parent-model for the flows of interest.

Engineers were often, and are still, confused by so many levels of turbulence models with so many versions at each level. Do we need all levels of turbulence models? Is there a best model at each level? The possible answer to above questions is yes.

From zero-equation to second order closure models, each level of turbulence models contains a certain degree of turbulence physics which gives each level of models a different degree of potential as well as limitation. For example, the mixing length models have the potential for modeling of attached boundary layer flows. While the good prediction of pressure distribution is often obtained, the predictions of skin friction and heat transfer are often poor, especially for separated flows. This is not surprising because the mixing length models were developed for that kind of flows and contain little turbulence physics. In this workshop, many presentations showed that two-equation isotropic eddy viscosity models perform much better than mixing length models for variety of flows. This is also not surprise because there are two turbulent transport equations involved in this type of models. However, for flows in which the anisotropy of turbulence plays main role, we must use anisotropic turbulence models. In addition, if the evolution of Reynolds stresses, streamline curvature, rotation, etc. are important, the second order closure schemes will be more appropriate.

In principle, the higher level of models (if they were well developed), give better performance in a wider range of turbulent flows and, of course, at the expense of higher computing cost and numerical difficulty. Further studies on the potential as well as deficiencies of each level of models in turbulent flow calculations are needed. Identifying the best model version at each level seems an urgent task for engineering CFD applications. To achieve this goal, systematic model testing for a variety of bench mark flows as well as real flows in propulsion systems is needed.
Page intentionally left blank
Research Activities at the Center for Modeling of Turbulence and Transition

Tsan-Hsing Shih

1. Introduction
2. General developments
   2.1 Turbulent constitutive relations
   2.2 Mechanical and scalar dissipation equations
   2.3 Eddy viscosity transport equation
3. One-point closure schemes
   3.1 One-equation eddy viscosity transport equation model
   3.2 Galilean and tensorial invariant realizable $k$-$\epsilon$ model
   3.3 Reynolds stress algebraic equation model
   3.4 Scalar flux algebraic equation model
   3.5 Reynolds stress transport equation model
   3.6 Non-equilibrium multiple-scale model
   3.7 Bypass transition model
   3.8 Joint scalar PDF model
4. RNG and DIA
5. Numerical simulation

Abstract

The main research activities at the Center for Modeling of Turbulence and Transition (CMOTT) are described. The research objective of CMOTT is to improve and/or develop turbulence and transition models for propulsion systems. The flows of interest in propulsion systems can be both compressible and incompressible, three dimensional, bounded by complex wall geometries, chemically reacting, and involve “bypass” transition. The most relevant turbulence and transition models for the above flows are one- and two-equation eddy viscosity models, Reynolds stress algebraic- and transport-equation models, pdf models, and multiple-scale models. All these models are classified as one-point closure schemes since only one-point (in time and space) turbulent correlations, such as second moments (Reynolds stresses and turbulent heat fluxes) and third moments ($\overline{u_i u_j u_k}$, $\overline{u_i \theta^2}$), are involved. In computational fluid dynamics, all turbulent quantities are one-point correlations. Therefore, the study of one-point turbulent closure schemes is the focus of our turbulence research. However, other research, such as the renormalization group theory, the direct interaction approximation method and numerical simulations are also pursued to support the development of turbulence modeling.
1. Introduction

The center for modeling of turbulence and transition was established as a special focus group within the Institute for Computational Mechanics in Propulsion at NASA Lewis Research Center in 1990. Its objective is to improve and/or develop turbulence and transition models for computational fluid dynamics (CFD) applied in propulsion systems. With the advance of computer technology and algorithms, accurate turbulence and transition modeling becomes the pacing item for improving flow calculations used in propulsion system design in all its key elements. The flows of interest in propulsion systems are, in general, very complex since there are wall-bounded three-dimensional complex geometries, chemical reactions, compressibility and transition, etc. In order to accurately predict these flows one must correctly model the turbulent stresses and scalar fluxes which are one-point (in time and space) turbulent correlations. For flows with finite rate chemical reactions, accurate modeling of the production rate of species is crucial for turbulent flow calculations. Based on the above considerations, turbulence modeling activities at CMOTT are focused on one-point closure schemes, that is, using the moment closure schemes for the turbulent velocity field and the joint scalar pdf method for the reacting scalar field.

There are various moment closure schemes which have been developed for various engineering applications. However, in practice, one often finds that the existing models need to be improved and/or re-developed in order to reasonably simulate complex flow structures appearing in propulsion systems. For this purpose, CMOTT devotes itself to improving and/or re-developing these moment closure schemes which include eddy viscosity (one- and two-equation) models, second moment algebraic- and transport-equation models, non-equilibrium multiple-scale models, and bypass transition models. In addition, other studies supporting the development of one-point closure schemes have been also carried out (for example, studies on renormalization group theory (RNG), direct interaction approximation (DIA), direct numerical simulation (DNS) and large eddy simulation (LES)).

In this report, we first describe the general development of turbulent constitutive relations, turbulent mechanical and thermal dissipation and a new eddy viscosity equation. Second, we describe the detailed developments on each moment closure scheme and the pdf method. Then the RNG and DIA methods and finally, the numerical simulation of particular turbulence phenomena, such as rotation and bypass transition, etc., are considered.

Each research subject is the joint project of several CMOTT researchers and visitors. In describing research activities, the names of involved researchers will be mentioned for reference.
2. General Developments

2.1 Turbulent Constitutive Relations

Reynolds stress

Using the invariant theory in continuum mechanics and Generalized Cayley-Hamilton formulas for tensor products, a turbulent constitutive relation (or a general turbulence model) for any turbulent correlations can be obtained, in principle. Therefore, this theory provides an avenue to develop better turbulence models than those existing. For example, a commonly used constitutive relation for Reynolds stresses \( u_i u_j \) (in terms of the mean deformation rate tensor \( U_{i,j} \) and the turbulent velocity and length scales characterized by the turbulent kinetic energy \( k \) and its dissipation rate \( \varepsilon \)) is

\[
-\overline{u_i u_j} = C_\mu \frac{k^2}{\varepsilon} (U_{i,j} + U_{j,i}) - \frac{2}{3} k \delta_{ij} \tag{2.1.1}
\]

The effective eddy viscosity \( \nu_T \) defined as

\[
\nu_T = \frac{-\overline{u_i u_j}}{U_{i,j} + U_{j,i}} = C_\mu \frac{k^2}{\varepsilon} \text{ for } i \neq j \tag{2.1.2}
\]

is isotropic since \( \nu_T \) is a scalar quantity. However, the invariant theory enables us to formulate the following general model (Shih and Lumley\(^1\), Johansson\(^2\)):

\[
\overline{u_i u_j} = \frac{2}{3} k \delta_{ij} + 2a_2 \frac{K^2}{\varepsilon} (U_{i,j} + U_{j,i} - \frac{2}{3} U_{i,i} \delta_{ij}) \\
+ 2a_4 \frac{K^3}{\varepsilon^2} (U_{i,j}^2 + U_{j,i}^2 - \frac{2}{3} \Pi_1 \delta_{ij}) \\
+ 2a_6 \frac{K^3}{\varepsilon^2} (U_{i,k} U_{j,k} - \frac{1}{3} \Pi_2 \delta_{ij}) \\
+ 2a_7 \frac{K^3}{\varepsilon^2} (U_{k,i} U_{k,j} - \frac{1}{3} \Pi_2 \delta_{ij}) \\
+ 2a_8 \frac{K^4}{\varepsilon^3} (U_{i,k} U_{j,k}^2 + U_{i,k}^2 U_{j,k} - \frac{2}{3} \Pi_3 \delta_{ij}) \\
+ 2a_9 \frac{K^4}{\varepsilon^3} (U_{k,i} U_{k,j}^2 + U_{k,j} U_{k,i}^2 - \frac{2}{3} \Pi_3 \delta_{ij}) \\
+ 2a_{10} \frac{K^5}{\varepsilon^4} (U_{i,k}^2 U_{j,k} - \frac{1}{3} \Pi_4 \delta_{ij}) \\
+ 2a_{11} \frac{K^5}{\varepsilon^4} (U_{i,k} U_{k,j}^2 - \frac{1}{3} \Pi_4 \delta_{ij}) \\
+ 2a_{12} \frac{K^5}{\varepsilon^4} (U_{k,i}^2 U_{k,j} - \frac{1}{3} \Pi_4 \delta_{ij}) \\
+ 2a_{13} \frac{K^5}{\varepsilon^4} (U_{k,i} U_{k,j}^2 - \frac{1}{3} \Pi_4 \delta_{ij}) \\
+ 2a_{14} \frac{K^6}{\varepsilon^5} (U_{i,k} U_{l,k} U_{l,j} + U_{j,k} U_{i,k} U_{l,i} - \frac{2}{3} \Pi_5 \delta_{ij}) \\
+ 2a_{15} \frac{K^6}{\varepsilon^5} (U_{i,k} U_{l,k}^2 U_{l,j} + U_{j,k} U_{l,k} U_{l,i}^2 - \frac{2}{3} \Pi_5 \delta_{ij}) \\
+ 2a_{16} \frac{K^6}{\varepsilon^5} (U_{i,k} U_{l,k} U_{l,m} U_{i,m} + U_{j,k} U_{l,k} U_{l,m} U_{i,m}^2 - \frac{2}{3} \Pi_7 \delta_{ij})
\]
where

\[
\begin{align*}
\Pi_1 &= U_{i,k} U_{k,i}, & \Pi_2 &= U_{i,k} U_{i,k}, & \Pi_3 &= U_{i,k} U_{i,k}^2, & \Pi_4 &= U_{i,k} U_{i,k}^2, \\
\Pi_5 &= U_{i,k} U_{i,k} U_{i,i}^2, & \Pi_6 &= U_{i,k} U_{i,k}^2 U_{i,i}, & \Pi_7 &= U_{i,k} U_{i,k} U_{i,m} U_{i,m}^2
\end{align*}
\]  

(2.1.4)

From Eq.(2.1.3), the effective eddy viscosity

\[
(\nu_T)_{ij} = \frac{-\bar{u}_i \bar{u}_j}{U_{i,j} + U_{j,i}}
\]  

(2.1.5)

is no longer a scalar and, hence, is an anisotropic eddy viscosity. It is noticed that the first two terms on the right hand side of Eq.(2.1.3) represent the standard \( k-\varepsilon \) eddy viscosity model (2.1.1) and that the first five terms of Eq.(2.1.3) are of the same form as the models derived from both the two-scale DIA approach (Yoshizawa\(^3\)) and the RNG method (Rubinstein and Barton\(^4\)).

Eq.(2.1.3) is a general model for \( \bar{u}_i \bar{u}_j \). It contains 11 undetermined coefficients which are, in general, scalar functions of various invariants of the tensors in question, such as \( S_{ij} S_{ij} \) (strain rate) and \( \Omega_{ij} \Omega_{ij} \) (rotation rate) which are \((\Pi_2 + \Pi_1)/2\) and \((\Pi_2 - \Pi_1)/2\) respectively. The detailed forms of these scalar functions must be determined by other model constraints, for example, realizability, and by experimental data. Eq.(2.1.3) contains 12 terms; however, its quadratic tensorial form may be sufficient for practical applications. We will see later in section 3.3 that the constitutive relation (2.1.3) has a significant impact on the development of Reynolds stress algebraic equation models.

**Turbulent scalar flux \( \bar{\theta} u_i \)**

We assume the following functional form:

\[
\bar{\theta} u_i = F_i(U_{i,j}, T_{i,i}, k, \varepsilon, \bar{\theta}^2, \varepsilon_\theta)
\]  

(2.1.6)

where \( \bar{\theta}^2 \) is the variance of a fluctuating scalar and \( \varepsilon_\theta \) is its dissipation rate. Eq.(2.1.6) indicates that the scalar flux depends on not only the mean scalar gradient \( T_{i,i} \) but also the mean velocity gradient \( U_{i,j} \) and the scales of both velocity and scalar fluctuations characterized by \( k, \varepsilon, \bar{\theta}^2, \varepsilon_\theta \).

Applying the invariant theory, we may obtain the following general constitutive
The coefficients $a_1 - a_{18}$ are, in general, functions of the time scale ratio $\frac{\epsilon}{\epsilon_\theta}$ and the other invariants formed by the tensors in question, for example, $T_{k,k}$, $T_{i,j}$, etc. Again, Eq.(2.1.7) implies that the effective eddy diffusivity

$$(\gamma_T)_i = \frac{-\overline{\partial u_i}}{T_{i,j}}$$

is not isotropic. It is noticed that the first term on the right hand side of Eq.(2.1.7) is the standard eddy diffusion model, and the models derived from the two-scale DIA (Yoshizawa$^5$) and the RNG method (Rubinstein and Barton$^6$) are similar to the first two terms of Eq.(2.1.7). In practice, a form containing the first two terms on the right hand side of Eq.(2.1.7) may suffice. Further development of this model for turbulent heat transfer is described in Section 3.4.

The Researchers involved with the subject in this section are T.-H. Shih, J. Zhu, A. Shabbir, J.L. Lumley$^\dagger$ and A. Johansson.$^\ddagger$

### 2.2 Mechanical and Scalar Dissipation Equation

#### Mechanical dissipation $\varepsilon$

In turbulence modeling, we often need turbulent characteristic velocity and length scales. While the turbulent kinetic energy $k$ is used to characterize the velocity scale, the mechanical dissipation rate $\varepsilon$ and the scalar dissipation rate $\varepsilon_\theta$ are used to characterize the length scales for mechanical and scalar fields, respectively. Comparing with the turbulent kinetic energy equation, the exact dissipation rate equation is
very complicated. In this equation, all the terms which represent important turbulence physics (for example, turbulent diffusion, generation and destruction) are unknown and are of complex forms that are all related to small scales of turbulence. Therefore, in the literature, the exact dissipation equation is not considered as a useful equation to work with. Instead, one creates a model equation by assuming an analogy to the turbulent kinetic energy equation, i.e., one assumes that the model dissipation rate equation also has generation and destruction terms which are assumed to be proportional respectively to the production and dissipation terms in the turbulent kinetic energy equation over the period of large eddy turn-over time characterized by $k/\varepsilon$. The resulting model dissipation rate equation is written as

$$
\varepsilon, t + U_i \varepsilon, i = \nu \varepsilon, ii - (\varepsilon u_i, i)
- C_{\epsilon 1} \frac{\varepsilon}{k} u_i u_j U_{i, j} - C_{\epsilon 2} \frac{\varepsilon^2}{k}
$$

(2.2.1)

Recently, Lumley\(^7\) proposed a dissipation rate equation based on the concept of spectral energy transfer caused by interactions between eddies of different sizes. This model equation mimics the physics of statistical energy transfer from large eddies to small eddies and is of a different form than equation (2.2.1).

In this study, we explore another rational way to obtain the model dissipation rate equation which contains certain important physics and hope it will work better than the existing one. The idea is that first, there is a relationship between the dissipation rate $\varepsilon$ and the mean-square vorticity fluctuation $\bar{\omega}_i \bar{\omega}_i$ at high Reynolds numbers or in homogeneous turbulence:

$$
\varepsilon = \nu \bar{\omega}_i \bar{\omega}_i
$$

and second, all the terms appearing in the $\bar{\omega}_i \bar{\omega}_i$ equation have more clear physical meanings than that in the $\varepsilon$ equation so that the $\bar{\omega}_i \bar{\omega}_i$ equation is easier to model. Once the $\bar{\omega}_i \bar{\omega}_i$ equation is modeled, a model dissipation rate equation will be readily obtained.

The exact equation for $\bar{\omega}_i \bar{\omega}_i$ is

$$
\left(\frac{\bar{\omega}_i \bar{\omega}_i}{2}\right)_t + U_j \left(\frac{\bar{\omega}_i \bar{\omega}_i}{2}\right)_{, j} = \nu \left(\frac{\bar{\omega}_i \bar{\omega}_i}{2}\right)_{, jj} - \frac{1}{2} \left(u_j \omega_i \omega_i\right)_{, j} + \omega_i u_{i, j} \Omega_j
- \bar{u}_j \bar{\omega}_i \Omega_{i, j} + \bar{\omega}_i \omega_j U_{i, j} + \bar{\omega}_i \bar{\omega}_j \bar{u}_{i, j} - \nu \bar{\omega}_i, j \bar{\omega}_i, j
$$

(2.2.2)

where $u_i$ and $U_i$ are the fluctuating and mean velocities, and $\omega_i$ and $\Omega_i$ are the fluctuating and mean vorticity which are defined by

$$
\omega_i = \epsilon_{ijk} u_{k, j} \quad \Omega_i = \epsilon_{ijk} U_{k, j}
$$

(2.2.3)

Tennekes and Lumley\(^8\) clearly described the physical meaning of each term in equation (2.2.2). Order of magnitude analysis shows that the first, third, fourth and fifth terms on the right hand side of Eq.(2.2.2) become small compared with all other
terms in the equation as the turbulent Reynolds number increases. The sixth and seventh terms are the production due to fluctuating vortex stretching and the dissipation due to the viscosity of the fluid. As the turbulent Reynolds number increases these last two terms become dominant and the balance between them determines the evolution of vorticity fluctuations. Neglecting terms \( \omega_i \omega_j \Omega_{ij}, -\overline{\omega_i} \overline{\omega_i} \Omega_{ij}, \overline{\omega_i} \overline{\omega_j} U_{ij}, \) and \( \nu(\overline{\omega_i \omega_j})_{ij} \), the evolution of \( \overline{\omega_i \omega_i} \) at large Reynolds number will be described by the following equation,

\[
\frac{(\overline{\omega_i \omega_i})}{2}, \dot{t} + U_j \left( \frac{(\overline{\omega_i \omega_i})}{2} \right)_j = -\frac{1}{2} (u_j \omega_i \omega_i)_j + \omega_i \omega_j u_{ij} - \nu \omega_i \omega_j \omega_{ij} \tag{2.2.4}
\]

To model \( \overline{\omega_i \omega_j} u_{ij} - \nu \overline{\omega_i \omega_j} \omega_{ij} \), let us first estimate \( \overline{\omega_i \omega_j} u_{ij} \). We define an anisotropic tensor \( b^\omega_{ij} \):

\[
b^\omega_{ij} = \frac{\omega_i \omega_j}{\overline{\omega_k}} - \frac{1}{3} \delta_{ij} \tag{2.2.5}
\]

then \( \overline{\omega_i \omega_j} u_{ij} \) can be written as

\[
\overline{\omega_i \omega_j} u_{ij} = b^\omega_{ij} \overline{\omega_k} u_{ij} \tag{2.2.6}
\]

We expect that the vortex stretching tends to align vortex lines with the strain rate so that the anisotropy \( b^\omega_{ij} \) would be proportional to the strain rate \( s_{ij} \), i.e.,

\[
b^\omega_{ij} \propto \frac{s_{ij}}{s}, \quad \text{where} \quad s = (2s_{ij} s_{ij})^{1/2}, \quad s_{ij} = (u_{ij} + u_{ji})/2 \tag{2.2.7}
\]

This leads to the following model:

\[
\overline{\omega_i \omega_j} u_{ij} \propto \omega_k^2 (2s_{ij} s_{ij})^{1/2} \propto \omega_k^2 \sqrt{2s_{ij} s_{ij}} \tag{2.2.8}
\]

where we have assumed that \( \omega_k^2 \) and \( (2s_{ij} s_{ij})^{1/2} \) are well correlated.

Using the relation, \( \omega_i = \varepsilon_{ijk} u_{kj} \), it is not difficult to show that at large turbulent Reynolds number,

\[
\overline{\omega_i \omega_i} \approx 2s_{ij} s_{ij} \tag{2.2.9}
\]

and Eq.(2.2.8) can be also written as

\[
\overline{\omega_i \omega_j} u_{ij} \propto \omega_k^2 \sqrt{\omega_i^2} \tag{2.2.10}
\]

Equation (2.2.10) indicates that this term is of the order \( (u^3/l^3) R_t^{3/2} \) as it should be. On the other hand, from eq.(2.2.4) the term \( \overline{\omega_i \omega_j} u_{ij} - \nu \overline{\omega_i \omega_j} \omega_{ij} \) must be of the order \( (u^3/l^3) R_t \) which is the order of magnitude of all the other terms in Eq.(2.2.4), therefore the term \( -\nu \overline{\omega_i \omega_j} \omega_{ij} \) must cancel the term (2.2.10) or (2.2.8) such that the difference of these two terms is smaller than the term (2.2.10) or (2.2.8) by an order
of $R_t^{1/2}$. This suggests that the combination $\overline{\omega_i \omega_j u_{i,j}} - \nu \overline{\omega_i \omega_i} \overline{\omega_i \omega_j}$ can be modeled by the following two terms:

$$\frac{\overline{\omega_k^2 \omega_i^2}}{\frac{k}{\nu} + \sqrt{\frac{\omega_i^2}{2}}} \overline{\omega_k^2 S}$$

(2.2.11)

because the ratio of $k/\nu$ to $\sqrt{\omega_i^2}$ and the ratio of $s$ to $S$ are of order $R_t^{1/2}$, where $k \approx u^2$ is the turbulent kinetic energy and $S$ is the mean strain rate $(2S_{ij}S_{ij})^{1/2}$. Equation (2.2.11) does give the right order of magnitude for $\overline{\omega_i \omega_j u_{i,j}} - \nu \overline{\omega_i \omega_i} \overline{\omega_i \omega_j}$. Therefore, the dynamical equation for fluctuating vorticity (2.2.4) at large Reynolds number can be modeled as

$$\left(\omega_j \omega_i \frac{\omega_i}{2}\right)_t + U_j \left(\frac{\omega_j \omega_i}{2}\right)_i = -\frac{1}{2} \left(\overline{u_j \omega_i \omega_i}\right)_j + C_{\omega 1} \frac{\overline{\omega_k^2 S}}{\frac{k}{\nu} + \sqrt{\omega_i^2}} - C_{\omega 2} \frac{\overline{\omega_i^2 \omega_i^2}}{\frac{k}{\nu} + \sqrt{\omega_i^2}}$$

(2.2.12)

Using $\varepsilon = \nu \overline{\omega_i \omega_i}$, we readily obtain the following model dissipation rate equation,

$$\varepsilon_t + U_j \varepsilon_j = -\left(\overline{u_j \varepsilon}\right)_j + C_{\omega 1} S \varepsilon - C_{\omega 2} \frac{\varepsilon^2}{\frac{k}{\nu} + \sqrt{\varepsilon \omega_i^2}}$$

(2.2.13)

where $C_{\omega 1}$ and $C_{\omega 2}$ are the model coefficients which are expected to be constant at large Reynolds number.

It should be noticed that Eq.(2.2.13) is different from the standard $\varepsilon$ equation (2.2.1) by both the generation and destruction terms. First, the Reynolds stresses do not appear in the generation term and the new form of the generation term is similar to that proposed by Lumley? which is based on the concept of spectral energy transfer. Second, the destruction term is well behaved so that equation (2.2.13) will not have a singularity anywhere in the flow field. We expect that equation (2.2.13) will be numerically much more robust than equation (2.2.1).

Equation (2.2.13) can be applied to any level of turbulence modeling including second order closure models; however the turbulent transport term $(\overline{e u_i})_i$ needs to be modeled differently at different levels of turbulence modeling. In an eddy viscosity model, the term $(\overline{e u_i})_i$ will be modeled as

$$(\overline{e u_i})_i = -\left(\frac{\nu T}{\sigma_\varepsilon} \varepsilon_i\right)_i$$

(2.2.14)

The coefficients $C_{\omega 1}$, $C_{\omega 2}$, $\sigma_\varepsilon$ and the eddy viscosity $\nu T$ must be calibrated using experimental data (Shih et al.).

**Scalar dissipation $\varepsilon_\theta$**

A similar analysis leads to the following model scalar dissipation rate equation:

$$\varepsilon_{\theta_t} + U_j \varepsilon_{\theta,j} = -\left(\overline{u_j \varepsilon_\theta}\right)_j + C_{\theta 1} S \varepsilon_\theta + C_{\theta 2} P \nu^{-1/2} \Phi \sqrt{\varepsilon \varepsilon_{\theta}} - C_{\theta 3} \frac{\varepsilon_\theta}{\frac{k}{\nu} + \sqrt{\varepsilon \omega_i^2}}$$

(2.2.15)

where $\Phi = \sqrt{T_i^2 T_i^2}$ and $T$ is the mean scalar quantity, such as, the mean temperature. Further development of heat transfer model is described in Section 3.4.

The Researchers involved with the subject in this section are T.-H. Shih, W. Liou, A. Shabbir and Z. Yang.
2.3 Eddy Viscosity Transport Equation

In eddy viscosity models, one accepts the following simple constitutive relation

\[ \bar{u}_i \bar{u}_j = -2\nu_T S_{ij} + \frac{2}{3} k \delta_{ij} \]  
\[(2.3.1)\]

and assumes that the eddy viscosity is characterized by some kind of velocity and length scales \( u' \) and \( \ell \):

\[ \nu_T \propto u' \ell \]  
\[(2.3.2)\]

In two-equation \( k-\varepsilon \) eddy viscosity models, for example, one specifies that

\[ u' \propto k^{\frac{3}{2}}, \quad \ell \propto \frac{k^{\frac{3}{2}}}{\varepsilon} \]  
\[(2.3.3)\]

and, hence, the eddy viscosity is assumed as

\[ \nu_T = C_\mu \frac{k^2}{\varepsilon} \]  
\[(2.3.4)\]

The eddy viscosity assumption (2.3.4) is commonly adopted in two-equation models. Eqs.(2.3.1) and (2.3.4) together with appropriate \( k \) and \( \varepsilon \) equations have been widely used in engineering calculations. However, for cases where the mean flow changes quickly or has a strong mean stream-line curvature or rotation, etc., this kind of model does not work very well, since the assumption (2.3.4) is too simple to account for the effect of the above mean flow structure on eddy viscosity.

The main purpose of this study is to drop the assumption (2.3.4) and to derive an exact equation for \( \nu_T \) based on Eq.(2.3.1) and other exact turbulence equations (i.e. first principles). In this way, we hope that some important turbulent physics can be brought into the eddy viscosity and that a physically sound turbulence eddy viscosity can be calculated.

Using Eq.(2.3.1), we may write for incompressible flows

\[ \bar{u}_i \bar{u}_j \bar{u}_i \bar{u}_j = 2\nu_T^2 S^2 + \frac{4}{3} k^2, \quad \text{where} \quad S^2 = 2S_{ij}S_{ij} \]  
\[(2.3.5)\]

Differentiating both sides, we obtain

\[ \frac{D}{Dt} \nu_T = -\frac{S_{ij}}{S^2} \frac{D}{Dt} \bar{u}_i \bar{u}_j - \frac{\nu_T}{2S^2} \frac{D}{Dt} S^2 \]  
\[(2.3.6)\]

The equation for \( \bar{u}_i \bar{u}_j \) can be written as

\[ \frac{D}{Dt} \bar{u}_i \bar{u}_j = D_{ij} + P_{ij} + \Pi_{ij} - \varepsilon_{ij} + C_{0ij} \]  
\[(2.3.7)\]
where

\[ D_{ij} = [\nu u_i u_j - \bar{u}_i u_j u_k],_k \]
\[ P_{ij} = -\bar{u}_i \bar{u}_k U_{j,k} - \bar{u}_j \bar{u}_k U_{i,k} \]
\[ \Pi_{ij} = \frac{1}{\rho} p_i u_j + p_j u_i \]
\[ \varepsilon_{ij} = 2\nu \bar{u}_i \bar{u}_j u_k,k \]
\[ C_{oij} = -2\varepsilon_{imk} \Omega_m \bar{u}_i \bar{u}_j - 2\varepsilon_{jmk} \Omega_m \bar{u}_i \bar{u}_j \]

Inserting Eq.(2.3.7) into Eq.(2.3.6), we obtain an exact transport equation for eddy viscosity

\[ \frac{D}{Dt} \nu_T = -\frac{S_{ij}}{S^2} (D_{ij} + P_{ij} + \Pi_{ij} - \varepsilon_{ij} + C_{oij}) - \frac{\nu_T}{2S^2} \frac{D}{Dt} S^2 \quad (2.3.8) \]

In this equation, all the important turbulence physics in the Reynolds stress equation, such as Reynolds stress diffusion term \( D_{ij} \), production term \( P_{ij} \), pressure-velocity gradient correlation term \( \Pi_{ij} \) and dissipation tensor \( \varepsilon_{ij} \), are involved. Comparing with the standard eddy viscosity assumption (2.3.4), this exact eddy viscosity equation (2.3.8) contains very rich turbulence physics. This equation also implies that a second order closure model will naturally lead to a corresponding eddy viscosity model.

Now, as an example, we use Launder Reece and Rodi's\(^1\) model and a gradient transport model for the triple velocity correlation \( -\bar{u}_i \bar{u}_j \bar{u}_k = \frac{\nu_T}{\sigma} \bar{u}_i \bar{u}_j,_{k} \) to derive a model equation for \( \nu_T \). The resulting equation is

\[ \frac{D}{Dt} \nu_T = [(\nu + \frac{\nu_T}{\sigma}) \nu_{T,k}],_k + (\nu + \frac{3 \nu_T}{2} \frac{\nu_{T,k}}{S^2}) \frac{\nu_{T,k}}{S^2} + (\nu + \frac{\nu_T}{\sigma}) \frac{2\nu_T S_{ij} S_{ij,kl}}{S^2} \]
\[ + \frac{4}{15} k - C_1 \frac{\varepsilon}{S^2} + 2(C_2 - 2) \nu_T \frac{S_{ik} S_{kj} S_{ji}}{S^2} - \frac{\nu_T}{2S^2} \frac{D}{Dt} S^2 \quad (2.3.9) \]

Note that the Coriolis terms do not explicitly appear in this equation; however the rotation effect on \( \nu_T \) could be carried over through the mean flow field. In addition, we also note that there are no extra model coefficients introduced in Eq.(2.3.9). All model coefficients (\( \sigma, C_1 \) and \( C_2 \)) are brought in from the second order closure model. The values of these model coefficients may need adjustment in model applications. Note that Eq.(2.3.9) is not a self-consistent equation since the turbulent kinetic energy \( k \) and its dissipation rate \( \varepsilon \) are also involved. Eq.(2.3.9) together with \( k-\varepsilon \) transport equations will provide a new three-equation model which may better represent the effect of mean flow structure as well as mean flow history on the eddy viscosity.

The Researchers involved with the subject in this section are T.-H. Shih, Z Yang, and W. Liou.

3. One-Point Closure Schemes

In this section, we describe the developments on each of the moment closure scheme and the pdf method which are of concern at CMOTT. The first two sections
3.1 and 3.2 describe the one- and two-equation isotropic eddy viscosity models. Sections 3.3 and 3.4 describe the new developments on Reynolds stress and scalar flux algebraic equation models. Section 3.5 assesses Reynolds stress transport equation models. Section 3.6 describes a multiple-scale model for non-equilibrium turbulence. Section 3.7 is about transition models. Finally, in Section 3.8 the pdf method for turbulent chemical reaction is described.

3.1 One-equation eddy viscosity model

Recently developed one-equation eddy viscosity models are either based on the assumption (Baldwin and Barth):

$$\nu_T = C_\mu \frac{k^2}{\varepsilon}$$

(3.1.1)

or created according to computational experience (Spalart and Allmaras). Both of them are successful in some flow calculations. This scheme is quite attractive in CFD because one only needs to solve one scalar \(\nu_T\) equation without bothering about other turbulence quantities. However, comparing with \(k-\varepsilon\) two equation models, the above mentioned one-equation \(\nu_T\) models do not contain any more turbulent physics. In fact, Baldwin and Barth's model is, basically, a change of dependent variable based on Eq.(3.1.1) plus some extra approximations. Therefore, in principle, we should not expect any superior performance over two-equation models. However, if we do not use the assumption (3.1.1), there is the possibility to improve and extend the capability of one-equation eddy viscosity models.

The objective of this study at CMOTT is to derive a physically sound eddy viscosity equation which contains rich turbulent physics and accounts for various effects from mean flow structures.

Note that in Section 2.3 we have already derived an exact equation for the eddy viscosity (2.3.8) and also a model equation (2.3.9) which is based on the Reynolds stress transport equation model of Launder, Reece and Rodi (LRR). All turbulent physics contained in the Reynolds stress equation can be brought into the eddy viscosity equation. Therefore, in principle, the transport equation (2.3.9) should be better than existing one-equation models based on Eq.(3.1.1). However, Eq.(2.3.9) is not self-consistent because \(k\) and \(\varepsilon\) are also involved. To make Eq.(2.3.9) self-consistent, we must model \(k\) and \(k/\varepsilon\) in terms of \(\nu_T\) and \(S\). In most shear flows, the energy-containing eddy turn-over time \(k/\varepsilon\) is of the same order as the mean flow time scale \(S^{-1}\), so that \(\varepsilon/k \propto S\) is a reasonable model. In addition, a crude dimensional analysis gives \(k \propto \nu_T S\) and this is, of course, reasonable only for shear flows. After the above considerations, the resulting self-consistent one-equation model is:

$$\frac{D}{Dt} \nu_T = (\nu + \frac{\nu_T}{\sigma})(\nu_T)_{,k} - \frac{C_{\nu_0}}{\sigma}(\nu_T)_{,k}(\nu_T)_{,k} + C_{\nu_1} S \nu_T$$

$$+ 2(C_{\nu_2} - 2)\nu_T \frac{S_{ik}S_{kj}S_{ji}}{S^2} - \frac{\nu_T}{2S^2} \frac{D}{Dt} S^2$$

(3.1.2)
where the diffusion terms from the Reynolds stress equation (2.3.7) have been manipulated and approximated. Eq.(3.1.2) clearly exhibits the various effects of the mean flow on the eddy viscosity.

The model coefficients $C_{\nu 1}$, $C_{\nu 2}$ and $\sigma$ can be determined by using the experimental data of homogeneous shear flows, free shear flows and boundary layer flows as well as the relations in the inertial sublayer. Extensive tests of this model in various flows are carrying out at the CMOTT.

The Researchers involved with the subject in this section are T.-H. Shih, W. Liou, Z. Yang and J. Zhu.

3.2 Galilean and tensorial invariant realizable $k$-$\varepsilon$ model

The two-equation $k$-$\varepsilon$ eddy viscosity model is one of the most widely used turbulence models in engineering calculations. The $k$-$\varepsilon$ model has versions for high Reynolds numbers and for low Reynolds numbers. For wall bounded turbulent flows, the high Reynolds number $k$-$\varepsilon$ model (for example, Launder and Spalding\(^{13}\)) must be applied together with a wall function as its boundary condition, while the low Reynolds number $k$-$\varepsilon$ model (for example, Jones and Launder\(^{14}\)) can be integrated to the wall. The high Reynolds number $k$-$\varepsilon$ model of Launder and Spalding is considered as a standard $k$-$\varepsilon$ model. We notice that even though the model dissipation rate equation is created by assuming an analogy with the turbulent kinetic energy, there was not much modification until Lumley\(^{7}\) and Shih et al.\(^{9}\) For near wall turbulence, in addition to Jones and Launder’s model, there are many other versions of low Reynolds number $k$-$\varepsilon$ models (such as Chien\(^{15}\), Shih and Lumley\(^{16}\), Yang and Shih\(^{17}\)) which have made better performance over Jones and Launder’s model.

There are, probably, four or five issues worth mentioning about existing low Reynolds number $k$-$\varepsilon$ models: the model constants are not consistent with those in the high Reynolds number $k$-$\varepsilon$ model; the wall correction terms and damping functions are related to the wall distance so that models are not tensorial invariant; a nonrealistic dissipation rate near the wall is introduced; they are not always realizable since normal stress could become negative; and finally, they do not work very well for boundary layer flows with various pressure gradients.

The objective of this study at CMOTT is to overcome the above mentioned problems. First, we propose a vorticity dynamics based dissipation rate equation as a part of high Reynolds number $k$-$\varepsilon$ base model.\(^{9}\) Second, based on the invariant theory, inhomogeneous terms for the dissipation rate equation are proposed which enable the model to better respond to the change of pressure gradients (Yang and Shih\(^{18}\)). Third, the wall distance parameter is removed from the damping function so that the model is tensorially invariant (Yang and Shih\(^{19}\)). The model constants are consistent with those in the high Reynolds number $k$-$\varepsilon$ model. Finally, the non-negativity of normal Reynolds stresses, the realizability condition, is imposed.

The Researchers involved with the subject in this section are Z. Yang, T.-H. Shih and C. Steffen.
3.3 Reynolds stress algebraic equation model

All eddy viscosity models including one- and two-equation models are isotropic. For the flows where anisotropy is important, for example, the secondary flows driven by turbulent normal stresses in a square duct or curved duct, eddy viscosity models do not produce correct flow structures. To overcome this intrinsic deficiency of isotropic eddy viscosity models, one proposes a Reynolds stress algebraic equation model which will provide an effective anisotropic eddy viscosity. The first such a model was proposed by Rodi²⁰ and it achieved some success in the prediction of anisotropic related flow structure. However, Rodi's formulation is a set of algebraic non-linear system equations for Reynolds stresses and it often creates numerical difficulty in obtaining a converged solution. Recently, Taulbee²¹ obtained an explicit algebraic expression for the Reynolds stress using Pope's²² tensor expansion formulation and solved this numerical difficulty. However, in general, Rodi's formulation assumes that the ratio \( \frac{\overline{u_i u_j}}{k} \) is constant and, of course, this is not really true for most turbulent flows of interest. Therefore, sometimes, this Reynolds stress algebraic equation model produces even worse results than the isotropic eddy viscosity models for cases where eddy viscosity models are appropriate.

Alternative ways for obtaining effective anisotropic eddy viscosity models have been tried by a few researchers, for example, the DIA method by Yoshizawa³, the RNG method by Rubinstein and Barton⁴ and invariant theory by Shih and Lumley.¹. It is interesting to point out that the RNG and DIA methods result in the same formulation and that this formulation is the first five terms of a general constitutive relation Eq.(2.1.3) except that the model coefficients are different.

One of our goals at CMOTT is to search for an effective anisotropic eddy viscosity model for complex turbulent flows where the nonequilibrium of turbulence is not very severe so that the constitutive relation (2.1.3) is more or less valid. We have explored the potential capability of Eq.(2.1.3) and found that a truncation of Eq.(2.1.3) up to the quadratic terms of the mean velocity gradients is sufficient for various flows of interest. The model coefficients are determined such that realizability for the normal stresses is ensured. The detailed analysis is described by Shih et al.²³.

The quadratic version of Eq.(2.1.3) together with the standard \( k-\varepsilon \) transport equations, successfully predicts many complex flows as well as simple flows which include backward-facing step flows; confined coflowing jets; confined swirling coaxial jets; flows in 180° curved duct; flows in a diffuser and a nozzle; boundary layer flows with pressure gradient and turbulent free shear flows. See references²³–²⁵ for detailed results.

The Researchers involved with the subject in this section are J. Zhu and T.-H. Shih.

3.4 Scalar flux algebraic equation model

In parallel with Reynolds stress algebraic equation model, we have also tried to develop an effective anisotropic scalar eddy diffusivity model for scalar (heat) fluxes based on the new constitutive relation (2.1.7) and the new thermal dissipation rate
equation (2.2.16). We have determined that it seems sufficient to truncate Eq.(2.1.7)
up to linear terms of the mean velocity gradient, i.e.,

$$\bar{\theta}u_i = a_1 k \left( \frac{k}{\varepsilon \theta} \right)^{1/2} T_{i} + \frac{k^2}{\varepsilon} \left( \frac{k}{\varepsilon \theta} \right)^{1/2} (a_2 U_{i,j} + a_3 U_{j,i}) T_{j}$$

(3.4.1)

This equation indicates that the heat flux and the mean temperature gradient are
not necessarily in alignment due to the distortion of the flow field. This means that
the effective scalar eddy diffusivity is anisotropic.

Eq.(3.4.1) together with the $\bar{\theta}^2$ and $\varepsilon_{\theta}$ equations will be a closed set of model
equations for turbulent heat fluxes. The model coefficients are calibrated from
homogeneous flows. Detailed analysis and a few model tests are described in this
research briefs by A. Shabbir.

The Researchers involved with the subject in this section are A. Shabbir and
T.-H. Shih.

3.5 Reynolds stress transport equation model

The Reynolds stress transport equation model is considered as a next generation
of advanced turbulence modeling for engineering applications. In principle, the
second moment equations describe various effects of the mean flow and external
agencies on the evolution of turbulence, hence, are the most attractive way (also
the simplest correct way) to study turbulent flows.

Various closure models for second moment equations have been developed. The
success of these closures are marginal and vary with each flow. To identify the
sources of their deficiencies, one often uses simple flows where the specific model
term in the second moment equations can be isolated, hence, the corresponding
model can be checked against experimental data or direct numerical simulation
(DNS). For example, using pre-distorted anisotropic homogeneous relaxation flows,
we may check the return-to-isotropy models with experimental data or DNS. However,
for other flows, several model terms, such as, triple velocity correlations, rapid
and slow pressure-strain correlations, etc., simultaneously exist and can not be iso-
lated in the experiments. In these cases (for example, in a homogeneous shear flow
or a channel flow) only DNS can provide all the information for simultaneously
checking various models.

We have examined various existing closure models using experimental data as
well as DNS data (Shih et al.\textsuperscript{26} and Shih and Lumley\textsuperscript{27}). Conclusive statements are
difficult to draw at this time. However, the following remarks can be made about
various closures for the second moment equations, i.e., the triple velocity correlation
$T_{ijk}$, the rapid and slow pressure related correlations $\Pi_{ij}^{R}$, $\Pi_{ij}^{S}$, and the dissipation
rate tensor $\varepsilon_{ij}$:

a) $T_{ijk}$. All the existing models, such as Daly and Harlow\textsuperscript{28}, LRR\textsuperscript{10}, Lumley\textsuperscript{20},
etc., are not very satisfactory for highly inhomogeneous flows, such as flow near the
wall. However, for flows where the inhomogeneity is not very high, the above closure
models become close to each other and also closer to the DNS data. In addition,
the triple velocity correlations in these situations are usually small comparing with
other terms in the equation, so that modeling of this term is not as critical as other terms for the results of turbulent flow calculations, except for the flow near the wall.
b) $\Pi_{i j}^p$. It is very clear from all the available DNS data that nonlinear models, such as, Shih and Lumley\textsuperscript{30} are much better than linear models, such as SSG\textsuperscript{31}. It seems also that the following constitutive relation

$$\Pi_{i j}^p = F(\bar{u}_i \bar{u}_j, U_{i,j})$$

is quite appropriate, i.e., its dependence on turbulent Reynolds number and other parameters is quite weak and can be neglected. However, one deficiency of this form observed by Reynolds\textsuperscript{32} is that it can not take the rotation effect into account.
c) $\Pi_{i j}^l$. This term is usually modeled together with the dissipation tensor $\varepsilon_{i j}$ and the combination of the two is called the return-to-isotropy term. All existing models are unsatisfactory at the present time. They are far from "universal", i.e., their performance varies from flow to flow. It is noticed that some strange behavior of return-to-isotropy (for example, for some pre-distorted flow relaxation, turbulence evolves toward anisotropy before it returns to isotropy) occurs and cannot be possibly modeled with the following constitutive relation:

$$\Pi_{i j}^l = F(\bar{u}_i \bar{u}_j, k, \varepsilon)$$

In addition, the behavior of return-to-isotropy was found to depend not only on the Reynolds stresses at the present time but also on their history according to DNS data (Lee\textsuperscript{33}). It may be also necessary to include triple velocity correlations into the above constitutive relations from the definition of $\Pi_{i j}^l$. The term $\Pi_{i j}^l$ seems highly dependent on the turbulent Reynolds number and slowly approaches to its asymptote as Reynolds number goes to infinity, so that, in general, one should not exclude its dependence on turbulent Reynolds number even for moderate Reynolds numbers. In addition, $\Pi_{i j}^l$ is also noticeably affected by the mean strain rate according to the DNS data\textsuperscript{34}, so that, in general, the mean strain rate should be also considered in the constitutive relation. In short, much more research is needed for developing a better model of $\Pi_{i j}^l$.

The Researchers involved with the subject in this section are T.-H. Shih and A. Shabbir.

3.6 Non-equilibrium multiple-scale model

To consider the effect of the nonequilibrium of energy spectrum on turbulent quantities, such as $k$, $\varepsilon$ and $\bar{u}_i \bar{u}_j$, etc., Hanjelic et al.\textsuperscript{35} are the first to propose a partition in the turbulent energy spectrum. Because of the nonequilibrium, the rate that energy enters the low wave number region, $\varepsilon_p$, does not equal to the energy transfer rate from low wave numbers to high wave numbers, $\varepsilon_t$. Therefore it is reasonable to describe the evolution of the energy contained in low wave number region, $k_p$, and high wave number region $k_t$, separately. As a result, the time scale or the length scale defined by different energy transfer rates will be different and this multiple-scale concept reflects the nonequilibrium effect of turbulence.
We think that this concept would be more appropriate for compressible flows because the compressibility often creates nonequilibrium interactions between large and small eddies. We first modify Hanjelic et al.'s model, test it in various free shear flows and boundary layer flows and then extend it to compressible flows by consideration of the effects of compressibility on the equations for $k_p$ and $\epsilon_p$. The proposed model is tested in both compressible free shear flows and boundary layer flows. For detailed analysis and flow calculations see the report by Duncan et al.\textsuperscript{36} and Liou and Shih\textsuperscript{37}.

The Researchers involved with the subject in this section are W. Liou, T.-H. Shih and B. Duncan.

3.7 Bypass transition model

The onset of turbulence transition in the propulsion system is often highly influenced by the free stream turbulence. This transition process does not go through the linear instability but is mainly controlled by nonlinear processes. Therefore, it is sometimes called "bypass" transition. Because of this highly nonlinear process of transition, turbulence models may be used to predict it. In fact, many two-equation models, for example, $k$-$\epsilon$ eddy viscosity models of Launder and Sharma\textsuperscript{38}, Chien\textsuperscript{15}, etc., do mimic bypass transition on a flat plate when the free stream has a certain amount of turbulent intensity. However, to obtain an accurate prediction of bypass transition, the study of the bypass transition process and physics is needed. The conventional turbulence models must be modified to take into account the intermittent phenomena of transitional flows.

We have proposed transition models based on a two-equation turbulence model using an intermittency factor to modify either the eddy viscosity or modeled $k$-$\epsilon$ equations. Successful results for a flat plate boundary layer under various free-stream turbulence intensities are obtained. For details see the report by Yang and Shih\textsuperscript{39}.

The Researchers involved with the subject in this section are Z. Yang and T.-H. Shih.

3.8 Joint scalar PDF model

One of the critical problems in turbulent combustion is how to treat the interaction between the chemical reaction on the turbulence. The estimation of the production rate of compositions based on the mean flow temperature would be in a very large error for flows with finite rate chemical reactions. The reason is that the production rate of compositions depends not only on the mean values of temperature $T$ and compositions $C_i$, but also very much depends on the detailed fluctuations of temperature $\theta$ and compositions $c_i$. The moment closure scheme of modeling the production rate of compositions in terms of the mean flow temperature, the mean compositions and various correlations consisting of the fluctuating temperature and composition, such as $\theta^2$, $\theta c_i$, $\bar{c}_i\bar{c}_j$, ..., has not been successful. However, the PDF method allows us to treat chemical reaction exactly without modeling (Pope\textsuperscript{40}). Therefore, for the study of turbulent combustion problems, we use the joint scalar PDF transport equation for the scalar field and the moment closure schemes for
the velocity field and develop a hybrid solver consisting of a Monte Carlo scheme and a conventional CFD method. For detailed description of this procedure and its applications see Hsu \textsuperscript{41} and Hsu \textit{et al.}\textsuperscript{42}

The Researchers involved with the subject in this section are A. Hsu, A.T. Norris and J.Y. Chen\textsuperscript{†}

4. RNG and DIA

In developing one point turbulence models, conventional modeling methods can be supplemented by “non-conventional” methods such as renormalization group theory (RNG) and the direct interaction approximation (DIA). These are two point theories formulated in wavevector or fourier space; one point models are derived by integration over wavevectors. This approach provides theoretical support for conventionally derived models and sometimes suggests theoretically derived forms for the empirical elements, whether constants or functions, which appear in these models.

We have applied RNG methods to both the eddy viscosity and Reynolds stress transport equation models. In addition to the $k - \varepsilon$ model proposed by Yakhot and Orszag\textsuperscript{43}, it is possible to obtain constitutive relations for Reynolds stress and heat fluxes (Rubinstein and Barton\textsuperscript{4,6}) which are special cases of the general results Eqs.(2.1.3) and (2.1.7). By applying the perturbation theories of Yakhot and Orszag\textsuperscript{43} to the relevant correlations, expansions in powers of the mean velocity gradient are obtained for the stresses and heat fluxes; quadratic truncation of the series leads to a stress model Eq.(2.1.3) with constant $a_4, a_6, a_7$ and a heat flux model Eq.(2.1.7) with constant $a_2, a_3$ in which the constants are in good agreement with empirically selected values. The forms derived are also consistent with the DIA analysis of Yoshizawa\textsuperscript{3,5}.

The RNG method also provides a formulation for closing the Reynolds stress transport equation (Rubinstein and Barton\textsuperscript{44}). Perturbative evaluation of the correlations $\Pi_{yy}^{II}$ and $\Pi_{y}^{II}$ leads to series expansions in powers of the mean velocity gradient. These series can be consolidated, or “resumed” using the known perturbation series for the Reynolds stresses by methods analogous to Pade approximation. Systematic lowest order summation leads to a Reynolds stress transport equation with a form identical to the LRR model equation and with constants in reasonable agreement with empirically chosen values. Higher order resummation leading to nonlinear models of the type described in Sec. 3.5 remains an open possibility. The possibility of such resummation in the context of DIA has been discussed by Yoshizawa\textsuperscript{45,46}.

Recent work has focussed on nonequilibrium time dependent relations between the Reynolds stress and the mean flow derived from a simplification of the DIA theory of shear turbulence. In this theory, shear turbulence is modeled by a non-Markovian eddy damping acting against the mean shear. The RNG and DIA Reynolds stress transport models and the LRR model all assume Markovian damping; as in the

\textsuperscript{†} Professor, University of California, Berkeley, CA
molecular theory of transport coefficients, Markovian damping describes long time behavior and is incorrect at short times. The most important consequence of non-Markovian damping is a strong suppression of eddy damping at short times. This leads to closer agreement between the present theory and rapid distortion theory at short time. This is important in modeling oscillating shear flows: recent work of Mankbadi\textsuperscript{47} shows that RDT based models best predict such flows. In transient homogeneous shear flow at high strain rates, the LRR model predicts rapid onset of eddy damping leading to excessive growth of turbulence kinetic energy at short times. The suppression of eddy damping at short times in the present model should lead to improved predictions for this flow as well.

Another consequence of this theory is a stress model Eq.(2.1.3) in which the coefficients $a_2, \ldots$ are functions of the mean strain rate. This theory can be described as RDT with a modified total strain determined by the response function of the DIA theory of isotropic turbulence. The introduction of a phenomenological modified total strain has often been advocated in the RDT literature to improve the agreement between RDT and shear flow data; here the modified total strain is deduced as a consequence of the theory. In the special but important case of simple shear flow in which $\partial U_i / \partial x_j = S_{ij} \delta_{j2}$, the result can be formulated in terms of Eq.(2.1.3) in which, for example, $a_2 = a_2(Sk/\varepsilon)$ and the function $a_3$ is found exactly from RDT. There are analogous results for the coefficients $a_4, a_6, a_7$; in simple shear flow, the remaining terms in Eq.(2.1.3) identically vanish. Extension of this theory to other mean shear tensors depends on the tabulation of the corresponding RDT solution.

The researchers involved with the subject in this section are R. Rubinstein and A. Yoshizawa.\textsuperscript{†}

5. Numerical Simulation

To obtain a better understanding of the effect of compressibility and rotation on turbulence, numerical simulations of compressible homogeneous shear flows and rotational flows are carried out. The effects of compressibility and rotation on the energy spectrum and energy cascade between turbulent eddies has been analyzed (Hsu and Shih\textsuperscript{48}). These simulations support the idea of the multiple-scale model for nonequilibrium compressible turbulent flows (W. Liou and Shih\textsuperscript{37}).

Another numerical simulation is the transition subjected to the free stream large disturbances. The objective of this simulation is to obtain some insight into the transition physics and to provide data base for bypass transition modeling. Based on the assumption that the transition process is mainly controlled by large scale motions, we use a high accuracy finite difference Navier-Stokes solver with course grids to simulate the large scale motions of transition. A preliminary calculation of bypass transition was carried out. Various statistics of the calculated flow field are under examination.

The Researchers involved with the subject in this section are A. Hsu, C. Liou\textsuperscript{‡}, Z. Yang, A. Shabbir, T.-H. Shih.

\textsuperscript{†} Professor, Tokyo University, Japan

\textsuperscript{‡} Professor, University of Colorado, Denver, CO
References


22 S.B. Pope, “A more general effective-viscosity hypothesis,” J. Fluid Mech., 72, 331 (1975)


Motivations and Objectives:

- $k - \epsilon$ model is the most widely used turbulence model in engineering calculations.

- However, the following deficiencies need to be fixed:
  - Currently, most $k - \epsilon$ models for near wall turbulence contain geometry parameters.
  - The form of the $\epsilon$ equation lacks a solid theoretical foundation.
  - The $k - \epsilon$ model performs rather inadequately for flows with adverse pressure gradient.
  - The capability of $k - \epsilon$ models in predicting bypass transition due to the freestream turbulence needs improving.
Modeling of Near Wall Turbulence

- Near wall $k - \epsilon$ model = Standard $k - \epsilon$ model + near wall effect.

- The near wall effect:
  - The time scale approaches the Kolmogorov time scale near the wall.
  - The damping function is parametrized by a new parameter which is independent of the coordinate system.

- The resulting model is Galilean and tensorial invariant.

- The resulting model is robust numerically.

Turbulent Channel Flow at $Re_\tau = 395$
ZPG Turbulent Boundary Layer at $Re_\theta = 1410$

FPG Boundary Layer

APG Boundary Layer

29
On the Wall Functions

- The advantages of the wall function approach:
  - Reduce the number of grid points by half, at least.
  - Reduce the numerical stiffness of the dissipation rate equation, by less grid stretching.

- The limit of wall function approach: the flow is assumed to be attached to the wall.

- Existing wall functions are based on the flat plate BL at zero pressure gradient. It is inadequate when the pressure gradient is not zero.

- A new set of wall functions are obtained:
  - They are based on the asymptotic behavior of the governing equations in the log layer.
  - They contain the effect of the pressure gradient.
A Vorticity Dynamics Based Model for the Dissipation Rate Equation

- The dynamic equation for the fluctuating vorticity is analyzed. (The terms in the fluctuating vorticity equation have clearer physical meanings than terms in the dissipation rate equation.)
- For large Reynolds numbers, $\epsilon = \nu \bar{\omega} \bar{\omega}$. The resulting model equation has a better foundation than the standard $\epsilon$ equation.
- The resulting model equation always gives a positive production in dissipation rate.
  - The model calculation is expected to be more robust for complex flow calculations.

Flow Inhomogeneity and the Dissipation Rate Equation

- The exact dissipation rate equation contains source terms due to the flow inhomogeneity.
- However, the existing $\epsilon$ equations are homogeneous. (The source terms are the same for both the homogeneous and inhomogeneous flows.)
- A new model equation for $\epsilon$ is proposed, which accounts for the inhomogeneity effect:
  - Flow inhomogeneity is represented by $\nabla S$ and $\nabla k$.
  - Invariant theory is used to derive such a model equation.
- The resulting model equation accurately account for the effect of the pressure gradient.
Rotating homogeneous shear flows

\( \Omega / S = 0.0 \)  \( \Omega / S = 0.5 \)  \( \Omega / S = -0.5 \)

Mixing Layer  Plane Jet  Round Jet
Modeling of Bypass Transition

- Low Reynolds number $k - \epsilon$ models could mimic transition.
  - The predictions are not very good.
  - Among these models, the Launder-Sharma model gives the best prediction.

- New model for bypass transition is proposed by introducing the effect of the intermittency.

- The transition model recovers to turbulence model at the end of the transition zone.

- Calculations of the benchmark flows show that the present model gives better predictions compared with the $k - \epsilon$ models without the intermittency effect.
Summaries:

The capabilities of $k - \epsilon$ model are enhanced in the following areas:

- A Galilean and tensorial invariant $k - \epsilon$ model for near wall turbulence.

- A new set of wall functions for attached flows.

- A new model equation for the dissipation rate:
  - It has a better theoretical basis.
  - It contains the contribution of flow inhomogeneity.
  - It captures the effect of the pressure gradient accurately.

- A better model for bypass transition due to freestream turbulence.
A REYNOLDS STRESS
ALGEBRAIC EQUATION MODEL

Presented by

J. Zhu

Motivation

- Two-equation turbulence models (K-ε, K-ω)
  - Simple, robust and
    computationally inexpensive
  - Isotropic
  - Not realizable. \( \bar{u}_i \bar{u}_i \) may become negative

- Reynolds stress transport equation models
  - Anisotropic
  - Capable of simulating \( \bar{u}_i \bar{u}_j \)-transport
  - Difficult to model higher-order correlations
  - Complex and computationally expensive
Objective

Develop a Reynolds stress algebraic equation model which combines the respective advantages of two-equation models and Reynolds stress transport equation models:
• Simple, robust and efficient
• Anisotropic
• Realizable

Derivation of Constitutive Relation

• First step: assume some relationship between quantities of interest $\bar{u}_i \bar{u}_j = F_{ij}(U_{i,j}, K, \epsilon)$

• Second step: use invariant theory to determine the function $F_{ij}$
  1. An invariant can only be a function of other invariants
  2. If the LHS of an equation is bilinear in arbitrary vectors $A_i$ and $B_j$, the RHS must also be bilinear in $A_i$ and $B_j$
A general constitutive model of $u_i u_j$

$$u_i u_j = \frac{2}{3} K \delta_{ij} + 2 a_2 \frac{K^2}{\varepsilon} (U_{i,j} + U_{j,i} - \frac{2}{3} U_{i,i} \delta_{ij}) + 2 a_4 \frac{K^3}{\varepsilon^2} (U_{i,j}^2 + U_{j,i}^2 - \frac{2}{3} \Pi_1 \delta_{ij})$$

$$+ 2 a_6 \frac{K^3}{\varepsilon^2} (U_{i,k} U_{j,k} - \frac{1}{3} \Pi_2 \delta_{ij}) + 2 a_7 \frac{K^3}{\varepsilon^2} (U_{k,i} U_{k,j} - \frac{1}{3} \Pi_2 \delta_{ij})$$

$$+ 2 a_8 \frac{K^4}{\varepsilon^3} (U_{i,k} U_{j,k}^2 + U_{j,k} U_{i,k}^2 - \frac{2}{3} \Pi_3 \delta_{ij}) + 2 a_{10} \frac{K^4}{\varepsilon^3} (U_{k,i} U_{k,j}^2 + U_{k,j} U_{k,i}^2 - \frac{2}{3} \Pi_3 \delta_{ij})$$

$$+ 2 a_{12} \frac{K^5}{\varepsilon^4} (U_{i,k} U_{j,k}^2 - \frac{1}{3} \Pi_4 \delta_{ij}) + 2 a_{13} \frac{K^5}{\varepsilon^4} (U_{k,i} U_{k,j}^2 - \frac{1}{3} \Pi_4 \delta_{ij})$$

$$+ 2 a_{14} \frac{K^5}{\varepsilon^4} (U_{i,k} U_{i,k} U_{j,i}^2 + U_{j,k} U_{i,k} U_{i,k}^2 - \frac{2}{3} \Pi_5 \delta_{ij})$$

$$+ 2 a_{16} \frac{K^6}{\varepsilon^5} (U_{i,k} U_{i,k} U_{j,k}^2 + U_{j,k} U_{i,k} U_{j,k}^2 - \frac{2}{3} \Pi_6 \delta_{ij})$$

$$+ 2 a_{18} \frac{K^7}{\varepsilon^6} (U_{i,k} U_{i,k} U_{i,m} U_{j,m} + U_{j,k} U_{i,k} U_{i,m} U_{i,m}^2 - \frac{2}{3} \Pi_7 \delta_{ij})$$

$$\Pi_1 = U_{i,k} U_{i,k} , \quad \Pi_2 = U_{i,k} U_{i,k} , \quad \Pi_3 = U_{i,k} U_{i,k} , \quad \Pi_4 = U_{i,k} U_{i,k} ,$$

$$\Pi_5 = U_{i,k} U_{i,k} U_{i,i} , \quad \Pi_6 = U_{i,k} U_{i,k} U_{i,i} , \quad \Pi_7 = U_{i,k} U_{i,k} U_{i,m} U_{i,m}$$

The first five terms are of the same form as that of Yoshizawa (1984) and Rubinstein and Barton (1990).

Realizable Algebraic Equation Model

$$\ddot{u}_i \ddot{u}_j = \frac{2}{3} K \delta_{ij} - \nu_t (U_{i,j} + U_{j,i})$$

$$+ \frac{C_{\tau 1}}{A_2 + \eta^3 + \xi^3 \varepsilon^2} K^2 (U_{i,k} U_{k,j} + U_{j,k} U_{k,i} - \frac{2}{3} \Pi \delta_{ij})$$

$$+ \frac{C_{\tau 2}}{A_2 + \eta^3 + \xi^3 \varepsilon^2} K^2 (U_{i,k} U_{j,k} - \frac{1}{3} \Pi \delta_{ij})$$

$$+ \frac{C_{\tau 3}}{A_2 + \eta^3 + \xi^3 \varepsilon^2} K^2 (U_{k,i} U_{k,j} - \frac{1}{3} \Pi \delta_{ij})$$

$$K_t + [U_j K - (\nu + \frac{\nu_t}{\sigma_K}) K_{,j}]_i = -\ddot{u}_i \ddot{u}_j U_{i,j} - \varepsilon$$

$$\varepsilon_t + [U_j \varepsilon - (\nu + \frac{\nu_t}{\sigma_\varepsilon}) \varepsilon_{,j}]_i = -C_1 \frac{\varepsilon}{K} \ddot{u}_i \ddot{u}_j U_{i,j} - C_2 \frac{\varepsilon^2}{K}$$
where
\[ \nu_t = \frac{2/3}{A_1 + \eta + \alpha \xi} \frac{K^2}{\epsilon} \]

\[ \eta = \frac{KS}{\epsilon}, \quad S = (2S_{ij} S_{ij})^{1/2}, \quad S_{ij} = \frac{1}{2}(U_{i,j} + U_{j,i}) \]

\[ \xi = \frac{K \Omega}{\epsilon}, \quad \Omega = (2\Omega_{ij}^{*} \Omega_{ij}^{*})^{1/2}, \quad \Omega_{ij}^{*} = \frac{1}{2}(U_{i,j} - U_{j,i}) + 4\epsilon_{mji} \omega_m \]

\[ C_{r1} = -4, \quad C_{r2} = 13, \quad C_{r3} = -2, \quad A_1 = 5.5, \quad \alpha = 0.1, \quad A_2 = 1000. \]

\[ C_1 = 1.44, \quad C_2 = 1.92, \quad \sigma_K = 1, \quad \sigma_\epsilon = 1.3 \]

**Solution Procedure**

- Conservative finite-volume method
- Non-staggered grids
- Momentum interpolation for velocity-pressure coupling
- Second-order accurate differencing schemes
- Quadratic terms in \( u_i u_j \) treated as source
- Strongly implicit solution algorithm of Stone
Performance in Complex Turbulent Flows

Diffuser flows

Backward-facing step flows

Confined jets

Confined swirling coaxial jets

**Diffuser flows**

- Fraser's case, $\alpha = 10^\circ$
- Trupp, Azad & Kassab's case, $\alpha = 8^\circ$
Wall pressure

Skin friction

SKE : Standard $K-\epsilon$ model
RNG : RNG-based $K-\epsilon$ model
RRSAE : Present model
KO : $K-\omega$ model
Centerline velocity

Shape factor
Wall pressure

Skin friction
Centerline velocity

Displacement thickness
Backward-facing step flows

- Driver & Seegmiller's case, $H_s/H_d = 1/8$
- Kim, Kline & Johnston's case, $H_s/H_d = 1/2$

Comparison of reattachment points

<table>
<thead>
<tr>
<th>case</th>
<th>experiment</th>
<th>K-ε</th>
<th>present model</th>
<th>RSM</th>
<th>ASM</th>
</tr>
</thead>
<tbody>
<tr>
<td>DS</td>
<td>6.1</td>
<td>4.99</td>
<td>5.82</td>
<td>-</td>
<td>5.66</td>
</tr>
<tr>
<td>KKJ</td>
<td>7 ±0.5</td>
<td>6.35</td>
<td>7.35</td>
<td>6.44</td>
<td>-</td>
</tr>
</tbody>
</table>
Wall pressure

![Graph of wall pressure for DS-case and KKJ-case](image)

Reynolds stresses (KKJ-case)

![Graphs of Reynolds stresses for different values of X](image)
Reynolds stresses

(DS-case)

Confined jets

- Barchilon & Curtet's case, $\theta = 0^\circ$
- Binder & Kian's case, $\theta = 2.5^\circ$
Wall pressure

Jet flow rate
Axial mean velocity profiles

BC-case

Turbulent shear stress (BK-case)
Separation and reattachment points (BC-case)

Recirculating flow rate (BC-case)
Confined swirling coaxial jets

- Roback & Johnson's case

![Diagram of flow patterns](image)

Centerline velocity

![Graph of centerline velocity](image)
Mean velocity profiles \((U, V, W)\)

![Mean velocity profiles](image)

Turbulent intensity profiles \((u', v', w')\)

![Turbulent intensity profiles](image)
Conclusions

- A general constitutive relation for $u_i u_j$ has been derived
  1. Its quadratic form seems sufficient
  2. It is realizable
- It significantly improves the performance of $K$-$\varepsilon$ based models
- Easy to implement
- No significant increase in computing time
- Very robust
MOTIVATION

- these models describe the effect of mean flow and external agencies (such as buoyancy) on the evolution of turbulence
- therefore, in principle, these models give a more accurate description of complicated flow fields than the two equation models
  - e.g. flows with large anisotropy in turbulence (such as near the leading edge of a turbine blade)
OBJECTIVE

• assess the performance of the various second order turbulence models in benchmark flows
• seek improvements where necessary
  - model for the pressure correlation term in the scalar flux equation
  - model for the scalar dissipation equation

Transport Equations for Second Moments

\[ \frac{D\bar{u}_i \bar{u}_j}{Dt} = P_{ij} + \Pi_{ij} + T_{ij} - D_{ij} \]
\[ \frac{Du_i \theta}{Dt} = P_i + \Pi_i + T_i \]
\[ \frac{D\bar{\theta}^2}{Dt} = P + T - D \]

These equations have to be closed by providing models for:

• Pressure correlation terms \((\Pi_{ij}, \Pi_i)\)
• Transport (Diffusion) terms \((T_{ij}, T_i)\)
• Dissipation terms \((D_{ij}, D)\)
HOW TO ASSESS MODELS?

Global computation

- Mean and turbulence equations are numerically solved

\[
\frac{D\bar{u}_i\bar{u}_j}{Dt} = p_{ij} + \ldots.
\]

- Results (e.g. Reynolds stresses) are then compared with experiments or DNS data

Direct comparison

Individual terms in the turbulence equations (such as pressure correlation terms) are directly compared with experiment or DNS data

Note that:

- In experiments pressure correlation terms can not be measured but can only be obtained indirectly through balance of second moment equations
- DNS allows direct computation of these correlations but is limited to low Reynolds number

Most of the results to be shown in this presentation are direct comparisons
Models for pressure correlation term in the scalar flux equation

- a. Launder model (1975)
- b. Zeman and Lumley model (1976)
  - linear in scalar flux
  - do not satisfy realizability
- d. Craft, Fu, Launder, Tselepidakis model (1989)
  - linear in scalar flux and Reynolds stress
  - satisfy realizability

Application to Homogeneous Shear Flow

(Experiment as well as DNS data)
Application to Round Buoyant Plume Flow Experiment

\[ \Delta T = f(r, z) \]

\[ W = g(r, z) \]

Shabbir and George Expt. (1990)
CONCLUSION

Models for pressure correlation term in scalar flux equation

- Models involving both scalar flux and Reynolds stress give better performance than the models which involve only scalar flux.

Models for pressure correlation term in the Reynolds stress equation

- a. Launder, Reece, Rodi model (1975)
- b. Speziale, Sarkar and Gatski model (1991)
  - linear or quasi-linear in Reynolds stress
  - do not satisfy realizability
- c. Shih and Lumley model (1985)
- d. Craft, Fu, Launder, Tselepidakis model (1989)
  - nonlinear in Reynolds stress
  - satisfy realizability
CONCLUSION

Models for pressure correlation term in Reynolds stress equation

- For the DNS data non-linear models give better performance than linear models. However, for the experiment no single model performs better for all the components.
- For the rapid part of the pressure correlation the relation $\Pi_{ij}^R = F(\overline{u_i u_j}, U_{i,j})$ is found to be adequate.

CONCLUSION (contd.)

- Performance of all the slow pressure correlation models varies from one flow to another.
- Furthermore, the relation $\Pi_{ij}^S = F(\overline{u_i u_j}, k, \epsilon)$ is inadequate in certain situations:
  - DNS shows that $\Pi_{ij}^S$ is dependent not only on the present time value of Reynolds stress but also on its past history.
  - Definition of $\Pi_{ij}^S$ implies that it is also a function of triple velocity moment $\overline{u_i u_j u_k}$.
- Therefore, more research is needed before any model for $\Pi_{ij}^S$ can be recommended for use.
A New Model Equation for Scalar Dissipation

- Traditional scalar dissipation rate equation is modeled in an analogue fashion to the mechanical dissipation equation
- Equation proposed here is modeled after the exact equation for scalar dissipation
- Its production/destruction mechanisms are different than the traditional model equation

Application to Homogeneous Benchmark Flows

1. Homogeneous turbulence subjected to constant scalar gradient
2. Homogeneous turbulence subjected to constant scalar gradient and constant shear

Global computation of the following two equations

\[ U_j \frac{\partial \theta^2}{\partial x_j} = -2u_i \theta \frac{\partial T}{\partial x_i} - 2\epsilon \theta \]

\[ U_j \frac{\partial \epsilon \theta}{\partial x_j} = C_{\theta 1} \epsilon \theta S + C_{\theta 2} \frac{\sqrt{\epsilon \theta \epsilon \Phi}}{\sqrt{Pr}} - C_{\theta 3} \frac{\epsilon \theta \epsilon}{k} \]

Mechanical field (i.e. \( k \), \( \epsilon \), etc.) and scalar flux, \( u_i \theta \), are taken as known. This way performance of the scalar dissipation equation is isolated.
CONCLUSION

- The transport equation for thermal dissipation rate proposed here gives improvement over the standard equation in at-least all the simpler benchmark flows. Its performance in the wall bounded flows is being assessed.
A Multiple-Scale Model for Compressible Turbulent Flows

presented by

William W. Liou

Workshop on Computational Turbulence Modeling September 15–16, 1993

Outline

☐ Motivation
☐ Model Development
  • formulation and determination of model coefficients
☐ Model Predictions:
  • compressible shear layer
  • compressible boundary layer
☐ Concluding Remarks
Motivation

- Incorporate the effects of non-equilibrium energy spectrum of compressible turbulence in computational models

DNS of Compressible Turbulence

- Formation of high gradient regions or eddy shocklets
- Enhanced vortex stretching and spectral energy transfer
- Non-equilibrium turbulent kinetic energy spectrum due to the flow compressibility
Multiple-Scale Model

Model Equations

- Large Scale

\[
\frac{\bar{D}k_p}{Dt} = \frac{\partial}{\partial y} \left[ (\bar{\mu} + \frac{\mu_T}{\sigma_{k_p}}) \frac{\partial \bar{k}_p}{\partial y} \right] + \mu_T \left( \frac{\partial \bar{u}}{\partial y} \right)^2 - \bar{\rho} \bar{\epsilon}_p + f_{c_1}
\]

\[
\frac{\bar{D}\bar{\epsilon}_p}{Dt} = \frac{\partial}{\partial y} \left[ (\bar{\mu} + \frac{\mu_T}{\sigma_{\bar{\epsilon}_p}}) \frac{\partial \bar{\epsilon}_p}{\partial y} \right] + C_p \frac{\bar{\epsilon}_p}{k_p} \mu_T \left( \frac{\partial \bar{u}}{\partial y} \right)^2 - C_{p_2} \frac{\bar{\epsilon}_p^2}{k_p} + f_{c_2}
\]

- \(f_{c_1}\) – exchanges between the turbulent kinetic energy and internal energy

- \(f_{c_2}\) – increased spectral energy transfer due to eddy-shocklets
Model Equations (cont.)

□ Small Scale

\[ \frac{\partial}{\partial y} \left[ \frac{\mu + \frac{\mu_T}{\sigma_{k_t}}}{} \right] \partial k_t \right] + \frac{\partial \tilde{\varepsilon}_p}{\partial y} - \frac{\partial \tilde{\varepsilon}_t}{\partial y} \]

\[ \frac{\partial}{\partial y} \left[ \frac{\mu + \frac{\mu_T}{\sigma_{\varepsilon_t}}}{} \right] \partial \tilde{\varepsilon}_t \right] + \frac{C_t_1 \tilde{\varepsilon}_t \tilde{\varepsilon}_p}{k_t} - \frac{C_t_2 \tilde{\varepsilon}_t}{k_t} \]

□ Eddy Viscosity

\[ \mu_T \approx \tilde{\rho} \tilde{\mu} \approx \tilde{\rho}(\tilde{k}_p + \tilde{k}_t)^{\frac{1}{2}} \left( \frac{(\tilde{k}_p + \tilde{k}_t)^{\beta}}{\tilde{e}_p} \right) \]

Model Coefficients

□ Decaying turbulence and homogeneous turbulence

\[ C_{p_1} = (1 - \frac{\beta}{\alpha}) + \frac{\beta}{\alpha} C_{p_2}, \quad C_{p_2} = \frac{n + 1}{n} \]

\[ C_{t_1} = 1 - \frac{1}{\beta} + \frac{C_{t_2}}{\beta}, \quad C_{t_2} = \frac{\beta - 1 + C_{p_2} \beta \frac{\tilde{k}_t}{k_p}}{\beta + \beta \frac{\tilde{k}_t}{k_p} - 1} \]

<table>
<thead>
<tr>
<th>$C_\mu$</th>
<th>$\sigma_{k_p}$</th>
<th>$\sigma_{k_t}$</th>
<th>$\sigma_\varepsilon$</th>
<th>$\sigma_\tilde{\varepsilon}$</th>
<th>$n$</th>
<th>$\alpha$</th>
<th>$\beta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.0</td>
<td>1.0</td>
<td>1.3</td>
<td>1.3</td>
<td>1.2</td>
<td>2.2</td>
<td>1.05</td>
</tr>
</tbody>
</table>
Model Equations (cont.)

- Large Scale
  - $f_{c1}$ - Sarkar et al. (1992)
    \[
    f_{c1} = -\alpha_2 M_t \mu_T \left( \frac{\partial \bar{u}}{\partial y} \right)_T^2 + \alpha_3 M_t^2 \bar{p} \bar{e}_p
    \]
  - $f_{c2}$ - dimensional analysis
    \[
    f_{c2} \equiv f_{c2}(\bar{p}, \bar{\rho}, \bar{k}, \bar{\varepsilon}, \bar{\mu})
    \]
    \[
    \Pi_1 = \frac{\bar{p}}{\bar{\rho}k} = \frac{1}{\gamma M_t^2}, \quad \Pi_2 = \frac{\bar{\mu} \bar{\varepsilon}}{\bar{\rho}k^2} = \frac{1}{Re_t}
    \]
    \[
    f_{c2} = b_1 + b_2 M_t^2 + O(M_t^4)
    \]

Model Predictions

- Compressible Free Shear Layers
  - mean velocity profiles. $M_c=0.51$
  - peak Reynolds shear stresses. $M_c=0 \sim 1.6$
  - vorticity thickness growth rates
Multiple-Scale Model

\( \square \) Wall Function

\[
\frac{u_c}{u^+} = \frac{1}{\kappa} \ln (y^+) + 5.2, \quad q_w = -\frac{\mu_t C_p}{Pr_t} \frac{dT}{dy} + u \tau_w
\]

\[
\tilde{k} = \left( \frac{\tau_w}{\rho} \right)/0.3, \quad \tilde{\varepsilon} = \left( \frac{\tau_w}{\rho} \right)^{3/2}
\]
Model Predictions

Compressible Boundary Layers

- mean velocity profiles. $M_\infty=2.831$, $Re_\theta=420,700$
- mean temperature profiles.
- skin friction coefficients. $M_\infty=0 \sim 5$, $Re_\theta=10^4$
Concluding Remarks

☐ The present compressible multiple-scale model predicts correctly both the spreading rate of compressible shear layers and the skin friction coefficient of compressible boundary layers.

☐ Need to implement the model into the calculation of more complex compressible turbulent flows.
PDF TURBULENCE MODELS FOR REACTIVE FLOWS

Presented by

A.T. Hsu

OUTLINE

1. Motivation: why PDF

2. Works accomplished:
   
   . Model development.
   
   . Numerical results.

3. Future studies
CLOSURE PROBLEM:

\[
  u_i = \bar{u}_i + u'_i, \\
  Y_i = \bar{Y}_i + Y'_i,
\]

\( \bar{u}_i u_j \) — Turbulence Modeling

\( \bar{Y}_i u_j \) — Analogy of shear stress: diffusion model.

\( \rho \bar{w}_i \) — ???

\[
  \rho w_i = \rho w(Y_1, ..., Y_n, T, \rho)
\]

But in general:

\[
  \rho \bar{w}_i \neq \rho w(Y_1, ..., Y_n, T, \rho)
\]

EXAMPLE: REACTION RATE FROM MEAN TEMPERATURE

\[
  \text{CH}_4 + \text{H}_2\text{O} \rightleftharpoons \text{CO}_2 + \text{H}_2
\]

![Diagram showing reaction rate](image)
PDF Method - Motivation

Avoid Chemical Closure Problem

- Probability Density Function \( P(Y_1, ..., Y_n, T, \rho) \)

\[
\bar{\rho} Y_i = \int \cdots \int \rho \omega_i(Y_1, ..., Y_n, T, \rho) P(Y_1, ..., Y_n, T, \rho) dY_1, ..., dY_n, d\rho
\]

\[
\bar{Y}_i = \int \cdots \int Y_i P(Y_1, ..., Y_n, T, \rho) dY_1, ..., dY_n, d\rho
\]

- Direct Solution of \( P(Y_1, ..., Y_n, T, \rho) \)

WORK ACCOMPLISHED

Developed continuous mixing model.

Developed compressible flow pdf model that can treat flows with shocks.

Developed 2D and 3D Monte Carlo pdf solvers.

Coupled pdf solver with the RPLUS code and made calculations on realistic combustion problems.
WORK ACCOMPLISHED

Numerical Results

1. Heated turbulent jet
2. H2-F2 non-premixed flame
3. Oblique shock
4. Supersonic wall jet flame
5. Supersonic round jet flame
6. 3D jet-in-crossflow, supersonic flame
III. Results

(U) The sketch of a heated plane jet is given on this page. The flow domain is divided into 10 cells in the cross direction, and 100 samples are assigned to each cell. The numerical results from the present study are compared with experimental data in the following figures.
Figure 5. Mean temperature in heated plane jet. — continuous mixing model, - - modified Curl model, Δ Browne et al. (1984), ○ Bashir and Uberoi (1975), o Uberoi and Singh (1975), o Jenkins and Goldschmidt (1973).

Figure 6. RMS of temperature variance in heated plane jet. — continuous mixing model, - - modified Curl model, ○ Antonia et al. (1983), ○ Bashir and Uberoi (1975), Δ Uberoi and Singh (1975).
COMPRESSIBLE EFFECT

The pdf solution of temperature rise caused by an oblique shock:

Fig. 2 probability mass function

\[ I_{\Delta T}(\Delta T; \Delta T_{ad}) \]

- \( y/\delta_x = 0.42 \)
- \( y/\delta_x = 0.15 \)
HYDROGEN WALL JET WITHOUT PDF

Fig. 6 Flow Configuration of Burrows and Kuchow's Experiment.

Fig. 7a Temperature Contour of Numerical Solution Using PDF.

Fig. 7b Hydrogen Mass Fraction Contour of Numerical Solution Using PDF.

Fig. 7c Oxygen Mass Fraction Contour of Numerical Solution Using PDF.
HYDROGEN WALL JET
PDF SOLUTION

COAXIAL HYDROGEN JET
(Beach)

D = 0.0653 m
d = 0.009525 m
Injector lip thickness = 0.0015 m

<table>
<thead>
<tr>
<th>Mass fraction:</th>
<th>Hydrogen jet</th>
<th>Free stream</th>
</tr>
</thead>
<tbody>
<tr>
<td>H₂</td>
<td>1.000</td>
<td>0</td>
</tr>
<tr>
<td>O₂</td>
<td>0.251</td>
<td>0.478</td>
</tr>
<tr>
<td>N₂</td>
<td>0</td>
<td>0.281</td>
</tr>
<tr>
<td>Water</td>
<td>0</td>
<td>0.281</td>
</tr>
</tbody>
</table>

Mačn number, M . . . . . . 2.00 1.90
Temperature, T, K . . . . 251 1495
Velocity, u, m/s . . . . . 2432 1510
Pressure, p, MPa . . . . . 0.1 0.1
COAXIAL HYDROGEN JET
WITHOUT PDF

Fig. 3a. Radial profiles of the predicted and measured mass fractions of major species for the CFD solver at x/d = 8.25.

Fig. 3c. Radial profiles of the predicted and measured mass fractions of major species for the CFD solver at x/d = 21.7.

Fig. 3b. Radial profiles of the predicted and measured mass fractions of major species for the CFD solver at x/d = 15.5.

Fig. 3d. Radial profiles of the predicted and measured mass fractions of major species for the CFD solver at x/d = 27.9.

COAXIAL HYDROGEN JET
PDF SOLUTION

Fig. 4a. Radial profiles of the predicted and measured mass fractions of major species for the pdf-CFD solver at x/d = 8.25.

Fig. 4c. Radial profiles of the predicted and measured mass fractions of major species for the pdf-CFD solver at x/d = 21.7.

Fig. 4b. Radial profiles of the predicted and measured mass fractions of major species for the pdf-CFD solver at x/d = 15.5.

Fig. 4d. Radial profiles of the predicted and measured mass fractions of major species for the pdf-CFD solver at x/d = 27.9.
than a pure CFD approach for reaching flows.

The pdl method performs consistently better.

existing CFD flow solvers with equal ease.

The pdl solver can be coupled with any

and coupling with CFD solvers feasible.

Efficiency and convergence of the pdl solver

sample in the Monte-Carlo computation.

Scheme allows one to use a relatively small

a combined time- and ensemble-averaging

been developed and implemented.

A compressible pdl influence model has

CONCLUSIONS
Future Research

Make realistic 3D combustion predictions.

Develop more accurate pdf models for compressible flames.

Use DNS data to validate new models.
Page intentionally left blank
ANALYTICAL THEORIES OF TURBULENCE
APPLIED TO TURBULENCE MODELING

presentation by
Robert Rubinstein
CMOTT

MOTIVATION

Turbulence models contain undetermined elements

- constants
- functions
- model forms

Apply analytical theories to determine these elements:

- direct interaction approximation (DIA) (Kraichnan, 1959)
- renormalization group (RNG) (Yakhot and Orszag 1986)
EXAMPLE 1

Determine constants in nonlinear eddy viscosity (NLEV) (Yoshizawa 1984) and nonlinear eddy diffusivity (Yoshizawa 1987) models

\[
< \mathbf{u} \mathbf{u} > \equiv \frac{2}{3} k I = C_\nu \frac{k^2}{\epsilon} (\nabla U + \nabla U^T)
+ C_{\tau 1} \frac{k^3}{\epsilon^2} (\nabla U \nabla U^T - \frac{1}{3} \nabla U : \nabla U^T)
+ C_{\tau 2} \frac{k^3}{\epsilon^2} (\nabla U \nabla U + \nabla U^T \nabla U^T - \frac{2}{3} \nabla U : \nabla U)
+ C_{\tau 3} \frac{k^3}{\epsilon^2} (\nabla U^T \nabla U - \frac{1}{3} \nabla U^T : \delta U)
\]

\[
< \theta \mathbf{u} > = C_\kappa \frac{k^2}{\epsilon} \nabla \theta + C_{\kappa 1} \frac{k^3}{\epsilon^2} \nabla U \cdot \nabla \theta + C_{\kappa 2} \frac{k^3}{\epsilon^2} \nabla U^T \cdot \nabla \theta
\]

Universal inertial range constants for isotropic turbulence:

\[
E(k) = C_K \epsilon^{2/3} k^{-5/3}
\]

\[
\nu(k) = C_D \epsilon^{1/3} k^{-4/3}
\]

\[C_K = 1.61 \hspace{1cm} C_D = 0.49 \hspace{1cm} \text{(Yakhot and Orszag 1986)}\]

Assume shear is a weak perturbation of isotropic background state. Then (Rubinstein and Barton 1990):

<table>
<thead>
<tr>
<th>exact</th>
<th>value</th>
<th>experiment</th>
</tr>
</thead>
<tbody>
<tr>
<td>(C_\nu)</td>
<td>(\frac{2}{3} C_D / C_K^2)</td>
<td>0.085</td>
</tr>
<tr>
<td>(C_{\tau 1})</td>
<td>(\frac{5}{105} \left( \frac{2}{3} \right)^2 / (C_D C_K)^2)</td>
<td>0.034</td>
</tr>
<tr>
<td>(C_{\tau 3})</td>
<td>(-\frac{2}{105} \left( \frac{2}{3} \right)^2 / (C_D C_K)^2)</td>
<td>(-.014)</td>
</tr>
</tbody>
</table>
Passive scalar results

\[ Pr_T = f(Pr, Re) \] (Yakhot and Orszag 1986)

\( f \) is a known function: for \( Re = \infty \), \( Pr_T \sim 0.7 \)

\( C_\kappa = C_\nu / Pr_T \)

Constants of the nonlinear theory are known functions of \( C_D, C_K, Pr_T \)
(Rubinstein and Barton 1991)

Diffusivity ratio \( \kappa_{12}/\kappa_{22} = 1.9 \) (Tavoularis and Corrsin), 2.3 (theory)

In principle, higher order nonlinearities in NLEV can be computed theoretically

EXAMPLE 2

Determine model form for rapid pressure strain correlation \( \Pi \)

RNG gives a series for \( \Pi \) in powers of \( \nabla U \).
The LRR model

\[ \Pi = C_{r1} \langle uu \rangle \nabla U + C_{r2} \langle uu \rangle \nabla U^T \]

arises from systematic lowest order consolidation (resummation) of this series. (Rubinstein and Barton 1992):

LRR            RNG

\begin{align*}
C_{r1} & \quad 0.7636 \quad \frac{15}{21} \sim 0.7619 \\
C_{r2} & \quad 0.1091 \quad \frac{2}{21} \sim 0.0952
\end{align*}
Second order resummations are possible:

$$\Pi = F(< uu >)\nabla U + G(< uu >)\nabla U^T$$

$$\Pi = < uu > \left[ C_1 \nabla U + C_2 \nabla U \nabla U + \cdots \right]$$

These are speculative at this point.

EXAMPLE 3
Determine unknown functions in generalized nonlinear eddy viscosity representation for simple shear flow

Nonlinear eddy viscosity does not apply at large strain rates: it predicts $< uu > / k \to \infty$. Introduce the generalization

$$< uu > - \frac{2}{3} k I = C_\nu(\eta) \frac{k^2}{\varepsilon} (\nabla U + \nabla U^T) + \cdots$$

where in simple shear flow $\partial U_i / \partial x_j = S \delta_{i1} \delta_{j2}$, $\eta = S k / \varepsilon$
DIA analysis of shear turbulence leads to

\[ < u_i u_j > = kF_{ij}(\eta/C_R) \]

where \( F_{ij} \) is known from rapid distortion theory (RDT) (Maxey 1982) and \( C_R \) is a universal constant (\( C_R \sim 1.59 \), Yakhot and Orszag 1986).

General time dependent theory

\[ < u_i u_j > (t) = kF_{ij}(\alpha^*(t)) \]

where \( \alpha^* \) is a modified total strain determined by DIA:

\[ \alpha^*(t) = \int_0^t ds \ G(t-s)S(s) \]

\( G \) is the response function of DIA.

Homogeneous shear flow at high strain rate

\[ \eta = Sk/\varepsilon \rightarrow \infty \]

\[
\begin{array}{cccc}
 b_{12} & b_{11} & b_{22} & b_{33} \\
 NLEV & O(\eta) & O(\eta^2) & O(\eta^2) & O(\eta^2) \\
 LRR & O(\eta^{-1}) & f_{11}(c) & f_{22}(c) & f_{33}(c) \\
 present & O(\eta^{-1}) & 2/3 & -1/3 & -1/3 \\
 RDT & O(\eta^{-1}) & 2/3 & -1/3 & -1/3 \\
\end{array}
\]

In LRR, \( f_{ij}(c) \) are model dependent constants. The limiting normal stresses are not necessarily realizable.

DNS (Lee, Kim and Moin) shows that RDT is a good description of these flows even for moderate times.
CURRENT WORK

Time dependent model for nonequilibrium turbulence based on DIA

New evaluation of NLEV constants

Nonmarkovian damping: short time suppression of eddy damping.
Required for oscillating and highly strained flows.

Two scale theory of shear turbulence:
large scales \sim RDT,
small scales \sim Kolmogorov inertial range dynamics
MOTIVATION

Provide better understanding of turbulence and transition physics.

DNS OF TURBULENCE, TRANSITION AND EFFECT OF ROTATION

Presented by

A.T. Hsu
DNS for Bypass Transition: Numerical Aspects

- Fully implicit, high-order finite difference scheme
  (fourth order in space and second order in time.)

- Acceleration Techniques:
  - Semi-coarsening multigrid
  - Line-box distributive relaxation

- A new buffer outflow boundary condition

- A fine-coarse-fine grid mapping
DNS for Bypass Transition: Cases Calculated

- 2D/3D linear instability stage (temporal simulation)
- 2D/3D linear instability stage (spatial simulation)
- Secondary instability stage (spatial simulation)
- Simulation of the whole transition zone (spatial simulation)
DNS: Effect of Rotation

Numerical Method:

- Compressible N-S equations;
- 8th order compact difference scheme;
- 3rd order time marching.

Results:

- Compressible, homogeneous, isotropic turbulence subject to constant rotation.

Contours plots of total vorticity magnitude on the $y = 0.0477$ $(x,z)$-plane. $Re^* = 500$, $Fr = 60$, $\beta = 0.1$, $\epsilon_{14} = 0.01$, $\epsilon_{16} = 0.01$. Flow direction is from left to right.
EQUATIONS FOR ROTATING TURBULENCE

The Reynolds stress equations:

\[
(u_i u_j)_{,t} = -u_k (u_i u_j)_{,k} - \frac{1}{\rho} (u_{i,p,j} + u_{j,p,i}) + \bar{u}_{i}^{\tau} \delta_{jk,k} \\
+ \bar{u}_{j}^{\tau} \delta_{ik,k} - 2 \epsilon_{ilm} \Omega_{l} \bar{u}_{m,j} - 2 \epsilon_{ilm} \Omega_{l} \bar{u}_{m} \bar{u}_{i}
\]

Turbulent kinetic energy:

\[
k_{,t} = \frac{1}{2} u_{i,j} u_{j,i} - u_{i,p,i} - \nu u_{i,j} u_{i,j} - \frac{\nu}{3} u_{i,i} u_{j,j}
\]

Incompressible dissipation:

\[
\epsilon_{,t} = -2 \nu u_{i,j} u_{k,j} u_{i,k} + \nu u_{i,j} u_{i,j} u_{k,k} - 2 \nu u_{i,j,p,i j} \\
- 2 \nu^2 u_{i,j} u_{k,k} u_{i,j,k} - \frac{2}{3} \nu^2 u_{i,j} u_{k,k} u_{i,j,k}
\]

TRIPLE CORRELATION EQUATIONS

\[
(u_i u_j u_{j,j})_{,t} = -\epsilon_{jlm} \Omega_{j} u_{i} u_{i} u_{m,j} + OT
\]

\[
(u_i u_j u_{j, l})_{,t} = -2 \epsilon_{l,m} \Omega_{m} u_{n,j} u_{i,j} + OT
\]

\[
(u_i u_j u_{i,j})_{,t} = -2 \epsilon_{jlm} \Omega_{l} u_{m,j} u_{i,j} + OT
\]

Effects of triple correlations: redistribute energy and dissipation.
Fig. 10 Time spectra of turbulent kinetic energy.

Fig. 11 One-dimensional energy spectra for the flow field at $t/t_{edd} = 1$.

Fig. 1 Time history of Reynolds stresses for $\Omega = 0$.

Fig. 2 Time history of Reynolds stresses for a moderate rotation rate of $\Omega = 2.74$.

Fig. 3 Time history of Reynolds stresses for a high rotation rate.
DNS: Reactive Flows

Numerical Method:

- Compressible N-S equations;
- 8th order compact difference scheme;
- 3rd order time marching;
- finite rate chemistry.

Results:

- H2 + Air combustion in homogeneous turbulence with or without shear.
DNS: Reduced H2 Chemistry

Temperature [K]

DNS: Reduced H2 Chemistry

NO Mass Fraction

106
**Presentation of the Problem**

- Variables
- Meaning of Bypass Transition
- Physics/Mechanisms to be Modeled

** Modeling of Transition Onset**

- Use of DNS
- Use of $k$-$\varepsilon$ Model
- Use of PSE
- Summary of Results

** Modeling of the Transition Region**

- Comment on Linear-Combination Method
- Use and Limits of $k$-$\varepsilon$ Methods
- Schmidt/Patankar Method
- Multi-Time-Scale Method
- Intermittency Method
- Reynolds Analogy
- Summary of Results

**Recommendations**
Effect of Streamwise Curvature on Bypass.
(Wang, 1984; Kim and Simon, 1991)

Transition Wall Radii of Curvature 90-100 CM; Free-Stream Distribution Level 0.6-0.7 Percent.

![Graph showing Stanton number vs. X-Reynolds number](image)

- **Concave wall**
- **Flat wall**
- **Convex wall**

---

**Idealized Sketch of Transition Process on a Flat Plate**
Linear Versus Bypass Path to Transition, From Suder, O'Brien, Reshotko (1988)

\[ U_\infty = 100 \text{ ft/sec} \]

LAMINAR BOUNDARY LAYER | TURBULENT B.L.

TRANSITION

**TU = 0.3%**

\[
\frac{Re}{1000}
\]

1402 1501 1600 1693 1696 1906 2294

Time, ms

**TU = 1%**

\[
\frac{Re}{1000}
\]

209.1 309.3 407.5 506.9 606.5 706.0 805.2 904.7

Time, ms

Computed Skin Friction Along the Length of the Flat Plate (Grids A and B), (Rai and Moin, 1991)

\[ \frac{8 \times 10^{-3}}{C_f} \]

0 2.5 5.0 7.5 10.0 12.5 \times 10^5

Re_x

- Computation (Grid A)
- Computation (Grid B)
- Experiment (Suder et al.)
- Experiment (Sohn et al.)

Turbulent correlation

Blasius (laminar)
Comparison of Computed Disturbance Energy from Computations Based on a $k-\varepsilon$ Model and on Direct Numerical Simulation (DNS) for Bypass Transition (Simoneau and Simon, 1993)

Onset Prediction Using PSE Approach (Stuckert & Herbert, 1992)  
Data of Sohn & Reshotko (1991)
Computed and Experimental Momentum Thickness Reynolds Number for transition Onset

- Kim (1990)
- Sohn (1991)
- Blair and Anderson (1987) $k = 0.2 \times 10^{-6}$
- Blair and Anderson (1987) $k = 0.75 \times 10^{-6}$
- Experiments
  - (1) Volino, Simon (1991)
  - (2) Crawford
  - (2) Crawford
  - (1) Simon, Stephens (1991)

- MTS
- TXM
- Jones-Launder

- Computed
  - (1) Based on intermittency
  - (2) Based on minimum Stanton number

- Launder-Sharma
- Nagano-Tagawa
- K.Y. Chien

- Abu-Ghannam and Shaw (1980)

---

CD-93-58127
The Numerical Prediction of Transition Flow Using MTS Model Compared with Experimental Data of T3A (Crawford, 1993)
Intermittency (Volino and Simon, 1991)

Transition Length Determination

\[ Re_{tr} = \frac{2.15}{\sqrt{N}} \left( \frac{Re_{tr}}{Re} \right)^{3/4} \]

\[ N = \text{Non-Dimensional Spot Formation Rate} \]

\[ = \frac{n \theta^3}{\nu} \]

\[ \theta = \text{Spot formation rate} \]

\[ \nu = \text{Dependence factor} \]

\[ \theta_k = \text{Momentum thickness} \]

\[ \gamma = 0 \to 0.99 \]
Variation of Nondimensional Spot Formation Rate with Free-Stream Turbulence (Narasimha (1985) method)

Transition Length Correlation

Equation 1, $N = 0.3 \times 10^{-3}$
($\alpha = 11^\circ$, $\beta = 0.65$)

Equation 1, $N = 0.039 \times 10^{-3}$
($\alpha = 5^\circ$, $\beta = 0.8$)
Use of Intermittency to Model Transition Region

With intermittency, $Tu = 2.8\%$

Without intermittency, $Tu = 2.8\%$

Use of an Intermittency Weighting Factor in Computations (Yang, 1992; Yang and Shih, 1992)

Skin friction variation for T3A

Skin friction variation for T3B
REYNOLDS ANALOGY EFFECTS

Problem: Reynolds analogy breakdown in flows with pressure gradient

Possible Solution: Use of additional modeled transport equations for the thermal turbulence to predict the turbulent heat flux/turbulent Prandtl number

Computed and Experimental Momentum Thickness Reynolds Number for Transition End

- **Kim (1990)**
- **Sohn (1991)**
- **Blair and Anderson (1987) k = 0.2x10^-6**
- **Blair and Anderson (1987) k = 0.75x10^-6**

- **MTS**
- **TXM**
- **Jones-Lauder**
- **K.Y. Chien**
- **Lauder-Sharma**
- **Nagano-Tagawa**

- **Simone, Stephens (1991)**
- **(1) Based on intermittency**
- **(2) Based on minimum Stanton number**

- **Crawford (1991)**
- **Sieger, Schulz, Crawford, Wittig (1992)**
- **(2) Crawford**
- **(2) Crawford**

- **(1) Experiments**
- **(2) Computations**

- **Abu-Ghannam and Shaw (1980)**

CD-93-66131

116
RECOMMENDATIONS

- Develop a DNS data base with/without heat transfer for turbine blade geometry for (1) understanding physics (2) supporting/guiding experiments (3) developing/testing models
- Continue development of models more faithful to physics
- Continue development of Multi-Time-Scale Model
- Develop Reynolds Stress Turbulence Models
- Initiate an effort in the use of Large Eddy Simulation (LES)
- Develop the PSE approach as design tool
- Develop models for unsteady flows (e.g. wakes)
- Develop capability to predict turbulent Prandtl number
- Test models for turbulence levels found in combustors (6-20%)
Page intentionally left blank
ALGEBRAIC MODELS FOR TURBINE BLADE HEAT TRANSFER

R. J. Boyle
NASA Lewis Research Center

Aim and Scope

- VALIDATE PREDICTIONS OF TURBINE BLADE HEAT TRANSFER & PRESSURE LOSS BETWEEN SMOOTH AND ROUGH BLADES
- ALGEBRAIC - BALDWIN-LOMAX TYPE MODELS
- IMPLEMENTED IN NAVIER-STOKES CODES - RVC3D & RVCQ3D
- ENGINEERING APPLICATIONS
• FREESTREAM TURBULENCE - ex LEADING EDGE
• TRANSITION MODEL - ex MAYLE'S
• ALGEBRAIC - BALDWIN-LOMAX TYPE MODELS

Tu∞ EFFECT ON LEADING EDGE HEAT TRANSFER

FORREST'S Tu∞ MODEL

EXPERIMENTAL HEAT TRANSFER DATA

○ DRING ET AL.
△ HIPPENSTEEL ET AL.
□ HYLTON ET AL.
a) Data of Blair, $Re = 2.37 \times 10^5$, Design Flow Angles.

b) Prediction, 1% Clearance.

**Appropaches for Rough Surfaces**

- **Taylor, Coleman, and Hodge**
  Explicitly account for blockage, drag and heat transfer source terms in Navier-Stokes equations

- **Cebeci and Chang** - used in analysis
  Modify mixing length
  Only eddy viscosity affected
REQUIRE ROUGHNESS PARAMETERS

- **HEIGHT**
- **SPATIAL DENSITY**
- **SHAPE**

---

**CORRELATION FOR EQUIVALENT HEIGHT RATIO FROM SIGAL and DANBERG**

![Diagram showing correlation between equivalent height ratio and roughness density parameter.]

---

Workshop on Computational Turbulence Modeling
EXPERIMENTAL DATA USED FOR COMPARISONS

- **HEAT TRANSFER**
  - HOSNI - FLAT PLATE
  - BLAIR & ANDERSON - LARGE LOW SPEED ROTOR
  - DUNN & KIM - SSME FUEL TURBINE

- **TURBINE EFFICIENCY**
  - BOYNTON - TWO-STAGE SSME HIGH PRESSURE FUEL TURBINE

---

**ROUGH FLAT PLATE HEAT TRANSFER**
(Data of Hosni et al.)

![Graph showing Stanton number vs. Reynolds number for rough flat plates with different L/D ratios and h_{eq}/h values.](image)

- L/D
  - 2
  - 4
  - 10
  - Smooth

---

*Workshop on Computational Turbulence Modeling*
RATIO OF PREDICTED TO MEASURED HEAT TRANSFER
FOR A FLAT PLATE. Re_x=400,000.
(Data of Hosni et al.)

<table>
<thead>
<tr>
<th>L/D</th>
<th>h^eq/h</th>
<th>St_p/St_m</th>
<th>h^eq/h</th>
<th>St_p/St_m</th>
<th>h^eq/h</th>
<th>St_p/St_m</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1.58</td>
<td>1.64</td>
<td>2.43</td>
<td>1.85</td>
<td>1.69</td>
<td>1.67</td>
</tr>
<tr>
<td>4</td>
<td>0.26</td>
<td>1.36</td>
<td>0.50</td>
<td>1.57</td>
<td>0.45</td>
<td>1.52</td>
</tr>
<tr>
<td>10</td>
<td>0.023</td>
<td>1.01</td>
<td>0.061</td>
<td>1.15</td>
<td>0.079</td>
<td>1.22</td>
</tr>
<tr>
<td>Smooth</td>
<td>1.10</td>
<td>1.10</td>
<td>1.10</td>
<td>1.10</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Workshop on Computational Turbulence Modeling

PREDICTED AND MEASURED BLADE HEAT TRANSFER
(Data of Blair and Anderson)

Workshop on Computational Turbulence Modeling
EFFICIENCY FOR SMOOTH AND ROUGH BLADES

TWO-STAGE TURBINE EFFICIENCY, $\eta$

NORMALIZED WHEEL SPEED, $U_m/V_{\text{IDEAL}}$

N-S PREDICTION

- SMOOTH
- ROUGH

DATA-BOYNTON et al.

- SMOOTH
- ROUGH

GRID & MODEL ASSUMPTION EFFECTS

<table>
<thead>
<tr>
<th>Grid</th>
<th>$y^+$</th>
<th>$K$</th>
<th>$A^+$</th>
<th>$\delta$</th>
<th>$\Delta\eta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>291x54</td>
<td>0.3</td>
<td>0.0</td>
<td>26.0</td>
<td>0.0321</td>
<td>0.011</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0</td>
<td>1.0</td>
<td>0.0483</td>
<td>0.002</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.3</td>
<td>26.0</td>
<td>0.0348</td>
<td>0.015</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.3</td>
<td>1.0</td>
<td>0.0550</td>
<td>0.009</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0</td>
<td>26.0</td>
<td>0.0455</td>
<td>0.026</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0</td>
<td>1.0</td>
<td>0.0703</td>
<td></td>
</tr>
<tr>
<td>291x54</td>
<td>0.6</td>
<td>0.0</td>
<td>26.0</td>
<td>0.0423</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0</td>
<td>1.0</td>
<td>0.0570</td>
<td>0.010</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.3</td>
<td>26.0</td>
<td>0.0449</td>
<td>0.002</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.3</td>
<td>1.0</td>
<td>0.0654</td>
<td>0.016</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0</td>
<td>26.0</td>
<td>0.0546</td>
<td>0.008</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0</td>
<td>1.0</td>
<td>0.0767</td>
<td>0.023</td>
</tr>
<tr>
<td>291x54</td>
<td>1.2</td>
<td>0.0</td>
<td>26.0</td>
<td>0.0375</td>
<td>0.011</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.0</td>
<td>1.0</td>
<td>0.0531</td>
<td>0.002</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.3</td>
<td>26.0</td>
<td>0.0402</td>
<td>0.014</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.3</td>
<td>1.0</td>
<td>0.0586</td>
<td>0.009</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0</td>
<td>26.0</td>
<td>0.0502</td>
<td>0.024</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.0</td>
<td>1.0</td>
<td>0.0730</td>
<td></td>
</tr>
</tbody>
</table>

Experimental: 0.021
• TEMPERATURE MEASUREMENTS BENEATH ROUGHNESS AFFECT HEAT TRANSFER COEFFICIENT

• \( h_{\text{ACTUAL}} / h_{\text{MEASURED}} = 1 + \frac{ht}{k} \)
  
  \( h \) - HEAT TRANSFER COEFFICIENT  
  \( t \) - ROUGHNESS THICKNESS  
  \( k \) - THERMAL CONDUCTIVITY OF ROUGHNESS

• POSSIBLE REASON FOR HIGHEST AUGMENTATION CORRESPONDING TO LARGEST SCALE

---

REQUIREMENTS FOR BENCHMARK ROUGHNESS HEAT TRANSFER DATA SETS

• COMPLETE DESCRIPTION OF THE ROUGHNESS CHARACTERISTICS

• SURFACE NOT DISTURBED BY MEASUREMENT TECHNIQUE

• KNOWN TEMPERATURE OF ROUGHNESS ELEMENTS

• VERIFICATION OF ACCURACY BY MEASUREMENTS ON SMOOTH SURFACES
<table>
<thead>
<tr>
<th>LeRC</th>
<th>9/93</th>
<th>CONCLUSIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>• SMOOTH SURFACES</td>
</tr>
<tr>
<td></td>
<td></td>
<td>REASONABLE ACCURACY</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TRANSITION BEHAVIOR GOVERNS</td>
</tr>
<tr>
<td></td>
<td></td>
<td>ACCURACY</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• ROUGH SURFACES</td>
</tr>
<tr>
<td></td>
<td></td>
<td>INCONSISTENT ASSUMPTIONS</td>
</tr>
<tr>
<td></td>
<td></td>
<td>NEEDED FOR LOSS AND HEAT</td>
</tr>
<tr>
<td></td>
<td></td>
<td>TRANSFER</td>
</tr>
</tbody>
</table>

*Workshop on Computational Turbulence Modeling*
Page intentionally left blank
TURBINE HEAT TRANSFER

OUTLINE

- Brief description of the method of solution
- 2-d examples using various models
- 3-d calculations using algebraic model and q-ω model
- Conclusions
TURBINE HEAT TRANSFER

TURBULENCE MODELS

LOW REYNOLDS NUMBER TWO-EQUATION MODELS
- COAKLEY'S $q-\omega$ MODEL
- CHIEN'S $K-\varepsilon$ MODEL

ALGEBRAIC MODEL
- BALDWIN-LOMAX -- WITHOUT TRANSITION MODEL

FORMULATION
- MASS AVERAGED COMPRESSIBLE N-S EQUATIONS
- + MODEL EQUATIONS
- NON-PERIODIC C GRID.
TURBINE HEAT TRANSFER

SOLUTION OF THE NAVIER-STOKES EQUATIONS

- FINITE VOLUME DISCRETIZATION
- 4 STAGE RUNGE-KUTTA SCHEME
- EIGENVALUE SCALING OF ARTIFICIAL DISSIPATION
- VARIABLE COEFFICIENT RESIDUAL SMOOTHING
- MULTIGRIDING

TURBINE HEAT TRANSFER

SOLUTION OF THE MODEL EQUATIONS

- FINITE VOLUME DISCRETIZATION
- 4 STAGE RUNGE-KUTTA SCHEME
- VARIABLE COEFFICIENT RESIDUAL SMOOTHING
- MULTIGRIDING
2-D CASES

- SSME HIGH-PRESSURE FUEL TURBINE 1ST STAGE VANE
- ALLISON'S C3X VANE DATA OF HYLTON ET AL.

3-D CASES

- LANGSTON'S CASCADE, THICK & THIN INLET BOUNDARY LAYERS.

Surface Pressure Distribution - First Stage Stator
TURBINE HEAT TRANSFER

Grid Spacing from the wall

First Stage Vane, High Reynolds Number
First Stage Vane, Low Reynolds Number

First Stator, Variation with length scale
$q-\omega$ Model
TURBINE HEAT TRANSFER

First Stator, Variation with length scale  
Chien's Model
Thick Inlet Boundary Condition

Experiment Graziani et al.

q-omega model, 193x49x65 grid

Surface Stanton Number x 1000., thick inlet boundary layer

Experiment, Graziani et al.

Computation, TRAF3D
Langston's Cascade
Endwall Stanton Number x1000.

Experiment, Graziani et al.  Computation, TRAF3D
Thick inlet boundary layer.

THICK BOUNDARY LAYER INLET CONDITION

q-omega Model, 193x49x65  Experiment Graziani
Surface Stanton Number $\times 1000$, thin inlet boundary layer

Experiment, Graziani et al.

Computation, TRAF3D

SURFACE STANTON NUMBER $\times 1000$, THIN INLET BOUNDARY LAYER

Experiment, Graziani et al.

$q-o$ model, $19.3 \times 49 \times 65$ grid
Langston's Cascade

Endwall Stanton Number x1000.

Experiment, Graziani et al.

Computation, TRAF3D

Thin inlet boundary layer.
CONCLUSIONS

- Two-equation models perform well in the fully turbulent regime.
- The uncertainty re. length scale of turb. needs to be resolved.
- Transition modeling is crucial to the success of H-T analysis.
- Need further improvements in convergence speed.
Thermal Turbulence Models
for
Turbine Blade Heat Transfer

John R. Schwab

Objective: Improved prediction of turbine blade surface
temperature and heat-transfer distributions

Rationale: Existing heat-flux models lack the fundamental
generality required for complex flows
<table>
<thead>
<tr>
<th>LeRC 9/93</th>
<th>Approach</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Fundamental analysis of thermal DNS results</td>
</tr>
<tr>
<td></td>
<td>Detailed evaluation of existing heat-flux models</td>
</tr>
<tr>
<td></td>
<td>Development of thermal time scale model</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LeRC 9/93</th>
<th>Kasagi, Tomita, and Kuroda DNS</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>fully-developed turbulent channel flow</td>
</tr>
<tr>
<td></td>
<td>constant-heat-flux walls with $Nu = 15.4$</td>
</tr>
<tr>
<td></td>
<td>$Re_\tau = 150$  $Re_{mean} = 4580$  $Pr = 0.71$</td>
</tr>
</tbody>
</table>
Evaluation of existing heat-flux models

Constant Pr_t models inadequate

Flux-transport models expensive

Two-equation thermal models offer affordable improvement
good overall agreement and proper asymptote for \( <uv>^+ \)

mediocre overall agreement, but proper asymptote for \( <v\theta>^+ \)
<table>
<thead>
<tr>
<th>LeRC</th>
<th>9/93</th>
<th>Two-equation thermal models</th>
</tr>
</thead>
</table>

- **Hattori, Nagano, and Tagawa (HNT)**  
  complex damping functions (evaluation in progress)

- **Sommer, So, and Lai (SSL)**  
  Shima coincidence condition (evaluation in progress)

- **Thermal time scale model** (development in progress)  
  simple wall boundary condition: \( \tau_\theta = 0 \)  
  simple near-wall asymptote: \( \tau_\theta \sim y^2 / 2 \alpha \)
Page intentionally left blank
A COUPLED IMPLICIT SOLUTION METHOD FOR TURBULENT SPRAY COMBUSTION IN PROPULSION SYSTEMS

K.-H. CHEN
The University of Toledo, Ohio
and
J.-S. SHUEN
Sverdrup Technology, Inc., Ohio

Aerothermochemistry Branch
Internal Fluid Mechanics Division
NASA Lewis Research Center, Cleveland, Ohio

OBJECTIVES

- Develop an efficient and robust algorithm for multi-phase chemically reacting flows at all speeds, with emphasis on low Mach number flows.
- Calculate turbulent spray combustion flow in a gas turbine combustor.
MOTIVATION

- Many reacting flows in propulsion devices cannot be efficiently calculated by modern compressible flow CFD algorithms, e.g.,
  - rocket motor — wide range of Mach numbers, from near zero velocity at closed end to supersonic at nozzle exit.
  - gas turbine combustor — low subsonic velocity, but large density variation precludes incompressible approach.

- Most low-speed reacting flow codes based on TEACH-type technologies – inefficient and lack of robustness for complex flows.

- Tremendous progress made in high-speed compressible flow CFD in past two decades. Extending application range to low-speed flow regime highly desirable.

OUTLINES

- GOVERNING EQUATIONS
  - Gas-Phase Equations
  - Liquid-Phase Equations

- NUMERICAL ALGORITHM

- NUMERICAL TEST RESULTS

- CONCLUSION

- FUTURE PLAN FOR ALLSPD CODE
GOVERNING EQUATIONS

* Gas-Phase Equations

\[
\Gamma \frac{\partial \hat{Q}}{\partial r^*} + \frac{\partial \hat{Q}}{\partial r} + \frac{\partial (\hat{E} - \hat{E}_v)}{\partial \xi} + \frac{\partial (\hat{F} - \hat{F}_v)}{\partial \eta} = \hat{H}_e + \hat{H}_i, \quad (1)
\]

\[
\hat{Q} = \frac{v^f}{J}, \quad \Gamma = \begin{bmatrix} p_0 & 0 & 0 & 0 & \cdots & 0 \\ u & \rho & 0 & 0 & 0 & \cdots & 0 \\ v & 0 & \rho & 0 & 0 & \cdots & 0 \\ h & \rho - 1 & \rho u & \rho v & \rho & \cdots & 0 \\ \kappa / \rho & 0 & 0 & \cdots & 0 & \cdots & 0 \\ \rho & 0 & \cdots & \cdots & 0 & \cdots & \cdots \\ \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots \\ \gamma_{N-1} & \gamma_{N-1} / \beta & 0 & 0 & \cdots & 0 & \cdots & \rho \end{bmatrix}
\]

\[
\hat{H}_e = \begin{bmatrix} \frac{1}{\beta} \\ \frac{1}{\gamma} \\ -\frac{3}{2} \frac{\partial \mu (\xi v)}{\partial \xi} \\ \frac{\delta (p - \tau + \frac{3}{2} \beta (\mu_v))}{\delta \xi} \\ \frac{\delta (\mu_v)}{\delta \xi} \\ \frac{2}{3} \frac{\delta (\mu_v)}{\delta \xi} + \frac{\delta (\mu_{v^2})}{\delta \xi} \\ \frac{1}{2} \frac{\delta (\mu_v)}{\delta \xi} + \frac{\delta (\mu_{v^2})}{\delta \xi} \end{bmatrix}
\]

\[
\hat{H}_i = \begin{bmatrix} \sum n_p \hat{m}_p \\ \sum n_p \hat{m}_p \hat{m}_p - \frac{4}{3} \rho \sum n_p \frac{du}{dt} \\ \sum n_p \hat{m}_p \hat{v}_p - \frac{4}{3} \rho \sum n_p \frac{du}{dt} \\ \sum n_p \hat{m}_p \hat{v}_p - \frac{4}{3} \rho \sum n_p \frac{du}{dt} \\ \sum n_p \hat{m}_p \hat{v}_p - \frac{4}{3} \rho \sum n_p \frac{du}{dt} \\ \sum n_p \hat{m}_p \hat{v}_p - 4 \pi r^2 \sum n_p \Delta T \end{bmatrix}
\]
ALLSPD MAIN FEATURES

Present Capabilities

- 2-D and Axisymmetric geometries
- Second-order central difference for both inviscid and viscous terms
- Fully coupled, fully implicit algorithm
- Efficient convergence for wide range of Mach numbers (from $M \leq 10^{-10}$ to supersonic)
- Finite-rate chemistry, realistic thermophysical properties
- Multi-block and body-fitted curvilinear coordinates for complex geometries
- $\kappa - \epsilon$ turbulence model
- Stochastic liquid spray model (dilute spray), vortex model for droplet internal circulation and diffusion
- Still a research code, require experience and knowledge of CFD and flow physics to use.

Future Plans

- Development of a more efficient solver
- Extension to 3-D
- PDF model for turbulence/chemical reaction closure
- Thermal radiation model
- Detailed soot and NOx kinetic models
- Multi-grid and unstructured grid capabilities
- Dense spray and high pressure (near-/super-critical) spray models

Difficulties with Compressible Flow Algorithms at Low Mach Numbers

- Disparities among system's eigenvalues (stiffness), $u, u + c, u - c$, resulting in significant slowdown in convergence rate.

- Singular behavior of pressure gradient term in momentum equations as Mach number approaches zero,

$$\rho^* u^2 + \frac{p^*}{\gamma M_r^2}$$

As Mach number is decreased, pressure variation ($\Delta p^* \propto M^2$) becomes of similar magnitude as roundoff error of the large pressure gradient term ($p^*/\gamma M_r^2$).
METHOD OF APPROACH

Pressure Singularity Problem

- Pressure decomposed into two parts:

\[ p = p_o + p_g \]

\( p_g \) replaces \( p \) in momentum equations and retains \( p_g \) as one of the unknowns.

- Employs conservative form of governing equations, but uses primitive variables

\[ (p_g, u, v, h, Y_1) \]

as unknowns. Conservation property preserved and pressure field accurately resolved for all Mach numbers.

Eigenvalue Stiffness Problem

- Pressure rescaled so that all eigenvalues have the same order of magnitude. Physical acoustic waves removed and replaced with pseudo-acoustic waves which travel at speed comparable to fluid convective velocity.

EIGENVALUES RESCALING

\[ \lambda = U, U, U, \frac{1}{2} \left[ U(1 + \frac{\beta}{c^2}) \pm \sqrt{U^2(1 - \frac{\beta}{c^2})^2 + 4\beta(\alpha_1 + \alpha_2^2)} \right], \ U, U \ldots \]

\[ \alpha_1 = \xi_x, \quad \alpha_2 = \xi_y, \quad U = \alpha_1 u + \alpha_2 v. \]

For well-conditioned eigenvalues, scaling factor \( \beta \) taken to be

\[ \beta = u^2 + v^2. \]
LIQUID-PHASE EQUATIONS

• Droplet Motion Equations

\[
\frac{dx_p}{dt} = u_p, \quad \frac{dy_p}{dt} = v_p, \quad \frac{du_p}{dt} = \frac{3}{16} \frac{C_D \mu_p}{\rho_p r_p^2} (u_p - u), \quad \frac{dv_p}{dt} = \frac{3}{16} \frac{C_D \mu_p}{\rho_p r_p^2} (v_p - v),
\]

\[
Re_p = \frac{r_p u_p}{\mu_p} \left[ \left( u_p - u \right)^2 + \left( v_p - v \right)^2 \right]^{1/2},
\]

\[
C_D = \begin{cases} 
0.44 & \text{for } Re_p < 1000, \\
& \text{for } Re_p > 1000.
\end{cases}
\]

• Droplet Heat and Mass Transfer Equations

\[
\frac{\dot{m}_p d_p}{\rho D_f} = 2 N_e \ln(1 + B),
\]

\[
\frac{hd_p}{k} = \frac{2 N_e (1 + B) Le^{-1}}{((1 + B) Le^{-1} - 1)}
\]

\[
N_e = 1 + \frac{0.276 Re_p^{1/3} Pr^{1/3}}{[1 + \frac{1.233}{Re_p Pr^{1/3}}]^{1/2}} \quad N_p = 1 + \frac{0.276 Re_p^{1/3} Sc^{1/3}}{[1 + \frac{1.233}{Re_p Sc^{1/3}}]^{1/2}},
\]

\[
B = \frac{Y_{fpp} - Y_f}{1 - Y_{fpp}} \quad Y_{fpp} = \frac{X_{fpp} W_f}{X_{fpp} W_f + (1 - X_{fpp}) W_a}
\]

152
• Droplet Internal Temperature Equations
  (Vortex Model)

\[
\frac{\partial T_p}{\partial t} = 17 \frac{k_l}{C_p \rho_l r_p^2} \left[ \alpha \frac{\partial^2 T_p}{\partial \alpha^2} + (1 + C(t)\alpha) \frac{\partial T_p}{\partial \alpha} \right]
\]

\[
C(t) = \frac{3}{17} \left( \frac{C_p \rho_l}{k_l} \right) r_p \frac{dr_p}{dt}
\]

\[
t = t_{inj}, \quad T_p = T_{inj},
\]
\[
\alpha = 0, \quad \frac{\partial T_p}{\partial \alpha} = \frac{1}{17} \left( \frac{C_p \rho_l}{k_l} \right) r_p \frac{\partial T_p}{\partial t},
\]
\[
\alpha = 1, \quad \frac{\partial T_p}{\partial \alpha} = \frac{3}{16} \left( \frac{C_p \rho_l}{k_l} \right) r_p \frac{\partial T_p}{\partial r}.
\]

**NUMERICAL ALGORITHM**

- Gas-Phase - ALLSPD code
- Liquid-Phase
  - Droplet motion equations (ODE) - Runge-Kutta method.
  - Droplet internal equations (PDE) - implicit method (Thomas algorithm).
  - Determination of spray time step for integration.
  - Stochastic separate flow model.
- Interaction Between Two Phases
SPRAY TIME STEP

• Droplet Velocity Relaxation time ($t_r$)

$$t_r = \frac{16}{3} \left( \frac{\rho_t}{\rho_p} \right) \left( \frac{r_p^2}{\nu} \right) (C_D \text{Re}_p)^{-1}.$$  

• Droplet Life Time ($t_l$)

$$t_l = \frac{r_p}{3 \dot{m}_{\text{p}} \rho_p}.$$  

• Droplet Surface Temperature Constraint Time ($t_s$)

$$t_s = \ln \left( \frac{1}{1 - \frac{\Delta T_p}{\Delta T_s}} \right) / \Lambda'$$

$$\frac{dT_p}{dt} = \frac{6}{\rho_l C_{\text{d}_p}} [h(T - T_p) - \dot{m}_{\text{p}} h_{\text{fg}}]$$

• Local Grid Time Scale ($t_g$)

• Turbulent Eddy-Droplet Interaction Time ($t_i$)

$$t_i = t_e, \text{ if } L_e > \tau |\ddot{u} - \bar{u}_p|$$

$$t_i = \min(t_e, t_l), \text{ if } L_e < \tau |\ddot{u} - \bar{u}_p|$$

where

$$L_e = C_{\mu}^{3/4} \kappa^{3/2} / \epsilon$$

$$t_e = L_e / (2\kappa / 3)^{1/2}.$$  

$$t_l = -\tau \ln [1 - L_e / (\tau |\ddot{u} - \bar{u}_p|)]$$

• Spray Time step - $\Delta T_{spr}$

$$\Delta t_{spr} = \alpha \min (t_r, t_l, t_s, t_g, t_i)$$
INTERACTION BETWEEN TWO PHASES

1. Initialize gas and liquid phase variables.
2. Solve liquid-phase equations.
3. Evaluate spray source term, $H_l$.
5. Update spray source term, $H_l$?
   - No, go to step 4.
   - Yes, go to step 2.

NUMERICAL TEST RESULTS

- Turbulent Backward-Facing Step Flow - non-reacting.
- Evaporating Turbulent Spray Flow.
- Gas Turbine Spray Combustion Flow.
OPERATING MODE 1
NO ATOMIZING AIR

LIQUID FLOW RATE: 1.26 grams/sec
AIR FLOW RATE: 0.00 grams/sec
CONCLUSION

- ALLSPD code efficiently coupled with the SSF spray model.
- Satisfactory convergence property for flows with and without spray.
Part A. On Turbulent Transport of chemical Species in Compressible Reacting Flows

Contents

Mixing of chemical species in reacting flows

Analysis

Chemically reacting, turbulent flow equations
  Density-weighted, time-averaged Navier-Stokes equations
  Multiple-time-scale turbulence equations
  Species conservation equations for reacting flows

Numerical Method

Mixing and combustion of hydrogen in vitiated supersonic airstream.
  Comparison with measured data and other numerical results obtained using k-ε turbulence models and a PDF.

Conclusions and Discussion
Mixing of chemical species in reacting flows

Chemically Reacting Laminar Flows

Numerical (or semi-analytical) methods

O.D.E. solvers
1-D Numerical methods
Numerical uncertainty is minimal

Chemical kinetics

Chemical kinetics (finite rate chemical kinetics & Reduced finite rate chemical kinetics) for certain fuels (hydrogen and some of hydro-carbons) have been tested and validated repeatedly.

Numerical and theoretical analyses yield accurate results.

Chemically Reacting Turbulent Flows

Numerical methods

Boundary layer or Navier-Stokes equations solvers
(Except for semi-analytical analyses of PSR cases)
Uncertainty is caused partly by numerical methods.

Chemical kinetics

Finite rate chemical kinetics & Reduced finite rate chemical are used.
Uncertainty is caused partly by over-simplified chemical kinetics.

Turbulence Equations

Numerical analysis yields not so accurate results. Uncertainty caused by turbulence equations is by far greater than that caused by chemical kinetics.*

The probability density functions are used to improve the predictive capability of turbulent mixing of chemical species. However, some pdf methods yield physically incorrect numerical results.
It is certainly important to better understand the turbulent mixing of chemical species in reacting flows.

Turbulent Mixing of Chemical Species in Reacting Flows

1. Large eddy mixing
2. Turbulent Mixing
3. Molecular Mixing

The use of accurate chemical kinetics alone cannot yield accurate results for numerical analysis of turbulent reacting flows unless turbulent mixing is resolved correctly.

Large Eddy Mixing
Caused by separated flows, recirculating flows, and organized structures.
The extent of calculated recirculation zone or the organized structures depends on the accuracy of both the numerical method and the turbulence equations since they are nonlinearly coupled with each other.

\[
\frac{D}{Dt}(p\mu_j) + \frac{\partial}{\partial x_j}(p\mu_j\mu) - \frac{\partial}{\partial x_j}\left[\left(\mu + \mu_t\right)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) + \frac{\partial p}{\partial x_i}\right] = \frac{\partial}{\partial x_j}\left[\frac{1}{\sigma_t}\frac{\partial k}{\partial x_j}\right] - P_r \cdot \varepsilon_t
\]

where \(\mu_t = \mu_t(k, \varepsilon, \cdots)\), and \(P_r = \frac{\mu_t}{\rho} \left(2\left(\frac{\partial u_i}{\partial x_j}\right)^2 + 2\left(\frac{\partial v_i}{\partial y}\right)^2 + \left(\frac{\partial u_i}{\partial x}\right)^2 + \left(\frac{\partial v_i}{\partial y}\right)^2\right)\)

Therefore, the capability to correctly resolve the large eddy mixing depends on the accuracy of the numerical method and the turbulence equations.

Numerical investigations carried out during last decades show that k-\(\varepsilon\), ARSM, and RSM do not yield accurate results for complex turbulent flows.
On the other hand, the multiple-time-scale turbulence equations* yield highly improved numerical results for various complex turbulent flows.

Turbulent Mixing

Caused mostly by energy-containing eddies and partly by fine-scale eddies.

The capability to resolve the turbulent mixing of chemical species depends on the capability of the turbulence equations to correctly describe the turbulent transport of scalar variables (i.e., heat transfer, turbulent kinetic energy, concentrations, and convection-diffusion of species, etc).

Need to be able to describe the chemical reaction-turbulence interaction (i.e., turbulent mixing is enhanced and shear layer thickness is widened by chemical reaction)*

The multiple-time-scale turbulence equations can resolve the cascade of turbulent kinetic and the nonequilibrium turbulence phenomena. (Present numerical results show that the M-S turbulence equations can resolve the chemical reaction-turbulence interaction.)


Molecular Mixing

Caused by molecular diffusivity.

Theoretical and numerical analyses of chemically reacting laminar flows yield accurate results.

Various molecular diffusion equations can accurately describe the molecular diffusion of species.

The Lennard-Jones 12-6 potential law is used in the present study.

Various numerical methods that yield accurate results for laminar flows can resolve the molecular mixing.
Multiple-time-scale turbulence equations

Energy containing eddies

\[
\frac{\partial}{\partial t} (p\kappa_p) + \frac{\partial}{\partial x_j} (p\nu_j\kappa_p) - \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_{kp}} \right) \frac{\partial \kappa_p}{\partial x_j} = \rho \mathcal{P}_t - \rho \varepsilon_p
\]

\[
\frac{\partial}{\partial t} (p\varepsilon_p) + \frac{\partial}{\partial x_j} (p\nu_j\varepsilon_p) - \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_{ep}} \right) \frac{\partial \varepsilon_p}{\partial x_j} = \frac{\rho}{k_p} \left( c_{p1} \mathcal{P}_p^2 + c_{p2} \mathcal{P}_t \varepsilon_p - c_{p3} \varepsilon_p^2 \right)
\]

Fine Scale Eddies

\[
\frac{\partial}{\partial t} (p\kappa_t) + \frac{\partial}{\partial x_j} (p\nu_j\kappa_t) - \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_{kt}} \right) \frac{\partial \kappa_t}{\partial x_j} = \rho \varepsilon_p - \rho \varepsilon_t
\]

\[
\frac{\partial}{\partial t} (p\varepsilon_t) + \frac{\partial}{\partial x_j} (p\nu_j\varepsilon_t) - \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_{et}} \right) \frac{\partial \varepsilon_t}{\partial x_j} = \frac{\rho}{k_t} \left( c_{t1} \varepsilon_p^2 + c_{t2} \varepsilon_p \varepsilon_t - c_{t3} \varepsilon_t^2 \right)
\]

Turbulent Eddy Viscosity

\[
\mu_t = \rho c_{\mu} k^2 / \varepsilon_p \quad \text{where} \quad c_{\mu} = c_{\mu f} \varepsilon / \varepsilon_p
\]

Remark: Single-time-scale turbulence models can not resolve nonequilibrium turbulence phenomena.

Species conservation equations for reacting flows

Chemical species concentration equation

\[
\frac{\partial}{\partial t} (pY_i) + \nabla \cdot \left( pY_i (v + V_i) \right) = \dot{w}_i
\]

where the diffusion velocity, \( V_i \), is approximated using the Fick's law given as

\[
Y_i V_i = \left( D_{i,i} + D_{i,i} \right) \nabla Y_i
\]

The production rate of the \( i \)-th species, \( \dot{w}_i \), is given as

\[
\dot{w}_i = \sum_{k=1}^{N_r} M_i \left( n_{i,k}^{*} - n_{i,k} \right) \omega_k
\]

where

\[
\omega_k = \prod_{j=1}^{N_s} c_{j,k}^{y_{j,k}} - \prod_{j=1}^{N_s} c_{j,k}^{y_{j,k}}
\]
Chemical reactions for the combustion of H2 in a vitiated supersonic airstream are described using 9 chemical species (H2, O2, H2O, OH, O, H, HO2, H2O2, and N2) and 24 pairs of reaction-steps (Burks and Oran, 1981; Kumar, 1989).

A fast chemistry can not be used to describe the fine details of chemically reacting flows.

A reduced chemical kinetics can not be used confidently due to the uncertainty contained in the reaction mechanisms.

The use of a detailed finite rate chemistry may make it difficult to obtain a fully converged solution due to the coupling between the large number of flow, turbulence, and chemical equations. The numerical method needs to be strongly convergent. Accuracy also depends on the capability of turbulence equations used.

**Numerical Method**

The numerical method is a finite volume method that incorporates a pressure-staggered mesh and an incremental pressure equation for the conservation of mass.

**Predictor Step:** Solve momentum equation.

\[(pC_1+A_i^*)u_i^{**} = \sum_{nb} A_k^* u_k^{**} + S_i^* - \frac{\partial p^*}{\partial x_i} + pC_2u_i^{p-1} - pC_3u_i^{p-2} \quad (1)\]

**Corrector Step:** Correct the velocity field to be divergence free.

Incremental pressure equation

\[\frac{\partial }{\partial x_j} \left[ \frac{p'}{RT} u_j^{**} \right] - \frac{\partial }{\partial x_j} \left[ \frac{1}{\rho C_1 + A_i^*} \right] \frac{\partial p'}{\partial x_j} = - \frac{\partial u_j^{**}}{\partial x_j} \quad (2)\]

Incremental velocity equation

\[u_i' = \frac{1}{(pC_1 + A_i^*)} \frac{\partial p'}{\partial x_i} \quad (3)\]
Velocity and pressure corrections

\[ u_i^{***} = u_i^{**} + u_i' \]  \hspace{1cm} (4)  
\[ p^{**} = p' + p' \]  \hspace{1cm} (5)  

Solve eqs. (1-5) iteratively until all flow variables are converged.

Combustion of \( \text{H}_2 \) in vitiated supersonic airstream (Burrows and Kurkov, 1973)
The numerical result obtained using the M-S turbulence equations indicates that the combustion occurs at approximately middle of the channel. The calculated flame location is in excellent agreement with that observed in the experiment.
Temperature contours obtained using the M-S turbulence equations

It can be seen in the temperature contours that the calculated flame location obtained using the M-S turbulence equations are in correct agreement with the measured data. The contours also indicate that the temperature increases within a short distance. The trend is in correct agreement with experimental observations that temperature increase occurs within a finite flame thickness.

Temperature contours obtained using the pdf*

The pdf fails to predict the correct flame location (i.e., ignition delay). The slowly increasing temperature field indicates that the pdf may not be able to predict a correct flame front. The numerical results obtained using the pdf are not in correct agreement with the physics of combustion.

Conclusions and Discussion

The calculated species concentration profiles are in as good agreement with the measured data as those obtained using the pdf.

The flame location (ignition delay) obtained using the multiple-time-scale turbulence equations is in excellent agreement with the experimentally observed onset of ultraviolet radiation.

Cascade of the turbulence filed is influenced by the extra strains caused by chemical reaction.

Both the numerical results and the measured data exhibit enhanced mixing of the hydrogen and vitiated airstream for the reacting case.

The M-S turbulence equations can resolve the chemical reaction-turbulence interaction.

The pdf produces slowly increasing temperature field and it fails to predict the ignition delay. Thus the numerical results obtained using the pdf are not in correct agreement with physics of combustion.

Part B. Unsteady Transitional Flows over Forced Oscillatory Surfaces*

Contents

Nomenclature

Unsteady turbulent flow equations for flows with moving boundaries

- Navier-Stokes equations defined on Lagrangian-Eulerian coordinates
- Multiple-time-scale turbulence equations

Numerical results

- Unsteady transitional flow field and comparison with measured data.

Nomenclature

Lagrangian-Eulerian coordinates

\( x \): fixed reference coordinates
\( \xi \): moving coordinates
\( J = \left| \frac{\partial \xi_j}{\partial x_i} \right| \)

Unsteady transitional flow equations with moving boundaries

Conservation of mass equation

\[
\frac{\partial}{\partial t}(\rho J) = J \frac{\partial}{\partial x_j} \left[ \rho \left( u_j^g - u_j \right) \right] + J \frac{\partial p}{\partial x_j} \frac{\partial}{\partial x_i}
\]

Conservation of linear momentum equation

\[
\frac{\partial}{\partial t} \left( \rho u_j J \right) = J \frac{\partial}{\partial x_j} \left[ \rho u_j \left( u_j^g - u_j \right) \right] + J \frac{\partial \tau_{ij}}{\partial x_j} - J \frac{\partial p}{\partial x_i}
\]

Convection-diffusion equation for scalar variables (i.e., \( \phi = \{ \kappa_p, \varepsilon_p, k_t, \varepsilon_t, \text{etc.} \} \))

\[
\frac{\partial}{\partial t} (\rho \phi J) = J \frac{\partial}{\partial x_j} \left[ \rho \phi \left( u_j^g - u_j \right) \right] + J \left( \mu_s \frac{\partial \phi}{\partial x_j} \right) - J \rho f(\phi)
\]
Numerical Method

The unsteady transitional flow equations are solved using the same finite volume method. Time-integration is made using an iterative-time-advancing scheme.

Comparison of Unsteady Flow Solution Techniques*

Iterative Time-Advancing Scheme (ITA).
Simplified Marker and Cell (SMAC).
Pressure-Implicit Splitting of Operators (PISO).

The ITA that can best resolve the nonlinearity of the Navier-Stokes equations, and yields the most accurate results.

The SMAC is the most efficient computationally and yields accurate numerical results for laminar flows.

The PISO is the most unstable numerically and yields less accurate results.


Oscillating airfoil and moving mesh
The deteriorated comparison at $\alpha = 19^\circ$d

(i) The hot wire can not accurately measure the velocity components when the flow is misaligned more than approximately $30^\circ$ from the hot wire axis.

(ii) The interaction between the DSV and the TEV occurs in a relatively coarse mesh region and the numerical method yields somewhat deteriorated results.
CONCLUSIONS AND DISCUSSION

Numerical method successfully predicts the Dynamic Stall Vortex and the Trailing Edge Vortex.

The calculated ensemble-averaged velocity profiles are in good agreement with the measured data.

Both the numerical results and the measured data show that the transition from laminar to turbulent state and relaminarization occur widely in space and in time.

The good comparison between the numerical results and the experimental data are attributed to the capability of

(i) the ITA that can best resolve the nonlinearity of the Navier-Stokes equations,

(ii) the new pressure correction algorithm that can strongly enforce the conservation of mass, and

(iii) the Multiple-time-scale turbulence equations that can resolve the transitional nonequilibrium turbulence field.
Page intentionally left blank
Direct Calculations of Waves in Fluid Flows Using A High-Order Compact Difference Scheme

Sheng-Tao Yu
Sverdrup Technology, Inc.
NASA Lewis Research Center Group
Brook Park, OH 44142
APPROACH

- Multiple step Runge-Kutta methods for time marching.
- High-order compact difference schemes for spatial discretization.
- MOC type Nonreflective boundary conditions.
- Fourier analysis for numerical dispersion relations.
- Benchmark testing:
  1. Acoustics admittance of nozzle flow.
  2. Shocked sound wave (N-wave).
  3. A Lamb vortex in an uniform flow.
  4. Vortex pairing

OBJECTIVES

- To simulate the unsteady flows accurately for:
  - Direct numerical simulations of turbulent flows.
  - Computational aeroacoustics.
  - Flow and/or combustion instability problems.
- Understand numerical dispersion relations of the finite difference schemes.
FOURIER ANALYSIS

- Time marching methods.
  - 3rd-order Runge-Kutta methods
    \[ g = 1 + Z + \frac{1}{2} Z^2 + \frac{1}{6} Z^3. \]
  - 4th-order Runge-Kutta methods
    \[ g = 1 + Z + \frac{1}{2} Z^2 + \frac{1}{6} Z^3 + \frac{1}{24} Z^4. \]

- Spatial discretization.
  - 4th-order compact difference scheme
    \[ Z^{(4)} = -\frac{6 F \sin(\hat{k}) i}{4 + 2 \cos(\hat{k})}, \]
  - 6th-order compact difference scheme
    \[ Z^{(6)} = -\frac{F[4 \sin(\hat{k}) \cos(\hat{k}) + 56 \sin(\hat{k})] i}{12[2 \cos(\hat{k}) + 3]} . \]

THE COMPACT DIFFERENCE SCHEMES

- Interior Nodes
  - 4th-order central difference scheme
    \[ u'_{i-1} + 4u'_i + u'_{i+1} = \frac{3}{\Delta x}(u_{i+1} - u_{i-1}) \]
  - 6th-order central difference scheme
    \[ u'_{i-1} + 3u'_i + u'_{i+1} = \frac{1}{12\Delta x}(u_{i+2} + 28u_{i+1} - 28u_{i-1} - u_{i-2}) \]

- 3rd-order biased difference on boundary Nodes
  \[ 2u'_1 + 4u'_2 = \frac{1}{\Delta x}(-5u_1 + 4u_2 + u_3) \]

- Implicit methods.
- Require inversion of a scalar tridiagonal matrix.
Fig. 1 Dissipation characteristics of the CD6-RK4 scheme for CFL=0.6.

1a $|q_1(p,q)|$ for $u = v$ and $c = \sqrt{u^2 + v^2}$.
1b $|q_2(p,q)|$ for $u = v$ and $c = \sqrt{u^2 + v^2}$.
1c $|q_3(p,q)|$ for $u = v$ and $c = \sqrt{u^2 + v^2}$.

Fig. 2 Dispersion characteristics of the CD6-RK4 scheme for CFL=0.6.
SHOCKED SOUND WAVES

- Initial condition:
  - Sinusoidal pressure profile.
  - Isentropic conditions to determine $\rho$ and $T$ profiles.
  - Simple wave relation,
    $$u(x) = \int_{\rho}^{\rho(x)} \frac{dp}{\rho(x)c(x)}$$
  - All flow properties are in phase.
- Straight tube with periodic boundary condition.

FOURIER ANALYSIS

- Use high-order spatial differencing to improve dispersive characteristics. ($\text{CD6} = \text{CD4} \geq \text{CD2}$)
- Use high-order time stepping to enlarge CFL limit.
- No dissipation but high dispersion at the highest wave number mode resolved by a given numerical grid.
- Artificial damping is required.
SHOCKED SOUND WAVE

- All flow properties are in phase.
- U+C
- U-C
- U
A LAMB VORTEX IN AN UNIFORM FLOW

- Velocity distribution.

\[ u_\theta = \frac{\Gamma}{2\pi} \frac{r}{r^2 + a^2}, \]  
(54)

- Rigid-body rotation near vortex center.
\[ u_\theta \approx r \]
- Irrotational far outside vortex core.
\[ u_\theta \approx \frac{1}{r} \]

- Momentum equation.

\[ \frac{\partial p}{\partial r} = \rho \frac{u_\theta^2}{r} \]

- Energy equation.

\[ \frac{\gamma}{\gamma - 1} \frac{p}{\rho} + \frac{u_\theta^2}{2} = h_0 \]

- Vortex in an uniform flow.

\[ u = \bar{u} + u', \]
\[ v = \bar{v} + v', \]
Fig. 11. The vorticity magnitude contours for the vortex pairing.
Fig. 11. The vortex constraint contours for the vortex pairing

Fig. 8. The spatial growth rate as a function of the angular frequency from linear
theory for the compressible flow shear layer.
Fig. 11 The Fourier coefficients of the instability waves in a compressible free shear layer perturbed by the most unstable mode at the magnitude of one thousandth.

Fig. 9 Instantaneous distribution of $|u'|$ along the center line of the free shear layer.
Fig. 13 Fourier coefficients of the unsteady motion in a compressible free shear layer perturbed by the most unstable mode at a magnitude of one hundredth.

Fig. 12 Reconstructed eigenfunction components $u'$ and $p'$ from the CFD results for the free shear layer perturbed by the most unstable mode.
Fig. 14 Contours of constant vorticity of the simulated free shear layer perturbed by the most unstable mode at a magnitude of one hundredth.
Fig. 10 Contours of constant vorticity of the simulated free shear layer perturbed by the most unstable mode and its subharmonic.

CONCLUDING REMARKS

- Successful development of quasi-1-D and 2-D Euler solvers using Runge-Kutta and compact difference methods to simulate unsteady flows.
- Fourier analysis to guide the numerical calculations: numerical scheme, CFL number, artificial damping.
- For flows of complex harmonic content, 4th-order central differencing is better than the 2nd central difference.

193
INTRODUCTION

CMOTT at NASA Lewis:
to address propulsion related turbulence issues
☐ assessment of state-of-the-art models
☐ development of new models
☐ validation of new techniques

Objective of this work:
to assess the performance of several low Reynolds number $k-\varepsilon$ formulations compared to the standard high Reynolds number form for separated flow over a step
Need to clarify this issue for code developers

Critical comparisons between \( k-\epsilon \) models for:

- Channel flow: Lang & Shih (1991)
- Flow past a hill: Michelassi & Shih (1991)

Avva (1990) compared the high Re\(_t\) and Chien \( k-\epsilon \) models for several flows: noted the deficiency of Chien for step flow skin friction results . . .

Contradicts idea of "more work \( \leftrightarrow \) better result"

Is this a property of all Low Reynolds Number \( k-\epsilon \) models?

\[ k-\epsilon \text{TURBULENCE MODEL} \]

Specifically, \( \nu_t = C_{\mu} f_{\mu} \left( \frac{k^2}{\epsilon} \right) \)

Model transport of \( k \) and \( \epsilon \):

\[
\begin{align*}
k_{t,i} + U_j k_{j,i} - \left( \left( \nu + \frac{\nu_t}{\sigma_k} \right) k_j \right)_{j,i} &= \mathcal{P} - \epsilon + D \\
\epsilon_{t,i} + U_j \epsilon_{j,i} - \left( \left( \nu + \frac{\nu_t}{\sigma_{\epsilon}} \right) \epsilon_j \right)_{j,i} &= C_1 f_1 \frac{\epsilon}{k} \mathcal{P} - C_2 f_2 \frac{\epsilon^2}{k} + \mathcal{E}
\end{align*}
\]

where the production of \( k \) is: \( \mathcal{P} = -\langle u_i u_j \rangle \frac{1}{2} (U_{i,j} + U_{j,i}) \).

Primary difference between formulations:
definition of \( f_{\mu}, f_1, f_2, D, E \)
Low Reₜ forms:

- numerical domain extends to wall
- near wall profiles of \( \bar{U}, k, \) and \( \epsilon \) are resolved not assumed
- typically first cell located \( y^+ \approx 1.0 \)
- damping functions (\( f_1, f_2, \) and \( f_\mu \)) and additional terms (D and E) tuned to specific flows

Generality suffers somewhat...

Formulation of above terms can affect the generality as well:

- JL and LS are functions of dependent variables alone
- CH and SL are functions of both dependent and independent variables

BACK-FACING STEP (BFS) FLOW

Experiment of Driver and Seegmiller (AIAA J., '85)
Configuration:
- Re_{step} of 33420
- inlet Mach Number of 0.128
- inlet tunnel 80H
- exit tunnel 60H

Chosen for:
- LDV data for velocity, turbulent stresses
- wall static C_p and wall C_f
- reattachment length (time averaged)

NUMERICAL EXPERIMENT

Code: modified version of DTNS2D (Gorski '88)

Method:
- pseudo-compressibility (Chorin '67)
- upwind differenced, TVD scheme for convective terms
  (Chakravarthy & Osher '85)
- approximate factorization for time integration
- multiblock configuration
- decoupled (lagged) treatment of turbulence equations
- implicit treatment of turbulent source terms
Velocity profiles:

Wall static pressure coefficient:
Turbulent kinetic energy profiles:

![Turbulent kinetic energy profiles diagram]

Turbulent shear stress profiles:

![Turbulent shear stress profiles diagram]
Skin friction coefficient:

![Graph showing skin friction coefficient with various data points and lines representing different studies.](image)

Eddy viscosity field, $\nu_T$:

![Images of eddy viscosity fields for Shih-Lumley and Chien with respective color scales.](image)
Primary reattachment length, $x_r$:

<table>
<thead>
<tr>
<th>model</th>
<th>JL</th>
<th>LS</th>
<th>CH</th>
<th>SL</th>
<th>HR</th>
</tr>
</thead>
<tbody>
<tr>
<td>$x_r$</td>
<td>4.9</td>
<td>5.4</td>
<td>5.4</td>
<td>5.1</td>
<td>5.5</td>
</tr>
</tbody>
</table>

Driver & Seegmiller recorded: 6.26H

Secondary reattachment length, $x_{r2}$:

<table>
<thead>
<tr>
<th>model</th>
<th>JL</th>
<th>LS</th>
<th>CH</th>
<th>SL</th>
</tr>
</thead>
<tbody>
<tr>
<td>$x_{r2}$</td>
<td>0.0</td>
<td>0.0</td>
<td>0.8</td>
<td>0.1</td>
</tr>
</tbody>
</table>

Driver & Seegmiller recorded: $0.5 \leq x_{r2} \leq 1.8$

---

**CLOSING COMMENTS**

- JL, LS, CH, SL and HR $k$-$\varepsilon$ models tested for back-step flow
- In general, results very similar between 5 different forms
- All do reasonably well for $\bar{U}$ and $C_p$, albeit LR slightly better
- Turbulence profiles possess misplaced peak values as well as overpredictions in recirculation zone, common to all
- Reattachment length and $C_f$ are difficult to properly resolve (14% error) especially for LR forms
- Definition of secondary recirculation zone inconclusive on this grid
- Perhaps a sensitivity to pressure gradient would help the LR damping functions . . .
Analysis of Supersonic Flows using $k-\varepsilon$ Model and the RPLUS code; Progress towards High Speed Combustor Analysis.

by J. Lee
Sverdrup Technology Inc./CFD Branch
for Workshop on Computational Turbulence Modeling
Sept. 1993
Problem of Interest

- Analysis of Chemically Reacting flow inside of Supersonic RAM jet Combustors-Two Key Parameters need to be determined.
  - Mixing/Combustion Efficiency
  - Kinetic Energy Efficiency (Flow Losses)
  - Inlet, Diffuser, etc..
- In order to do get some ideas on those parameter following (Potential Loss Mechanisms) must be modeled/determined correctly.
  - Mixing, Shear,
  - Turbulence, Vorticity,
  - Shock-waves, Heat Transfer,
  - Fuel Injector Drag, Poor Wall Pressure Integral,
  - Chemical Dissociation.

from 2nd JANNAF workshop on SCRAMjet Combustor performance workshop
Mixing and Injector Design

- At High Mach Number (M ~ 5.0 +).
  - Doesn’t mix well!
  - The Natural diffusion mechanism very INEFFECTIVE.
  - Fuel Residence Time Extremely Small - Even with Fast Fuel Such as H₂
- Geometrical Complexities
  - To induce Favorable mixing and Flame holding features
  - Back-Step/Stream Wise Vorticity/Shock-Wave Interactions
  - Unsteady Mechanism also being Envisioned as mixing enhancement
    - Kumar, Bushnell and Hussani (1987)

Introduction of Externally Generated Mixing Enhancements

- Some External helping hand needed => Modeling Difficulties.
- Externally Generated Vorticity Through Sweep angle of the Ramp injector.
- Multiple Transverse Injection.
  - Hartfield et. al. (1991)
- Flame holding tricks/ Back-step with Recirculation.
  - Hartfield et. al. (1991)
- Simplified analysis of these features very difficult because of limited database/understanding (Attempts are being made using CFD solutions - JANNAF Combustor Subcommittee).
Numerical Modeling (CFD) of Combustor Flow Field

- CFD Analysis.
- Numerical Modeling => Overall Analysis of performance => Difficult
- Overall Laminar Flow Fields with Complex Geometry/Finite Rate Chemistry has been demonstrated.
- Finite Rate Chemistry Model: Yoon and Shuen (1989)
- Multiple Grid Blocks: Moon (1991)
- Analysis of a typical Injector Configuration with Zero Equation Turbulence Model using LU Scheme (RPLUS) code: Lee (1993)

Simple Zero Equation Turbulence model with multiple wall scaling Buleev-Inverse square rule can be used to extend model into three-dimensional form. (Lee (1993))

- Good News/Bad News
- Typical velocity profiles can be reasonably predicted.
- Over all combustor flow features can be reasonably predicted.
- Near-wall temperature characteristics near non-equilibrium region around the injector and separated flow were poorly predicted.
- Overall spreading behavior of shear region poorly predicted.

- Two Equation Transport Turbulence Model has the potential to ease some of these difficulties.
THREE-BLOCK GRID SYSTEM

HARTFIELD ET. AL. (1990)

TEST SECTION GEOMETRY

FLOW SCHEMATIC

Mach 2.0
Pl = 565Kpa
T1 = 170K

18.1mm(H)
3.3mm(D)
Mach 1.7

As = 9.5
Ar = 9.5

9.5 Degree Ramp
Shock Injector Boundary

Under-Expanded Jet

Ramp-Induced Shock
Jet-Induced Shock
Two Equations Transport Turbulence model are being Analyzed

- High speed turbulence models are somewhat Deficient (The deficiencies are well documented(Marvin(1986), Wilcox(1993)).
  
  Effect of Compressibility
  An-isotropy (Low/High Speed).
  Non-Equilibrium Flow Features (Low/High Speed).
  Near-Wall Flow(Low-Reynolds Number Features (Low/High Speed)).
  Inflexibility of handling Complex Geometry- Invariance Principle
  (Low/High Speed)
  Large Dependence in the Numerical Methods Used
  (especially elliptic Solvers).
  Appropriate Initial/Boundary Conditions
  Etc ...
K-ε Model-RPLUS Development

- LU Based k-ε Model Solver-De-coupled Approach.  
  Mean-Turbulence Transport Equations
  LU-SSOR- Yoon and Shuen- Explicit Terms Centrally Differenced  
  LU-SW -Steger and Warming- Explicit Terms Upwind Differenced

k-ε Models
- Convective Terms + Diffusive Terms + Source Terms = 0.0  
  Model Only differ in Low-Reynolds Number Character.  
  Models performance are being Evaluated.  
  Implicit Source Term Handling Strategy also Being Studied

k-ε Turbulence Models being studied for potential used in Three Dimensional RPLUS Code.

- Low-Reynolds Number Model plus Dilatational Terms  
  Chien (1976)  
  Launder-Shima(1976)  
  Shih(1990)  
  Various CMOTT derivatives of k-ε Model  
  Realizability  
  Invariance  
  Simplified Boundary-Conditions

- Performance of the Low-Reynolds number K-ε model in low-Mach number flows have been demonstrated (Patel, Rodi and Scheuerer(1985), Steffen(1993), Launder(1992)).

- Some of the Potential Difficulties in high speed turbulence model are well documented (Marvin(1993), Coakley and Huang(1992)).
Evaluation and Development of the RPLUS/k-ε Model Solver

• Various 2D-3D problems are being studied to optimize the numerical method and to evaluate model performance in supersonic flows in context to the LU based numerical technique.

• Simple 2D k-ε models are also being used to study various components of the flowfield generated by the complex combustor geometry previously shown.

• Studying the Numerical method/Model Behavior/Model Performance.
  - 2D Shear-Layer - Mixing (H. Lai (1993))
  - 3D Fin/Flat Plate Interaction- 3D Corner Flows-Interaction Developed through a Fin generated Shock-Waves. (D. Davis (1992))
Supersonic Mixing Layer

RPLUS vs. Dutton (case 2)

- RPLUS
- Dutton, case 2

Boundary Layer
Mach 2.87
Re/m = 6.3x10^7/m
Other Factors

- Optimum Numerical Strategy with in LU frame work.
- Effects of Initial condition.
- Modeling of Compressibility terms/Dilatational terms.
- Modeling of Turbulent terms in the Finite Rate Chemistry Model.
- Anisotropy of Turbulence
- Effects Upstream and Down stream Influences (Inlet(K. Kapoor) and Diffuser(?)).
- Chemistry-Turbulence Model Interactions (A. Hsu-PDF).
- Numerical Robustness(A. Suresh).
Validation of a k–ε Model in RPLUS2D Code for Non–reacting/reacting Subsonic Shear layers

H. T. Lai
Workshop on Computational Turbulence Modeling
September 15–16, 1993

OVERVIEW

– To simulate an experiment at NASA Lewis by Marek and co–workers for subsonic shear layers
– Computations for nonreacting/reacting flows
– Grid refinement study for nonreacting case
– Compressibility effects in k–ε models (nonreacting)
– Torch effects to sustain combustion (reacting)
The RPLUS Code

- Navier–Stokes and species conservation equations
- Finite–volume scheme
- Options for central or upwind differences
- Jameson–type dissipation
- LU–ADI algorithm
- Vectorization on oblique grid lines/planes
- Hydrogen–air chemistry
- Implicit chemical source terms
- High Reynolds number k–e model
- k–e equations uncoupled from other equations

NONREACTING SHEAR LAYER

<table>
<thead>
<tr>
<th>U=140 m/s, T=304.3°K, P=1.013x10^5 N/m^2</th>
<th>y=5.08cm</th>
</tr>
</thead>
<tbody>
<tr>
<td>ti=2.25%, AIR</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>U=390 m/s, T=822° K, P=1.1013x10^5 N/m^2</th>
<th>y=-5.08cm</th>
</tr>
</thead>
<tbody>
<tr>
<td>ti=2.25%, AIR</td>
<td></td>
</tr>
</tbody>
</table>

GEOMETRY AND FLOW CONDITIONS

GRID DISTRIBUTION
NONREACTING SHEAR LAYERS

PROFILES OF AXIAL MEAN VELOCITY

PROFILES OF TOTAL RMS FLUCTUATION

NONREACTING SHEAR LAYER

MACH NUMBER CONTOURS

TURBULENT KINETIC ENERGY CONTOURS

DISSIPATION RATE CONTOURS

M=0.401

M=0.679

\(k=100\)  \(k=1900 \text{m}^2\text{s}^2\)

\(e=1\times10^5\)  \(e=3.5\times10^7 \text{m}^2\text{s}^3\)
PROFILES OF DISSIPATION RATE

SHEAR LAYER THICKNESS VARIATION

GROWTH RATE VS. COMPRESSIBILITY EFFECT
**REACTING SHEAR LAYER**

- $U=135 \text{m/s}, T=366.5^\circ \text{K}, P=1.013 \times 10^5 \text{N/m}^2$
- $t_i=5\%, 3.94\% \text{H}_2 \text{ and } 96.06\% \text{N}_2$

- $U=385 \text{m/s}, T=810.9^\circ \text{K}, P=1.013 \times 10^5 \text{N/m}^2$
- $t_i=6\%, \text{AIR}$

**FLOW CONDITIONS**

- Hydrogen torch: dimension $=1 \text{cm}$
  - $t_i=5\%, T=1250^\circ \text{K}, 1.3\% \text{H}_2 + 21\% \text{O}_2 + 77.7\% \text{N}_2$
  - $V=0$ (parallel inflow)

**PARALLEL INFLOW**

- Hydrogen torch: dimension $=2.12 \text{cm}$
  - $t_i=11\%, T=1250^\circ \text{K}, 0.1\% \text{H}_2 + 21\% \text{O}_2 + 78.9\% \text{N}_2$
  - $V=50 \text{m/s}$ (oblique inflow)

**OBlique INFLOW**

**REACTING SHEAR LAYERS**

- **MEASURED**
- **RPLUS(OBLIQUE)**
- **RPLUS(PARALLEL)**
- **TSC-PDF**

**PROFILES OF AXIAL MEAN VELOCITY**

**PROFILES OF TOTAL RMS FLUCTUATION**

219
REACTING SHEAR LAYERS
TEMPERATURE CONTOURS

T = 300° K
T = 1900° K

PARALLEL INFLOW

OBlique INFLOW

REACTING SHEAR LAYERS
H_2O MASS FRACTION CONTOURS

c = 0.01
c = 0.125

PARALLEL INFLOW

OBlique INFLOW
CONCLUSIONS

- Growth rate underpredicted, especially reacting
- Large decay of k near shear layer edges
- Small production of k or large dissipation rate e
- But center peaks in k are constant and overpredicted
- Small improvement with grid refinement
- Inhibition of growth with compressibility effect
- Significant torch effects for reacting flows
Page intentionally left blank
Computational Aero-Acoustics as a Branch of Turbulence Research

Reda R. Mankbadi
NASA Lewis Research Center
Cleveland, Ohio

Noise Produced by Large-Scale Coherent Structure in Subsonic Jets

• The initial region of jet is dominated by large-scale, wave-like coherent structure, which is believed to be the dominant sound source.

• The coherent structure can be calculated by:
  Splitting the flow field into three components: time-averaged, coherent, and random.
  The coherent component is represented by few frequency modes which are taken to resemble a nonlinear instability wave interacting with the mean flow, turbulence, and other coherent components.
  Integral equations are then obtained for each scale of motion.
  Lighthill theory is used with the stress term given by the coherent structure.

I. Background

• Acoustic Analogy - Lighthill's Theory:

  \[ P_j = \frac{1}{4\pi R_0^2} \iiint \frac{\partial}{\partial t} \left( \frac{\partial\rho \mu}{\partial t} \right) \, dV. \]  

  The curly brackets denote that the source term is calculated at the retarded time

  \[ t_r = t - \frac{|X - Y|}{a_0}. \]  

  where \( X \) and \( Y \) are the observer's and the source's locations, respectively.

• Working with time-averaged properties

• Modeled time-dependent sound source

• Noise radiation from linear instability wave

• Noise radiation from large-scale coherent structure

CAA : Jet Noise

II. Prediction of the time-dependent sound source using Large-Eddy Simulations (LES)

III. Sound propagation to the far-field

What is Computational Aero-Acoustics (CAA)?

CAA is concerned with calculations of the aerodynamically-generated sound source and its propagation:

• The time-dependent flow fluctuations (sound source) are obtained starting from the time-dependent differential equations.

• High-order accurate schemes and appropriate boundary conditions for wave-like solutions are needed.

• Differential or integral techniques for sound propagation.
Calculated Spectra of Sound Intensity in Decibels Referred to $10^{-12}$ W m$^{-2}$ Due to Coherent Structures at Various Emission Angles for $U_0 = 195$ m s$^{-1}$
II. Large-Eddy Simulations For Prediction of The Sound Source in a Supersonic Jet

- DNS can predict the full spectrum of the sound source—But, the resolution requirements are prohibitive.

- In large-eddy simulations (LES) the unresolved, small scales are modelled. The acoustically active, larger scales are obtained directly from simulation.

The Study Indicates:

- The large-scale structure seems to be the dominant sound source.

- Results are sensitive to approximation in the sound source.

- Lighthill's theory predicts some results consistent with observations and some are not. No explicit Acoustic-Flow interactions
Definition of the source term is debatable
Harmonic excitation

Inflow disturbances in the form

\[ \tilde{u} = \bar{u} + \epsilon e^{-\omega^2 t^2} \sum_{i=1}^{16} \sin k \omega_i t, \]

(5)

where \( \omega \) was taken \( \omega/8 \). The Gaussian profile of the disturbance was introduced to reduce the adjustment zone.

Discretization:

- A fourth order accurate in space and second-order accurate in time MacCormick Scheme is used (2-4, Gottlieb & Turkel).
- An operator splitting is used to maintain the 2-4 accuracy, namely

\[ Q^{n+1} = L_p L_2 L_1 Q^n, \]

(3)

where \( L_1 \) and \( L_2 \) are one-dimensional solution operators corresponding to the scheme applied to the equations

\[ Q_s^{n+1} = Q_s^n + \Delta t S, \quad Q_s^{n+1} = Q_s^n + \Delta t S, \]

(4)

SGS Model:

- Smagorinski's model is used for the SGS turbulence.

Boundary conditions:

- Objective is to obtain the time-dependent ("wave-like") structure. Boundary conditions could create artificial disturbances or could dampen the physical disturbances. Special attention is needed.
- Several outflow boundary conditions are evaluated.
- Linearized characteristics (e.g., Bayless & Turkel) are used to derive the B.C. used herein.
III. Far-Field Sound

(1) Extend Navier-Stokes computational domain to the far field:
   (a) Prohibitive storage requirements
   (b) Acoustic scales are different from fluid scales.

(2) Lighthill's acoustic analogy

(3) Finite-difference of linearized equations

(4) Kirchhoff's method

---

Strouhal number, $St$

- $0.125$
- $0.25$
- $0.375$
- $0.5$

Amplitude vs. Streamwise location, $x/D$

Amplitude of harmonic disturbances.
(3) Finite-difference approach ... Linearized Euler equations

- Finite-differencing can be used to solve the linearized Euler equations or other equations (Liley, Phillips) describing the sound propagation to the far-field.
- The problem is that numerical dissipation and dispersion can lead to erroneous results for or the far-field sound.
- Goodrich (1993) developed a new algorithm that seems to be useful for this purpose. The scheme is tested for 1D linearized Euler equations and seems to be accurate at very long times with few mesh points.
SUMMARY

FLOW FIELD:
- The time-dependent sound source can be predicted via careful large-eddy simulations.
- For random inflow-disturbances the wave-like nature of the unsteady structure is evident for Strouhal numbers of up to about 1.2.
- The large-scale structure (St < 1.2) could be enhanced via harmonic excitation -- Potential for control.
- The higher frequency modes peak closer to the jet exit and the lower frequency ones peak farther downstream.

FAR-FIELD SOUND:
- Lighthill's: Limited success
  - No explicit account for sound-flow interaction
  - Source is assumed to be compact, but it is no-compact for supersonic jets.
- Finite-Difference:
  A high-accuracy scheme is being evaluated.
- Kirchhoff's method:
  - The predicted pressure on a cylindrical surface enclosing the jet is used to predict the far-field sound -- Most promising.

- Outside the source region the sound transmission is governed by the convective wave equation.
- The sound pressure field is given in terms of a surface integral involving the numerically calculated surface pressure.
- Evaluated for a point source.

• Predicted directivity of jet noise seems to be consistent with observation.
JET NOISE PREDICTION USING A k-ε TURBULENCE MODEL

A. Khavaran
Sverdrup Technology, Inc.
NASA Lewis Research Center

Lewis Internal Workshop
on
Computational Turbulence Modeling
September 15-16, 1993

MODELING APPROACH

• Source Spectrum Calculations
  - Acoustic Analogy
  - Ribner and Batchelor Assumptions
  - Calculation of the Source Spectrum and its Characteristic Frequency Based
    on Time-Averaged Flow Calculation (with a k-ε Turbulence Model)

• Sound/Flow Interaction
  - High Frequency Asymptotic Solution to Lilley's Eq. for Multipole Sources
    Convecting in an Axisymmetric Parallel Flow (Balsa & Mani)
GOVERNING EQUATIONS

Lighthill’s Equation

\[
\frac{\partial^2 p}{\partial t^2} - c^2 V^2 p = \frac{\partial^2 T_{ij}}{\partial x_i \partial x_j}
\]

\[T_{ij} = \rho V_i V_j + \delta_{ij}(p - c^2 \rho) - e_{ij}\]

\[e_{ij} = \mu \left( \frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial V_k}{\partial x_k} \right)\]

\[
\frac{e_{ij}}{\rho V_i V_j} \sim O \left( \frac{1}{Re} \right), \quad Re = \frac{\rho U L}{\mu}
\]

\[\frac{1}{c^2} \frac{dp}{dt} - dp = \left( \frac{\partial p}{\partial s} \right)_p ds
\]

- The effects of source convection and refraction are included in the source term

Lilley’s Equation

\[
\frac{D}{Dt} \left[ \frac{D^2 \sigma}{Dt^2} - \frac{\partial}{\partial x_i} \frac{\partial \sigma}{\partial x_i} \right] + 2 \frac{\partial V_j}{\partial x_i} \frac{\partial}{\partial x_j} \frac{\partial \sigma}{\partial x_i} = -2 \frac{\partial V_j}{\partial x_i} \frac{\partial V_i}{\partial x_j} \frac{\partial V_k}{\partial x_k} + \frac{D}{Dt} \left[ \frac{D}{Dt} \left( \frac{1}{\gamma} \frac{Ds}{Dt} \right) \right] + \text{viscous terms}
\]

\[
\sigma = \frac{1}{\gamma} \frac{p}{p_0}
\]

\[
\frac{D}{Dt} = \frac{\partial}{\partial t} + V_k \frac{\partial}{\partial x_k}
\]

- The effects of source convection and refraction are included in the operator term of Lilley’s eq.
Source Spectrum Calculations

- Mean-square sound pressure autocorrelation in the far field due to a finite volume of turbulence (in absence of convection and fluid shielding)

\[ \overline{p^2}(R, \theta, \phi) = \frac{R_t R_j R_k R_l}{16\pi^2 C_4^2 R_0^5} \int_0^\infty \int_0^{\pi} \frac{\partial^4}{\partial \sigma^4} (\rho V_i V_j) (\rho' V'_k V'_l) \, d\xi \, d\eta, \]

- Fourth-order velocity correlation tensor

\[ S_{ijkl} = \overline{v_i v_j v'_k v'_l} = \int_{-\infty}^{+\infty} (v_i v_j)(v'_k v'_l) \, dt \]

- Source strength (Quasi-incompressible turbulence)

\[ I_{ijk} = \rho^2 \int \frac{\partial^4}{\partial \xi^4} S_{ijkl} \, d\xi \]

- Reduction in order of correlation tensor

\[ S_{ijkl} = S_{ik} S_{jl} + S_{il} S_{jk} + S_{ij} S_{kl} \]

\[ S_{ij}(\tau, \xi) = \int_{-\infty}^{+\infty} v_i v_j \, dt \]
- Separable second-order tensors

\[ S_{ij}(\tau, \xi) = R_{ij}(\xi)G(\tau) \]

- Isotropic turbulence model of Batchelor

\[ R_{ij}(\xi) = T e^{-\pi (\xi/L_z)^2} \times \left\{ \left[ 1 - \pi (\xi/L_z)^2 \right] \delta_{ij} + \pi \xi_i \xi_j/\xi_2 \right\} \]

\[ T = \frac{1}{3} \frac{1}{v_i v_i}, \quad \xi^2 = \xi_1^2 + \xi_2^2 + \xi_3^2 \]

- Gaussian correlation time delay

\[ G(\tau) = e^{-\left(\tau/\tau_o\right)^\gamma} \]

- Source spectrum component

\[ I_{1111}(\Omega) \sim \rho^4 k^4 (\Omega \tau_o)^4 e^{-\Omega \tau_o^\gamma} \]

\[ L_z \sim \frac{k^{3/2}}{\epsilon}, \quad \tau_o \sim \frac{L_z}{\sqrt{k}}, \quad k = \frac{1}{2} \frac{1}{v_i v_i} \]

- Characteristic time delay of correlation

\[ \tau_o \sim \frac{1}{(\partial U/\partial r)} \text{ or } \tau_o \sim \frac{k}{\epsilon} \]

- Doppler shifted frequency

\[ \Omega = 2\pi f \bar{C}, \quad \bar{C} = \sqrt{(1 - M_c \cos \theta)^2 + (\sigma_c k^3/C_{\infty})^2} \]

\[ M_c = 0.5 M + \beta_c \dot{M}_j \]
Sound/Flow Interaction

- Mean square pressure in the far field

\[ \overline{p^2}(R, \theta, \Omega) = \int Y(a_{xx} + 4a_{xy} + 2a_{yy} + 2a_{yx})d\Omega \]

- Source term

\[ \Lambda \sim \frac{(\frac{E_0}{\rho})^2 I(\Omega)}{(4\pi RC_{\infty} C)^2 (1 - M\cos\theta)^2 (1 - M_c\cos\theta)^2} \]

- Shielding function

\[ g^2(r) = \frac{(1 - M\cos\theta)^2 (\frac{E_0}{\rho})^2 - \cos^2\theta}{(1 - M_c\cos\theta)^2} \]

\[ M(r) = \frac{U(r)}{C_{\infty}} \quad M_c = \frac{U_c}{C_{\infty}} \]

Design Parameters for C-D Nozzle

- Throat Diameter: 5.1 in.
- Exit Diameter: 5.395 in.
- Distance from Throat to Exit: 5.525 in.
- Exit Velocity: 2400 fps
- Ambient Velocity: 400 fps
- Pressure Ratio: 3.121
- Stagnation Temp.: 1716°F
Comparison of the Velocity Profiles With Data

Centerline

Comparison of Velocity Profiles With Experimental Data at Four Different Axial Locations
Comparison of the Turbulent Intensity Profiles With Data

Lip-line

\[ \frac{\partial U}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial y} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial y} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]

\[ \frac{\partial (\partial U)}{\partial x} (D/U_t) \]
OASPL DIRECTIVITY (ARC=40 ft)

![OASPL Directivity Graph]

Design Parameters for C-D Nozzle
- Exit Diameter: 5.395 in.
- Throat Diameter: 5.1 in.
- Exit Velocity: 2409 fps
- Pressure Ratio: 3.121
- Stagnation Temperature: 1716 R

NOISE SPECTRA (CD Nozzle)

- θ=20°
- θ=60°
- θ=100°
- θ=120°

Band 15 is 125 Hz
Design Parameters for CPN

Exit Inner Diameter   .597 in.
Exit Outer Diameter   1.77 in.
Pressure Ratio        3.61
Stagnation Temperature 532 R
**SUMMARY**

- Source strength was evaluated using the PARC code with a k-ε turbulence model.
- The characteristic Strouhal no. was obtained from k and ε (with an empirical proportionality constant).
- Time-averaged velocity and temperature predictions were used to assess the sound/flow interaction.
- Constants used in the supersonic convection factor are determined empirically.
- The SPL directivity and spectra demonstrate favorable agreement with data (specially at angles close to the jet axis).
- The empirical constants used in these predictions need to be investigated for other geometries.
NUMERICAL SIMULATION OF UNSTEADY SUPersonic FLOW USING AN IMPLICIT ALGORITHM FOR THE STRONGLY COUPLED NAVIER-STOKES AND K-E EQUATIONS

by

S.H. Shih*, A. Hamed# and J.J. Yeuan#

* ICOMP, NASA Lewis Research Center
# Dept. of Aero. Eng. & Eng. Mechanics
University of Cincinnati, Cincinnati, Ohio
METHODOLOGY

• MEAN FLOW AND TURBULENCE EQUATIONS ARE INTEGRATED SIMULTANEOUSLY
• FULL N-S EQUATIONS IN STRONG CONSERVATION LAW FORM
• NICHOLS K-E TURBULENCE MODEL (1990)
• BEAM-WARMING SCHEME (1978)
• FULLY IMPLICIT TREATMENT OF TURBULENCE SOURCE TERMS
• NEWLY PROPOSED 2nd-ORDER DAMPING

INTRODUCTION

TWO-EQUATION TURBULENCE MODEL:
• TRANSPORT EQUATIONS
• CONSIDER FLOW HISTORY, LESS EMPIRICISM.

PREVIOUS WORKS BY OTHER RESEARCHERS:
• UNCOUPLED N-S AND K-E EQUATIONS
• STEADY FLOW:
  TIME LAGGING BETWEEN MEAN FLOW AND TURBULENCE
  [Huang & Coakley (1992), Sahu & Danberg (1986), PARC Code (1989)]
• UNSTEADY FLOW:
  SUBITERATION TECHNIQUE REQUIRED
  [Rizzetta & Visbal (1992)]
\[ P_t = C_{s1}P_{R2} + C_{s3}P_{RJ} - C_{s4}\left( \frac{\partial u}{\partial x} \frac{\partial u}{\partial y} \right) + \tau_{\nu} \left( \frac{\partial u}{\partial y} \frac{\partial v}{\partial x} \right) + p k \]

(A.17)

where

\[ P_{RJ} = (\tau_{\nu})_{w} \frac{\partial u}{\partial x} + (\tau_{\nu})_{w} \frac{\partial v}{\partial y} \quad P_{R2} = (\tau_{\nu})_{w} \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \]

(A.18)

with the incompressible turbulent stress expressed as

\[ (\tau_{\nu})_{w} = \frac{\mu_t}{Re} \left( \frac{\partial u}{\partial x} \frac{\partial v}{\partial y} \right) \]

(A.19)

and the coefficients \( C_{s1} \) and \( C_{s3} \) given as

\[ C_{s1} = 1.35 \quad C_{s3} = C_{s4} + 3.0 \exp(-3.5(PRJ/PR2)^{0.3}) \]

(A.20)

\[ \mu_t = Re C_{s6} \rho \frac{k^2}{\varepsilon} \]

(A.21)

The empirical constants for the model are given by (28) as follows:

\[ a_2 = 1.0 \quad c_1 = 1.3 \quad C_2 = 1.8 \left( 1 - \frac{2}{3} \exp \left( -\frac{R_t}{50} \right) \right) \]

(A.22)

\[ C_{\gamma} = 0.09 \left[ 1 - \exp(-0.0115\psi) \right] \]

(A.23)

\[ R_t = Re \frac{k^2}{\mu \varepsilon} \quad y^+ = Re \frac{\rho u_{\tau}}{\mu \varepsilon} \quad u_{\tau} = \left( \frac{\tau_{\nu}}{\mu} \right)_{w} \]

(A.24)

**Turbulence Model**

- Based on Chien’s (1982) Low Reynolds number turbulence model
- Modified for compressibility effects and irrotational strains, Nichols (1990)
- Integrated to wall, without the use of wall functions

\[ \frac{\partial U_t}{\partial t} + \frac{\partial F_t}{\partial x} + \frac{\partial F_t}{\partial y} = S_t \]

(A.11)

where the turbulent state vector \( U_t \), flux vectors \( E_t, F_t \), and source vector \( S_t \), are given by:

\[
U_t = \begin{bmatrix}
\rho k \\
\rho e \\
\end{bmatrix}, \quad E_t = \begin{bmatrix}
\rho u_k - \frac{1}{Re} \frac{\partial \rho u_k}{\partial x} \\
\rho u_e - \frac{1}{Re} \frac{\partial \rho u_e}{\partial x} \\
\end{bmatrix}, \quad F_t = \begin{bmatrix}
p + \frac{1}{Re} \frac{\partial \rho u_k}{\partial y} \\
p + \frac{1}{Re} \frac{\partial \rho u_e}{\partial y} \\
\end{bmatrix}
\]

(A.12)

The turbulence production terms \( P_t \) is expressed as follows:

\[ F = \frac{\partial \rho u}{\partial t} + \tau_{\nu} \left( \frac{\partial u}{\partial y} \frac{\partial v}{\partial x} \right) + \frac{\partial \rho u}{\partial y} \]

(A.13)

\[ \tau_{\nu} = \frac{\mu_t}{Re} \left( \frac{\partial u}{\partial y} \frac{\partial v}{\partial x} \right) - \frac{2}{3} \frac{\rho u_{\tau}}{Re} \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) + \frac{1}{Re} \frac{\partial \rho u_k}{\partial y} \]

(A.14)

\[ C_{\mu} = C_{\mu}(\gamma - 1) M_{\alpha}^2, \quad M_{\alpha} = \frac{\alpha \rho u_{\tau}}{a} \quad a = \text{sonic velocity}, \quad C_{\mu} = 4.0 \]

(A.16)
NUMERICAL ALGORITHM

- GOVERNING EQUATION

\[
\frac{\partial \hat{U}}{\partial \hat{t}} + \frac{\partial \hat{E}}{\partial \hat{X}} + \frac{\partial \hat{F}}{\partial \hat{V}} + \frac{\partial \hat{W}}{\partial \hat{S}} = \hat{H}
\]

- BEAM-WARMING SCHEME

\[
\Delta^* U = \Delta U + \Delta^* U_1 + \Delta^* U_2 + \Delta^* U_3 + \Delta^* U_4 + \Delta^* U_5
\]

- THE N-S AND K-E EQUATIONS ARE INTEGRATED SIMULTANEOUSLY

- SOURCE TERMS ARE TREATED FULLY IMPLICIT

- LINEARIZATION

- INVISID TERMS

\[
E^{*1} = E^* + \frac{\partial E}{\partial U} (U^{*1} - U^*) + O((\Delta t)^2)
\]

\[
\Delta^* E = \Delta^* (U^{*1} - U^*) + O((\Delta t)^2)
\]

\[
A = \frac{\partial \hat{E}}{\partial \hat{U}}
\]

where

VISCIOUS TERMS

The viscous delta term \( \Delta^* V_i(U,U) \) is linearized as follows:

\[
\Delta^* V_i = \left( \frac{\partial V_i}{\partial U} \right)_J U + \left( \frac{\partial V_i}{\partial U} \right)_J U + O(\Delta t)
\]

where the Jacobian matrices are written as:

\[
[J] = \frac{\partial V_i}{\partial U}, \quad [J] = \frac{\partial V_i}{\partial U}
\]

If the viscosity is assumed locally constant, then

\[
[J] = [J] = 0
\]

The viscous Jacobian matrix \( R \) is given by:

\[
R = \frac{1}{\rho} \begin{bmatrix}
0 & 0 & 0 & 0 & 0 \\
\lambda & \lambda_1 & \lambda_2 & 0 & 0 \\
-\lambda & \lambda_1 & \lambda_2 & 0 & 0 \\
0 & \lambda & \lambda_1 & \lambda_2 & 0 \\
0 & 0 & \lambda & \lambda_1 & \lambda_2 \\
\end{bmatrix}
\]
ARTIFICIAL DISSIPATION

Step 1:
\[ (I + \frac{\theta_1 \Delta t}{1+\theta_2} \frac{\partial R^* - D^* e_\mu e_\mu}{\partial t}) \Delta^* U^* \]
= RHS[Eq.(4.5)] - \frac{\theta_1 \Delta t}{1+\theta_2} (e_\mu (\Delta^* e_\mu) + e_\nu (D_1^* + D_2^*)) U^*

Step 2:
\[ (I + \frac{\theta_1 \Delta t}{1+\theta_2} \frac{\partial R^*}{\partial \eta} - \frac{\partial U^*}{\partial \eta} + e_\mu, e_\mu, e_\nu, e_\nu) \Delta^* U = \Delta^* U^* \]

Step 3:
\[ U^{n+1} = U^* + \Delta^* U \]

where
\[ e_\mu = e_\mu + e_\nu D_1 \quad e_\nu = e_\nu + e_\mu D_2 \]

\[ D_1 = \frac{1}{4 \rho} \frac{\partial^2}{\partial t^2} + \frac{1}{4 \kappa} \frac{\partial^2}{\partial t^2} \]
\[ D_2 = \frac{1}{4 \rho} \frac{\partial^2}{\partial \eta^2} + \frac{1}{4 \kappa} \frac{\partial^2}{\partial \eta^2} \]

* SOURCE TERMS

\[ D = \frac{\partial R}{\partial U} \quad D^* = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ h_{11} & h_{12} & h_{13} & h_{14} & h_{15} & h_{16} \\ h_{21} & h_{22} & h_{23} & h_{24} & h_{25} & h_{26} \end{bmatrix} \]

* THE FINAL DIFFERENCE FORMULA

\[ \begin{align*}
U &= \frac{\theta_1 \Delta t}{1+\theta_2} \left[ \frac{\partial R^*}{\partial t} - \frac{\partial U^*}{\partial \eta} \right] \Delta^* U \\
&= \frac{\Delta t}{1+\theta_2} \left[ \frac{\partial}{\partial t} (-E^* \cdot \nabla U^* - E^* \cdot \nabla U^*) - \frac{\partial}{\partial \eta} (-F^* \cdot \nabla W^* - H^*) \right] \\
&= \frac{\theta_1 \Delta t}{1+\theta_2} \left[ \frac{\partial}{\partial t} (\Delta^* U^*) - \frac{\partial}{\partial \eta} (\Delta^* W^*) \right] - \frac{\theta_1 \Delta t}{1+\theta_2} \Delta^* U
\end{align*} \]

* BLOCK TRIDIAGONAL STRUCTURE

* 6x6 JACOBIAN MATRICES IN 2-D FLOWS
Fig. 41. Computational Grid for Supersonic Flow Over an Open Cavity.

\[(\xi_x + \xi_y \frac{\partial u}{\partial x} + (\eta_x + \eta_y \frac{\partial u}{\partial y}) \frac{\partial u}{\partial \eta} = 0 \]

\[\theta = \arcsin(1/M)\]

Fig. 40. Schematic of the Solution Domain for Supersonic Flow Over an Open Cavity.
RESULTS

- SUPersonic FLOW OVER AN OPEN CAVITY.

RIZZETTA & VISBAL (1992)

- BEAM-WARMING ALGORITHM
- LAUNDER & SHARMA (1974) LOW REYNOLDS NUMBER TWO-EQUATION TURBULENCE MODEL
- TURBULENCE EQUATIONS ARE UNCOUPLED FROM THE N-S EQUATIONS AND FROM EACH OTHER
- NEWTON-LIKE SUBITERATION TECHNIQUE
Instantaneous Density Contours

Fig. 2.

Typical Pressure Oscillation Cycle.
Ref. [67].
Instantaneous Velocity Vectors

Instantaneous Static Pressure Contours
Fig. 48. Overall Sound Pressure Level Distribution Along the Cavity Floor.

$SPL = 10 \log_{10}(\overline{p^2}/q^2)$

$\overline{p^2} = \frac{1}{(t-t_i)} \int_0^t (p-p_i)^2 \, dt$
CONCLUSIONS

A NUMERICAL PROCEDURE WAS DEVELOPED FOR
THE SIMULTANEOUS IMPLICIT SOLUTION OF THE
COUPLED NAVIER-STOKES AND K-E EQUATIONS

THE RESULTS DEMONSTRATE:
• SELF-SUSTAINED FLOW OSCILLATION WITHIN
  THE CAVITY IS SIMULATED
• OVERALL SOUND PRESSURE LEVELS ARE
  WELL-PREDICTED
• RELIEVE STIFFNESS, ENHANCE STABILITY,
  AND LARGER CFL NUMBER
• 20 CPU HOURS ON CRAY-YMP

Fig. 49. Pressure Time History on the Aft Bulphead at y/D = 0.4.
IMPLEMENTATION OF A $k-\varepsilon$ MODEL IN SPECTRAL ELEMENT CODE

Wai-Ming To

Sverdrup Technology, Inc.

OBJECTIVES

- Study the flow physics in problems that are generic to turbomachinery flows.
- Assess accuracy of turbulence models.
EQUATIONS OF MOTION

\[
\frac{\partial u_i}{\partial x_i} = 0
\]

\[
\frac{\partial u_i}{\partial t} = N_i - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ v_e \left\{ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right\} \right]
\]

where \( v_e = v_l + v_t \)

TIME DISCRETIZATION

- 2nd order predictor-corrector
- Capacitance matrix method to solve for pressure
TIME DISCRETIZATION (cont.)

\[
\begin{align*}
\frac{\partial u_i^{(1)}}{\partial x_i} &= 0 \\

u_i^{(2)} &= u_i^n + \Delta t f_i - \frac{\Delta t}{2} \frac{\partial \bar{p}^{(1)}}{\partial x_i} + \frac{\Delta t}{2} \frac{\partial}{\partial x_j} \left[ v_e \left\{ \frac{\partial u_i^n}{\partial x_j} + \frac{\partial u_j^n}{\partial x_i} \right\} \right] + \frac{\Delta t}{2} \frac{\partial}{\partial x_j} \left[ v_e \left\{ \frac{\partial u_i^{(1)}}{\partial x_j} + \frac{\partial u_j^{(1)}}{\partial x_i} \right\} \right] \\
\frac{\partial u_i^{(2)}}{\partial x_i} &= 0
\end{align*}
\]

TIME DISCRETIZATION (cont.)

\[
\begin{align*}
\frac{\partial u_i^{n+1}}{\partial x_i} &= 0 \\
u_i^{n+1} &= u_i^n + \Delta t f_i - \frac{\Delta t}{2} \frac{\partial \bar{p}^{(1)}}{\partial x_i} + \frac{\Delta t}{2} \frac{\partial}{\partial x_j} \left[ v_e \left\{ \frac{\partial u_i^{n+1}}{\partial x_j} + \frac{\partial u_j^{n+1}}{\partial x_i} \right\} \right] + \frac{\Delta t}{2} \frac{\partial}{\partial x_j} \left[ v_e \left\{ \frac{\partial u_i^n}{\partial x_j} + \frac{\partial u_j^n}{\partial x_i} \right\} \right]
\end{align*}
\]

where \( \bar{p} = p^{(2)} + p^n \)
TURBULENCE MODEL

- Eddy viscosity concept
  subgrid model $\rightarrow$ 2-equation model

- Zero equation model

$$v_t = \kappa^2 y^2 \left[ 1 - e^{-y^+/26} \right] \left| \frac{\partial u}{\partial y} \right|$$

$$v_t = 0.018 u_{\infty} \delta^*$$

TURBULENCE MODEL (cont.)

- Wall boundary conditions

$$\frac{\tau_w}{\rho} = \begin{cases} 
  \frac{v_t}{\kappa y} \frac{u}{1 \ln Ey^+} & y^+ > B \\
  v_l \frac{u}{y} & y^+ \leq B 
\end{cases}$$

where

$$y^+ = \frac{y (\tau_w/\rho)^{1/2}}{v_l} = \frac{1}{\kappa} \frac{v_t}{v_l}$$
NUMERICAL RESULTS (laminar flow)

\[ u = 0, \quad \frac{\partial v}{\partial y} = 0 \]

\[ \frac{\partial u}{\partial x} = 0, \quad \frac{\partial v}{\partial x} = 0 \]

\[ u = 0, \quad v = 0 \]
INTRODUCTION

Real flow simulation is needed
(laminar region + turbulent region)

for:

(1) cruise drag prediction
(2) separation angle of attack prediction
Difficulties:

- Transition is a function of many variables
  - free stream turbulence
  - surface roughness
  - streamwise surface curvature
  - pressure gradient
  - etc.

- Skin friction coefficient which is the best transition indicator is difficult to calculate

METHODS OF TRANSITION PREDICTION

- Euler/boundary layer approach

- Euler/boundary layer/stability analysis

- Navier–Stokes calculation with turbulence models
  - Baldwin–Lomax with transition
  - RNG turbulence model
TRANSITION PREDICTION

- Flat plate
- NACA0012
- ADP inlet
Boundary Layer Thickness of NACA0012 Airfoil
Local Skin Friction Coefficient for NACA0012 Airfoil

ADP CONVENTIONAL NACELLE AT CRUISE MACH NUMBER
Laminar/Turbulent Transition

Laminar region prediction:
Lewis' s boundary layer analysis : 8.17 % of chord length
Langley's stability analysis : 8.00 % of chord length

On going projects:
RNG turbulence model in PARC3D

266
CONCLUDING REMARKS

Several methods have been tested to predict the transition of an ADP inlet. The prediction of the transition at cruise conditions agrees very well among the methods investigated except for the Baldwin–Lomax/Transition model which predicts early transition. The accuracy of prediction methods will be judged by comparing with upcoming laminar flow experiment.
A Comparative Study of Turbulence Models in Predicting Hypersonic Inlet Flows

by

Kamlesh Kapoor
TEST CASE AND TURBULENCE MODELS CONSIDERED

- The NASA P8 inlet, which represents cruise condition of a typical hypersonic air-breathing vehicle, was selected as a test case for present study.
- PARC2D code, which solves the full two-dimensional Reynolds-averaged Navier-Stokes equations, was used for this study.
- The results are presented for a total of six versions of zero- and two-equation turbulence models.

- **Zero-equation models**
  - The Baldwin-Lomax model
  - The Thomas model
  - A combination of the B.L./Thomas model

- **Two-equation models**
  - Low-Reynolds number models
    - The Chien model
    - The Speziale model
  - High-Reynolds number model
    - The Launder and Spalding model

EXPERIMENTAL BACKGROUND

- The experimental investigation of the P8 inlet was conducted at NASA Ames' 3.5-foot hypersonic wind tunnel.
- The inlet was a Mach 7.4 rectangular mixed compression (with internal compression ratio of 8) design with exiting supersonic flow.
  - Inlet cowl height = 18.33 cm.
  - Overall length = 136.2 cm.

- Test conditions:
  - Mach No = 7.4
  - Total pressure = 4.14 x 10^6 N/m²
  - Total temperature = 811° K
  - Reynolds No = 8.86 x 10^6/m
  - Model was water cooled and isothermal wall conditions were maintained; the walls temperature = 302° K

- The transition points:
  - Centerbody - 40 percent from wedge L.E. edge to inlet entrance.
  - Cowl - halfway between inlet entrance and throat.
• Grid size was 221 x 91.
• Grid was nonuniform in X direction:
  - packed on both ends from the wedge L.E. to the cowl L.E.
  - geometrically stretched from the cowl L.E. to the exit of the inlet.
• In Y direction, the grid was packed using hyperbolic tangent function. Yplus was approximately 1 away from both walls.
• A separate grid was made for the launder and Spalding model and Yplus of approximately 30 was used away from the walls.
BOUNDARY CONDITIONS

NON-REFLECTIVE
NO-SLIP ISOTHERMAL
FIXED
SLIP
NO-SLIP ISOTHERMAL
EXTRAPOLATION

DENSITY CONTOURS FOR P8 INLET

CONTOUR LEVEL
PRESSURE CONTOURS FOR P8 INLET

CONTOUR LEVEL

MACH NUMBER CONTOURS FOR P8 INLET

CONTOUR LEVEL
PITOT PRESSURE AND TOTAL TEMPERATURE DISTRIBUTIONS AT X/XREF = 6.09

PITOT PRESSURE DISTRIBUTIONS

SOURCE: AGARD-AR-270
SEPT. 1991

AT X/XREF = 6.09

AT X/XREF = 6.37
PITOT PRESSURE AND TOTAL TEMPERATURE DISTRIBUTIONS AT X/XREF = 6.37

MACH NUMBER DISTRIBUTIONS AT THE THROAT
CONCLUSIONS

- A computational study has been conducted to evaluate the performance of various turbulence models.
- The Thomas model compares very well with the experimental data, and it performs best among the zero-equation models.
- The Baldwin–Lomax model and its combination with Thomas model are not able to resolve the problem of shock wave and boundary-layer interaction accurately. The Baldwin–Lomax model predicts separation near the interaction of the cowl shock with the wedge boundary layer, where none is known to exist in experiments.
- The Chien and Speziale model compare very well with the experimental data, and performs better than the Thomas model, particularly near the walls. The Launder and Spalding model does not perform as good as the Chien and Speziale models.
- As the CPU time required for the Thomas model is far less than the two-equation models, it is concluded that the Thomas model is best suited for the predictions of pressure distributions, and the Chien and Speziale models are best to calculate flow quantities near the walls.
Page intentionally left blank
AN ALGEBRAIC TURBULENCE MODEL
FOR TURBOMACHINERY

by

Rodrick V. Chima
NASA Lewis Research Center
Cleveland, Ohio
OVERVIEW

- MOTIVATION - TURBINE ENDWALL HEAT TRANSFER
- DESCRIPTION OF NEW MODEL
- RESULTS
  1. FLAT PLATE
  2. ANNULAR TURBINE CASCADE
  3. TURBINE ENDWALL HEAT TRANSFER
  4. SUPERSONIC COMPRESSOR BLADE
- SUMMARY

EXPERIMENTAL ENDWALL STANTON NUMBER CONTOURS
AS A FUNCTION OF $\delta_{\text{inlet}}$ AND $Re_{\text{chord}}$
RVC3D (ROTOR VISCOUS CODE 3-D)
BY R. V. CHIMA

DESCRIPTION
• EULER OR NAVIER-STOKES ANALYSIS
  FOR STEADY 3-D FLOWS IN TURBOMACHINERY

FEATURES
• CARTESIAN FORMULATION, ROTATION ABOUT X-AXIS
  RECTANGULAR OR ANNULAR GEOMETRIES
• SOLVES NAVIER-STOKES EQUATIONS
  THIN-LAYER FORMULATION, (NO STREAMWISE VISCOUS TERMS)
  RETAINS HUB-TO-TIP & BLADE-TO-BLADE VISCOUS TERMS
  BALDWIN-LOMAX OR CEBECI-SMITH TURBULENCE MODEL
  SIMPLE TIP CLEARANCE MODEL
• NODE-CENTERED FINITE-DIFFERENCE FORMULATION
  EXPLICIT 4-STAGE RUNGE-KUTTA TIME-MARCHING SCHEME
  2ND + 4TH ORDER ARTIFICIAL VISCOSITY, EIGENVALUE SCALING
  VARIABLE $\Delta t$ & IMPLICIT RESIDUAL SMOOTHING
  HIGHLY VECTORIZED & AUTOTASKED FOR CRAY Y-MP
• STACKED C-TYPE GRIDS

![Diagram of Baldwin-Lomax model with contour levels and Balakin-Lomax model with contour levels.](image)
**CEBECI-SMITH & BALDWIN-LOMAX MODELS**

**INNER LAYER: PRANDTL-VAN DRIEST FORMULATION**

**CEBECI-SMITH**

\[ \mu_i = \rho \frac{\partial u}{\partial y} \]

\[ l = \kappa y D \]

\[ D = 1 - \exp(-y^*/A^*) \text{ VAN DRIEST DAMPING} \]

**BALDWIN-LOMAX**

\[ \mu_i = \rho^2 |\omega| \]

**OUTER LAYER: CLAUSER FORMULATION**

**CEBECI-SMITH**

\[ \mu_o = K \rho \gamma^6 u_s \]

\[ \gamma = \left[ 1 + 5.5 \left( \frac{y}{\delta} \right)^6 \right]^{-1} \text{ KLEBANOFF INTERMITTENCY FUNCTION} \]

**BALDWIN-LOMAX**

\[ \mu_o = K \rho \gamma C_e \min \left\{ \frac{y_{max}}{f_{max}} \right\} \text{ wake option} \]

\[ f(y) = y |\omega| D \]
**Baldwin-Lomax Model Analysis**

(See paper for details)

1. Assume sublayer—wall—wake velocity profile

2. Calculate Baldwin-Lomax function \( f(y) \)
   - Max. occurs at \( y_{max} = 0.465 \)
   - Independent of pressure gradient
   - No max. for infinitely favorable \( \partial p/\partial x \)

3. Spurious max. can occur at edge of viscous sublayer
   - Most likely at low \( Re \) & favorable \( \partial p/\partial x \)

**Proposed Turbulence Model**

**Inner Layer (Similar to Baldwin-Lomax)**

\[
\begin{align*}
\mu_i &= \rho I|\omega| \\
I &= \kappa y D \\
D &= 1 - \exp(-y^+ A^+) \\
v^+ &= \frac{u^+}{\nu} \\
u^+ &= \sqrt{\frac{\tau_{wall}}{\rho}}
\end{align*}
\]
PROPOSED TURBULENCE MODEL

PRESSURE GRADIENT EFFECTS

- ACCELERATING FLOWS TEND TO RELAMINARIZE
- MODELLED BY INCREASING $A^+$ IN FAVORABLE $\partial p/\partial s$
- CEBECI'S EXPRESSION FOR $A^+$ USED:
  \[
  A^+ = \frac{26}{\sqrt{1 + 11.8p^*}}
  \]
  \[
  p^* = \frac{\nu}{\rho s} \frac{\partial p}{\partial s}
  \]
- PRESSURE GRADIENT EVALUATED USING:
  \[
  \frac{\partial p}{\partial s} \approx \frac{\Delta V}{\Delta s} \cdot \nabla p
  \]
- "EDGE VELOCITY" $\vec{V}_e$ EVALUATED AT A GRID LINE FAR ENOUGH FROM THE WALL TO GIVE THE GENERAL FLOW DIRECTION
- KAYS-MOFFATT EXPRESSION WAS TESTED, EFFECTS TOO STRONG

PROPOSED TURBULENCE MODEL

LOCAL SHEAR MODEL

- IN STRONGLY ACCELERATING FLOWS $r^+$ DECREASES WITH $y^+$
- MODELLED BY REPLACING $\tau_{\text{wall}}$ WITH $\tau(y)$ IN $D$

  \[
  D = 1 - \exp(-y^+/4^+)
  \]

  \[
  y^+ = \sqrt{\frac{\rho (\mu + \mu_t)}{\mu} |\omega|}
  \]
- ERROR IN ORIGINAL PAPER - USED $\mu |\omega|$ ONLY
- USED BY KAYS, PATANKAR-SPALDING, OTHERS
- ALSO USED TO AVOID PROBLEMS AT SEPARATION WHEN $\tau_{\text{wall}} \to 0$
PROPOSED TURBULENCE MODEL

OUTER LAYER

\[
\begin{align*}
\mu_e & = K \phi \gamma \min \left\{ \frac{F}{C_{w^+} \left( V_{max} - V_{min} \right)} \right\} \\
\gamma & = \left[ 1 + 5.5 \left( \frac{C_{Ku} \bar{y}}{y} \right)^{1/4} \right]^{-1} \\
C_{w^+} & = 0.825 \\
C_{Ku} & = 0.55
\end{align*}
\]

OUTER LAYER - FUNCTION F

- DEFINE \( F = \int f \, dy \)
- INTEGRATE BY PARTS ASSUMING \( |\omega| \to 0 \) AS \( y \to \delta \)

\[
F = \int_0^\infty y \omega \, dy
\approx \int_0^\delta y \frac{\partial u}{\partial y} \, dy
= \left. uy \right|_0^\delta - \int_0^\delta udv
= \int_0^\delta (u_e - u) \, dy
F = \delta u_e
\]

- USE F DIRECTLY IN CEBECI-SMITH OUTER FORMULATION
- ELIMINATES CONSTANT \( C_p \)
- DOES NOT REQUIRE KNOWLEDGE OF \( \delta \) OR \( u_e \)
- DISCOVERED INDEPENDENTLY BY D. A. JOHNSON, AIAA 92-0026
PROPOSED TURBULENCE MODEL

OUTER LAYER - LENGTH SCALE $\bar{y}$
- $\bar{y}$ IS THE CENTROID OF THE $f(y)$ CURVE
  \[
  \int_{0}^{\bar{y}} f(y)dy = \int_{0}^{\bar{y}} f(y)dy
  \]
- EVALUATE USING COLE'S VELOCITY PROFILES
  \[
  \begin{array}{c|c}
  \bar{y}/\delta & 0 \\
  0.5 & .5 \\
  0.55 & .55 \\
  \infty & .606
  \end{array}
  \]
- USE EQUILIBRIUM VALUE $C_{\text{rad}} = \bar{y}/\delta = 0.55$

PROPOSED TURBULENCE MODEL

OUTER LAYER - WAKE MODEL

$\mu_{w} = K_{\text{rad}} \gamma \min \{F, C_{\text{rad}} \bar{y}(|\bar{V}_{\text{max}}| - |\bar{V}_{\text{min}}|)\}$
- LOWER OPTION IS A CONVENTIONAL WAKE MODEL
- EVALUATE $C_{\text{rad}}$ BY EQUATING TWO OPTIONS, ASSUMING
  
  $g_{\text{exp}} = 0.506 \delta$
  $F_{\text{exp}} = u_{\text{e}} \delta / 2$
  $\Delta V / u_{\text{e}} \approx 1$

- GIVES $C_{\text{rad}} = 0.825$
PROPOSED TURBULENCE MODEL

3-D IMPLEMENTATION

• GRANVILLE BLENDING FUNCTION
  \[ \mu_{eff} = \mu_e \tanh \left( \frac{\mu_i}{\mu_o} \right) \]

• MODEL APPLIED INDEPENDENTLY IN BLADE-TO-BLADE (\( \eta \)) AND SPANWISE (\( \zeta \)) DIRECTIONS

• INNER LAYER - USE BULEEV LENGTH SCALE
  \[ y_i = \frac{2s_s s_c}{s_s + s_c + \sqrt{s_s^2 + s_c^2}} \]

• OUTER LAYER - USE ACTUAL DISTANCE ACROSS PROFILE
  \[ y_o = s_s \text{ OR } s_c \]

• BLEND \( \eta \) AND \( \zeta \) PROFILES VECTORALLY
  \[ \mu_{corr} = \sqrt{\mu_\eta^2 + \mu_\zeta^2} \]

COMPARISON OF FLAT PLATE VELOCITY PROFILES TO SPALDING’S COMPOSITE LAW OF THE WALL
STATIC PRESSURE DISTRIBUTION

COMPUTED & MEASURED PRESSURE DISTRIBUTIONS
FOR THE ANNULAR TURBINE CASCADE
COMPUTED & MEASURED LOSS COEFFICIENT PROFILES FOR THE ANNULAR TURBINE CASCADE

COMPUTED & MEASURED EFFICIENCY CONTOURS IN THE WAKE OF THE ANNULAR TURBINE CASCADE
BOYLE'S LINEAR CASCADE
STANTON NUMBER x 1000
$R_e = 78,000$

COMPUTED

MEASURED
BOYLE'S LINEAR CASCADE
STANTON NUMBER x 1000
$RE = 490,000$

MEASURED

COMPUTED
SUMMARY

- SPURIOUS MAXIMUM IN B-L FUNCTION \( f(y) \) CAN GIVE INCORRECT TURBULENT LENGTH SCALE & ERRATIC \( St \) OR \( C_f \) PATTERNS
  - MOST LIKELY AT LOW \( Re \) AND FAVORABLE \( \delta p/\delta s \)

- NEW TURBULENCE MODEL PROPOSED
  - INTEGRAL RELATIONS FOR \( \delta u_i \) AND \( \delta \) USED WITH C-S MODEL
  - EFFECTS OF \( \delta p/\delta s \) MODELED
  - WAKE MODEL PROPOSED

- FLAT PLATE
  - B-L & NEW MODEL AGREE WITH LAW OF THE WALL
  - LOCAL SHEAR MOD. DOES NOT AGREE WITH LAW OF THE WALL

- ANNULAR TURBINE
  - GOOD AGREEMENT WITH EXPT. PRESSURE DISTRIBUTION
  - WAKE MIXING UNDER-PREDICTED

- TURBINE ENDWALL HEAT TRANSFER
  - VARIATIONS IN ENDWALL \( St \) WITH \( Re \) PREDICTED WELL
  - EFFECTS OF \( \delta p/\delta s \) IMPORTANT

- TRANSONIC FAN
  - SHEAR LAYER FROM BOW SHOCK ACTS LIKE VISCOUS LAYER
  - NEW MODEL OVERPREDICTS L.E. \( \mu_i \)
  - B-L MODEL PREDICTS REASONABLE L.E. \( \mu_i \)
LOW EMISSIONS COMBUSTORS

J. M. DEUR
SVERDRUP TECHNOLOGY, INC.
LEWIS RESEARCH CENTER GROUP
BROOK PARK, OHIO

APPLIED ANALYTICAL COMBUSTION/EMISSIONS RESEARCH

- Analyze low emissions combustors to aid in-house experiments and contractor combustor development programs.

- Present:
  - Utilize existing codes, principally KIVA-II.
  - Improve numerics and physical models (e.g., PDF combustion-turbulence interaction) on limited basis to satisfy critical needs.

- Future:
  - Adopt 3-D ALLSPD code when available.
KIVA-II FEATURES

- **MULTI-DIMENSIONAL TIME ACCURATE FINITE DIFFERENCE CODE.**
- **COMPRESSIBLE FLOWS.**
- **k-ε TURBULENCE MODEL WITH WALL FUNCTIONS OR SUB-GRID SCALE TURBULENCE MODEL.**
- **LAMINAR KINETICS FOR ARBITRARY REACTION SET WITH QUASI-EQUILIBRIUM OPTION (MIXING CONTROLLED COMBUSTION MODEL ALSO).**
- **STOCHASTIC SPRAY MODEL WITH VAPORIZATION, AERODYNAMIC BREAKUP, TURBULENT DISPERSION, AND COLLISION SUB-MODELS.**
- **ADIABATIC OR CONSTANT TEMPERATURE WALL BOUNDARIES.**
- **ARBITRARY MESH.**
RICH BURN - QUICK MIX - LEAN BURN (ROL) FLAME TUBE

RICH BURN SECTION  MIXER SECTION  LEAN BURN SECTION

ROL FLAME TUBE MIXER CONFIGURATION PARAMETRIC STUDY

SLANTED SLOT MIXER (W/O SWIRL)  TWO HOLE MIXER (W/O SWIRL)

ONE HOLE MIXER (W/O SWIRL)  ONE HOLE MIXER (W/ SWIRL)
**ROL FLAME TUBE NOₓ EMISSION INDEX COMPARISON**

<table>
<thead>
<tr>
<th>Configuration</th>
<th>NOₓ Index</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slanted Slot Mixer (W/O Swirl)</td>
<td>1.00</td>
</tr>
<tr>
<td>Two Hole Mixer (W/O Swirl)</td>
<td>1.25</td>
</tr>
<tr>
<td>One Hole Mixer (W/O Swirl)</td>
<td>1.39</td>
</tr>
<tr>
<td>One Hole Mixer (W/ Swirl)</td>
<td>0.75</td>
</tr>
</tbody>
</table>

**NOTE:**

- Values are spatial averages taken at sampling location B.
- Values are normalized by slanted slot mixer experimental reading.

---

**ROL MIXER CONFIGURATION PARAMETRIC STUDY NOₓ COMPARISON**

<table>
<thead>
<tr>
<th>Configuration</th>
<th>NOₓ Index</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slanted Slot Mixer W/O Swirl</td>
<td>1.00</td>
</tr>
<tr>
<td>Two Hole Mixer W/O Swirl</td>
<td>1.25</td>
</tr>
<tr>
<td>One Hole Mixer W/O Swirl</td>
<td>1.39</td>
</tr>
<tr>
<td>One Hole Mixer W/ Swirl</td>
<td>0.75</td>
</tr>
<tr>
<td>Slanted Slot Mixer W/O Swirl (Experimental)</td>
<td>1.00</td>
</tr>
</tbody>
</table>

**NOTE:**

- Values are spatial averages taken at sampling location B.
- Values are normalized by slanted slot mixer experimental reading.
THE EFFECTS OF TURBULENCE MODELING ON THE NUMERICAL SIMULATION OF CONFINED SWIRLING FLOWS

G. J. MICKLOW AND M. R. HARPER
UNIVERSITY OF FLORIDA
GAINESVILLE, FLORIDA

J. M. DEUR
SVERDRUP TECHNOLOGY, INC.
BROOK PARK, OHIO

AIAA/SAE/ASME/ASEE 29TH JOINT PROPULSION CONFERENCE
MONTEREY, CALIFORNIA

\[
\frac{DK}{Dt} = \frac{\partial}{\partial x_j} \left( \frac{u_i}{\sigma_k} \frac{\partial K}{\partial x_j} \right) + \nu_l \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \epsilon
\]

\[
K = \frac{\bar{u}_i \bar{u}_i}{2}
\]

\[
\nu_l = \frac{C_s K^2}{\epsilon}
\]

\[
C_s = 0.09
\]
**K-ε TURBULENCE MODEL**

\[
\frac{DE}{Dt} = \frac{\partial}{\partial x_i} \left( u_i \frac{\partial E}{\partial x_i} \right) + C_1 u_i \frac{E}{K} \frac{\partial u_i}{\partial x_i} \left( \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} + \frac{\partial u_j}{\partial x_i} \frac{\partial u_i}{\partial x_j} \right) - C_2 \frac{E^2}{K}
\]

\[C_1 = 1.44\]

\[C_2 = 1.92\]

---

**C_μ MODIFICATIONS TO K-ε TURBULENCE MODEL**

\[C_\mu = \frac{A u_D}{K^{1/2}}\]

\[A = 0.0083\]

\[C_\mu|_{\text{wall}} = 0.09\]
\[ C_{2} \text{ MODIFICATIONS TO } K - \varepsilon \text{ TURBULENCE MODEL} \]

\[ C_{2} = C_{2}^{*} (1 - C_{3} R_{i}) \quad (0.1 < C_{2} < 2.4) \]

\[ C_{2}^{*} = 1.92 \]

\[ C_{3} = 0.2 \]

\[ R_{i} = \frac{2 v_{i}^{2} \frac{\partial (v_{i})}{\partial x}}{\left( \frac{\partial w}{\partial x} \right)^{2} + \left( x \frac{\partial (v_{i})}{\partial x} \right)^{2}} \]

\[ C_{2} |_{\text{wall}} = 1.92 \]

\[ \text{UCI AXISYMMETRIC CAN COMBUSTOR} \]
$C_{31}$ VARIATION IN UCI AXISYMMETRIC CAN COMBUSTOR

$C_{2}$ VARIATION IN UCI AXISYMMETRIC CAN COMBUSTOR
TURBULENT TIME SCALE IN UCI AXISYMMETRIC CAN COMBUSTOR

BASELINE CASE

C2 CASE

Cµ CASE

TURBULENT VISCOSITY IN UCI AXISYMMETRIC CAN COMBUSTOR

BASELINE CASE

C2 CASE

Cµ CASE
VELOCITY VECTORS IN UCI AXISYMMETRIC CAN COMBUSTOR

BASELINE CASE  C1 CASE  C2 CASE

COMPARISONS TO UCI AXISYMMETRIC CAN COMBUSTOR DATA (z=24 cm)

AXIAL VELOCITY  CIRCUMFERENTIAL VELOCITY
CONCLUSIONS

- INLET VELOCITY IS BEING MODIFIED TO MATCH FIRST INLET STATION.
- PDF COMBUSTION-TURBULENCE MODEL OF HSU, ET AL., IS BEING ADDED.
- MUCH MORE WORK IS REQUIRED.
DEVELOPMENT OF A RELIABLE ALGEBRAIC TURBULENCE MODEL GIVING ENGINEERING ACCURACY AT REASONABLE COST

by

B.P. Leonard and J.E. Drummond
The University of Akron
I. CHOICES FOR TURBULENCE MODELLING
   A. ZERO-EQUATION (ALGEBRAIC MODELS)
   B. MULTIPLE-EQUATION MODELS

II. THE NEED FOR A RELIABLE ALGEBRAIC MODEL
   A. RELATIVE SIMPLICITY
   B. ENGINEERING ACCURACY
   C. COST-EFFECTIVE APPLICATION

III. DEVELOPMENT OF THE MODIFIED MIXING LENGTH (MML) MODEL

IV. COMPARISON OF MODELS
   A. MML
   B. BALDWIN-LOMAX
   C. TWO-EQUATION (k-ε) MODEL

V. WHERE DO WE GO FROM HERE?

EFFECTIVE VISCOSITY: \( \mu_{\text{eff}} = \mu + \mu_t \) (1)

TURBULENT VISCOSITY: \( \mu_t = \rho \ell^2 |\omega| \) (2)

MIXING LENGTH:
\[
\ell = \kappa \left( \frac{C_1}{C_2} \right) y^{*} \left[ 1 - \left( 1 - \frac{y^{*}}{C_1} \right)^{C_1} \right] \left[ 1 - \exp \left( \frac{-y^{*}}{A^{*}} \right) \right] \text{ for } y^{*} < C_1
\]

\[
\ell = \kappa \left( \frac{C_1}{C_2} \right) y^{*} \text{ for } y^{*} > C_1
\]

where: \( y^{*} = y/y^{*} \); \( y^{*} = \mu/\sqrt{\rho |\tau_\omega|} \)

SHEAR STRESS "FILTER":
\[
|\bar{\tau}_j| = 0.1 |\tau_{j-2}| + 0.2 |\tau_{j-1}| + 0.4 |\tau_j| + 0.2 |\tau_{j+1}| + 0.1 |\tau_{j+2}|
\]
OUTER REGION LENGTH SCALE (CAPPING LENGTH):

\[ \ell = \ell_{\text{CAP}} = \kappa \frac{C_1}{C_2} y^*, \]

CAPPING LENGTH BASED ON BOUNDARY LAYER THICKNESS:

\[ \ell_{\text{CAP}} \equiv B \cdot \delta \]

CAPPING LENGTH BASED ON LOCAL SHEAR STRESS:

\[ \ell_{\text{CAP}} = (3.31 \cdot 10^{-6})B |C_\ell|^{-3.5} y^* \]
<table>
<thead>
<tr>
<th>LeRC CTM WORKSHOP</th>
<th>EFFECT OF VARYING C₁ IN EQUATION (3) WHILE KEEPING C₂ FIXED (=5)</th>
</tr>
</thead>
<tbody>
<tr>
<td>9/93</td>
<td></td>
</tr>
</tbody>
</table>

\[ C₁ = 500 \]
\[ C₁ = 400 \]
\[ C₁ = 300 \]

<table>
<thead>
<tr>
<th>LeRC CTM WORKSHOP</th>
<th>EFFECT OF VARYING C₂ IN EQUATION (3) WHILE KEEPING C₁/C₂ FIXED (=8)</th>
</tr>
</thead>
<tbody>
<tr>
<td>9/93</td>
<td></td>
</tr>
</tbody>
</table>

\[ C₂ = 200 \]
\[ C₂ = 400 \]
\[ C₂ = 800 \]
INNER REGION TURBULENT VISCOSITY:
\[ \mu_t = \rho \ell^2 |\omega| \]

WHERE
\[ \ell = \kappa y \left[ 1 - \exp \left( -y^*/\Lambda^* \right) \right] \]

OUTER REGION TURBULENT VISCOSITY ASSUMES THE SMALLER VALUE OF:
\[ \mu_t = \rho K C_{cp} F_k(y) y_{\text{max}} F_{\text{max}} \]

OR
\[ \mu_t = 0.25 \rho K C_{cp} U_{\text{diff}}^2 y_{\text{max}}^2 / F_{\text{max}} \]

WHERE
\[ F(y) = y |\omega| \]
\[ F_k(y) = \left[ 1 + 5.5 \left( \frac{C_h y}{y_{\text{max}}} \right)^{6.1} \right]^{-1} \]
### MML Flat Plate Skin Friction Calculations for Varying $C_{l}/\delta^{+}$

<table>
<thead>
<tr>
<th>$C_{l}/\delta^{+}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.004</td>
</tr>
<tr>
<td>0.0035</td>
</tr>
<tr>
<td>0.003</td>
</tr>
<tr>
<td>0.0025</td>
</tr>
<tr>
<td>0.002</td>
</tr>
<tr>
<td>0.0015</td>
</tr>
<tr>
<td>0.001</td>
</tr>
<tr>
<td>0.0005</td>
</tr>
</tbody>
</table>

![Graph of skin friction coefficients](image)

### MML Flat-Plate Velocity Defect Profiles for Varying $C_{l}/\delta^{+}$

![Profiles of velocity defects](image)
TURBULENT VISCOSITY PROFILES FOR VARIOUS MODELS
LENGTH SCALE FOR UNBOUNDED REGION FOR THOMAS MODEL:

\[ \ell = \frac{\ell_0 \{ \text{Max}(|u_x|) - \text{Min}(|u_x|) \}}{\omega_c} \]

TURBULENT VISCOSITY IN TRANSITION REGION BETWEEN BOUNDARY LAYER AND UNBOUNDED REGION:

\[ \mu_t = \mu_{\text{MML}} \left( C_4 - y^* \right) + \mu_{\text{Th}} \left( y^* - C_3 \right) \]

\[ \frac{\mu_{\text{MML}}}{C_4 - C_3} \]
LeRC CTM WORKSHOP 9/93 BACKWARD-FACING STEP SKIN FRICTION CALCULATIONS FOR VARIOUS MODELS 22

![Graph showing skin friction coefficients for various models.]

LeRC CTM WORKSHOP 9/93 REATTACHMENT POSITION FOR VARIOUS TURBULENCE MODELS 23

<table>
<thead>
<tr>
<th>CASE</th>
<th>REATTACHMENT POSITION (STEP HEIGHTS, $H_p$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Driver-Seegmiller Data</td>
<td>6.25</td>
</tr>
<tr>
<td>MMLT ($C_{f}/b+ = .7$, $C_3 = 10f_{CAP}, C_4 = 20f_{CAP}$)</td>
<td>7.42</td>
</tr>
<tr>
<td>Thomas</td>
<td>12.28</td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>5.41</td>
</tr>
<tr>
<td>Chien k-ε</td>
<td>6.27</td>
</tr>
<tr>
<td>LeRC CTM WORKSHOP</td>
<td>9/93 TURBULENT VISCOITY CONTOURS FOR BACKWARD-FACING STEP</td>
</tr>
<tr>
<td>-------------------</td>
<td>----------------------------------------------------------</td>
</tr>
<tr>
<td></td>
<td><img src="image1" alt="Contour Diagram" /></td>
</tr>
<tr>
<td></td>
<td><img src="image2" alt="Contour Diagram" /></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LeRC CTM WORKSHOP</th>
<th>9/93 TURBULENT VISCOITY CONTOURS FOR BACKWARD-FACING STEP</th>
<th>27</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td><img src="image3" alt="Contour Diagram" /></td>
<td></td>
</tr>
<tr>
<td></td>
<td><img src="image4" alt="Contour Diagram" /></td>
<td></td>
</tr>
</tbody>
</table>
### CONVERGENCE CHARACTERISTICS FOR VARIOUS TURBULENCE MODELS

<table>
<thead>
<tr>
<th>MODEL</th>
<th>ITERATIONS</th>
<th>CRAY Y/MP CPU TIME (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>MMLT</td>
<td>45,000</td>
<td>7100</td>
</tr>
<tr>
<td>Thomas</td>
<td>40,000</td>
<td>6000</td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>45,000</td>
<td>7000</td>
</tr>
<tr>
<td>Chien k-ε</td>
<td>110,000</td>
<td>50,000</td>
</tr>
</tbody>
</table>

### WHERE DO WE GO FROM HERE?

A. APPLICATION TO MORE COMPLEX FLOWS
   - NOZZLE FLOWS
   - RECIRCULATING INTERNAL FLOWS
   - EXTERNAL FLOWS

B. COMPARISON WITH OTHER TURBULENT CALCULATIONS

C. ESTABLISH REASONABLE ALGEBRAIC MODEL AS AN OPTION TO GIVE RESULTS WITH ENGINEERING ACCURACY FOR A VARIETY OF APPLICATIONS
Page intentionally left blank
Application of Algebraic and Two-Equation Turbulence Models to HSR Nozzle Flow Calculations

N. J. Georgiadis
and
J. R. DeBonis

Nozzle Technology Branch
Propulsion Systems Division

September 16, 1993

FULL NAVIER-STOKES ANALYSES
OF NOZZLES (WITH PARC CODE)

- OVERVIEW OF PARC CODE AND AVAILABLE TURBULENCE
  MODELS

- VALIDATION TEST CASES:
  1. AXISYMMETRIC PLUG NOZZLE
  2. EJECTOR NOZZLE

- HIGH-SPEED RESEARCH (HSR) NOZZLES WITH
  POTENTIAL FOR NOISE REDUCTION:
  1. NASA/GE 2DCD MIXER/EJECTOR NOZZLE
  2. PRATT & WHITNEY 2D MIXER/EJECTOR NOZZLE
OVERVIEW OF PARC:

- 3D AND 2D/AXISYMMETRIC VERSIONS
- NAVIER-STOKES AND EULER MODES
- CENTRAL DIFFERENCE DISCRETIZATION
- BEAM AND WARMING ALGORITHM
- SEVERAL TURBULENCE MODEL OPTIONS
- CAPABILITY TO HANDLE MULTIPLE GRID BLOCKS (NONCONTIGUOUS INTERFACING)
- GENERALIZED BOUNDARY CONDITIONS

Full Navier-Stokes Equations in PARC

\[
\frac{\partial Q}{\partial t} + \frac{\partial F_j}{\partial x_j} = \frac{1}{Re} \frac{\partial G_j}{\partial x_j}
\]

\[
Q = \begin{bmatrix} \rho \\ \rho u_i \\ E \end{bmatrix}, \quad F_j = \begin{bmatrix} \rho u_j \\ \rho u_i u_j + P \delta_{ij} \\ (E + P) u_j \end{bmatrix}, \quad G_j = \begin{bmatrix} 0 \\ \tau_{ij} \\ u_k \tau_{jk} - q_j \end{bmatrix}
\]
\[\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial u_k}{\partial x_k} \delta_{ij}\]

\[q_j = \frac{-K_e}{(\gamma - 1)Pr} \frac{\partial T}{\partial x_j}\]

\[\mu_{\text{eff}} = \mu + \mu_t\]

\[K_{c\text{-eff}} = K_c + \frac{\mu_t C_p}{Pr}\]

**Thomas Model**

\[\mu_t = \rho \ell^2 |\omega|\]

Wall bounded flows:

\[\ell = K_y \left( 1 - e^{\frac{-\gamma}{\lambda}} \right)\]

Free shear layers:

\[\ell = \ell_c \left[ \frac{\text{Max}(u_j) - \text{Min}(u_j)}{\omega_c} \right] \]
Baldwin-Lomax Model

\[ \mu_t = \begin{cases} (\mu_t)_{\text{inner}}, & y \leq y_{\text{crossover}} \\ (\mu_t)_{\text{outer}}, & y \geq y_{\text{crossover}} \end{cases} \]

\[ (\mu_t)_{\text{outer}} = \rho \left[ \kappa C_{\text{DP}} F_{\text{wake}} \right] F_{\text{kleb}} \]

\[ F_{\text{wake}} = \min \left( y_{\text{max}} F_{\text{max}}, C_{\text{w}} y_{\text{max}} u_{\text{dif}}^2 / F_{\text{max}} \right) \]

\[ F(y) = y \left[ \alpha \left( 1 - e^{\frac{k}{\sigma \kappa}} \right) \right] \quad F_{\text{kleb}}(y) = \left[ 1 + 5.5 \left( \frac{C_{\text{kleb}} y}{y_{\text{max}}} \right)^{0.7} \right]^{-1} \]

Chien k-\( \varepsilon \)  Model

\[ \mu_t = C_{\mu} f_{\mu} \rho k^2 / \varepsilon \]

\[ \frac{D(\rho k)}{D_t} = \frac{\partial}{\partial x_1} \left[ \left( \mu + \frac{\mu_h}{\sigma_k} \right) \frac{\partial k}{\partial x_1} \right] + \Pi - \rho \varepsilon - D \]

\[ \frac{D(\rho \varepsilon)}{D_t} = \frac{\partial}{\partial x_1} \left[ \left( \mu + \frac{\mu_h}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_1} \right] + C_{\varepsilon f_1} \Pi \frac{\varepsilon}{k} - C_{\varepsilon f_2} \frac{\varepsilon^2}{k} - 2\mu \frac{\varepsilon}{y^2} e^{\left( -0.5 \varepsilon \right)} \]
\[ \Pi = \mu \frac{\partial u_j}{\partial x_i} \left[ \frac{\partial u_l}{\partial x_i} + \frac{\partial u_k}{\partial x_j} \right] \]

\[ D = 2\mu k / y^2 \]

\[ f_\mu = 1 - e^{-0.0115y^*} \]

\[ f_1 = 1.0 \]

\[ f_2 = 1.0 - 0.22 e^{-\left(\frac{R_y}{\delta}\right)^2} \]

\[ \text{Re}_t = \frac{\rho k^2}{\mu\epsilon} \]

**TWO-EQUATION MODEL OF SPEZIALE**

- SOLVED AS A k-\(\epsilon\) MODEL IN PARC
- ORIGINALLY DEVELOPED AS A k-\(\tau\) MODEL IN NASA CR-182068
- UPWIND DIFFERENCE DISCRETIZATION (MAIN FLOW EQUATIONS IN PARC USE CENTRAL DIFFERENCING)
BENCHMARK VALIDATION CASES

1. Mach 0.2 flat plate
2. Mach 0.128 backward-facing step

SHEAR STRESS PREDICTIONS WITH THE DIFFERENT TURBULENCE MODELS IN PARC
SHEAR STRESS AFTER THE STEP WITH THE DIFFERENT TURBULENCE MODELS IN PARC

LANGLEY SINGLE FLOW PLUG NOZZLE

- VENTED AND NON-VENTED PLUGS
- 15° PLUG HALF ANGLE
- HEAVILY INSTRUMENTED TO MEASURE:
  1. PLUG SURFACE TEMPERATURES, PRESSURES, SHEAR STRESS
  2. JET PLUME QUANTITIES (INCLUDING LDV & FLOW VISUALIZATION)
  3. FLOWFIELD ACOUSTICS
GEOMETRY FOR GRID GENERATION AND PARC2D CALCULATIONS

DESIGN CONDITIONS:

- JET MACH NO. (JET) = 1.50
- NPR = 3.67
- T_0 (PRIMARY) = 2060°R

VELOCITY PROFILES FOR FINE GRID SOLUTIONS

- X = 25 in.
- X = 50 in.
- X = 75 in.
TOTAL TEMPERATURE PROFILES
FOR FINE GRID SOLUTIONS

SHOCK FUNCTION
(BASED ON PRESSURE GRADIENT)
STATIC PRESSURE DISTRIBUTION ALONG PLUG

2D EJECTOR NOZZLE TEST CASE

- REPRESENTATIVE OF MIXER-EJECTOR NOZZLES THAT ARE BEING CONSIDERED FOR SUPersonic TRANSPORT APPLICATION

- FLOW DOMINATED BY TURBULENT MIXING OF A HIGH ENERGY STREAM WITH SECONDARY AIR
2D EJECTOR NOZZLE OF GILBERT AND HILL

VELOCITIES IN MIXING SECTION

X = 7 in.

X = 10.5 in.
TOTAL TEMPERATURES
IN MIXING SECTION

MIXING EFFECTIVENESS
TURBULENT VISCOSITY CONTOURS
FOR 2D EJECTOR NOZZLE

\[ \mu_1 / \mu \]

Thomas

Speziale \( k-\varepsilon \)

Chien \( k-\varepsilon \)

AXISYMMETRIC INVERTED VELOCITY
PROFILE NOZZLE OF VON GLAHN,
GOODYKOONTZ, AND WASSERBAUER
Mixer/Ejector Nozzles

- Entrain large amounts of secondary flow
- Rapidly mix two flows together to lower jet velocity
- Lower jet velocity results in lower noise
- Maintain high thrust due to large mass augmentation

\[ F = \dot{m}v \]
Typical Mixer/Ejector Nozzle Chute Geometry

Primary Flow
Secondary Flow

NASA/GE 2DCD Mixer/Ejector Nozzle
Shroud Static Pressures
MAR = 1.20
Thomas Turbulence Model

Mach Number
\( A_0/A_{\text{mix}}^{1.2} \)

Primary Centerline

Secondary Centerline

Shroud Static Pressures
MAR = 1.20
Thomas Turbulence Model

\( P/P_{\text{amb}} \)

PARC Primary
PARC Secondary
Data Primary
Data Secondary

\( X/L \)
Shroud Static Pressures
MAR = 1.20
K-e Turbulence Model

Total Temperature in Mixing Section
$A_e / A_{mix} = 1.2$

X/L = 0.0
X/L = 0.333
X/L = 0.667
X/L = 1.0

337
Anatomy of the HSR P&W 2-D Mixer Ejector Nozzle

Inventory:

- 2 Mixer Designs
- 3 Shroud Lengths
- 3 Shroud/Sidewall Acoustic Treatments
- Sidewalls with Windows for Flow Visualization
3D COMPUTATIONAL GRID FOR NOZZLE FLOWFIELD

P&W MIXER/EJECTOR NOZZLE
Mach Number Contours

Peak Side

Valley Side
Conclusions

- Algebraic models are inadequate for predicting mixer nozzle flowfields
- Two-equation model available in PARC3D give no improvement over algebraic models
- A reliable two-equation model is required in PARC3D for complex nozzle flows
Page intentionally left blank
INTRODUCTION

The Icing and Cryogenic Technology Branch develops computational tools which predict ice growth on aircraft surfaces and uses existing CFD technology to evaluate the aerodynamic changes associated with such accretions.

Surface roughness, transition location, and laminar, transition, or turbulent convective heat transfer all influence the ice growth process on aircraft surfaces.

Turbulence modeling is a critical element within the computational tools used both for ice shape prediction and for performance degradation evaluation.
CURRENT CODE DEVELOPMENT

2D CODES

• LEWICE - POTENTIAL FLOW / INTEGRAL BOUNDARY LAYER
• LEWICE/IBL - POTENTIAL FLOW / INTERACTIVE BOUNDARY LAYER

3D CODES

• LEWICE3D - PANEL CODE / INTEGRAL BOUNDARY LAYER
• LEWICE3DGR - ANY GRID BASED FLOW SOLUTION

ICE ACCRETION MODELING

CURRENT MODEL USED FOR ICE GROWTH

• MASS AND ENERGY BALANCE IN CONTROL VOLUMES ALONG THE SURFACE

• CONVECTIVE HEAT TRANSFER IS MAJOR FACTOR IN ENERGY BALANCE

• INTEGRAL BOUNDARY LAYER FORMULATION USED TO DETERMINE LAMINAR AND TURBULENT HEAT TRANSFER COEFFICIENTS

• SURFACE ROUGHNESS MODELED AS SAND-GRAIN ROUGHNESS; ACTUAL ICE ROUGHNESS VARIES FROM SMALLER TO LARGER THAN BOUNDARY LAYER THICKNESS
ICE ACCRETION MODELING

CONVECTIVE HEAT TRANSFER MODEL USED FOR ICE GROWTH

SKIN FRICTION COEFFICIENT

\[ \frac{c_f}{2} = 0.1681 \left[ \ln \left( \frac{864.0 \theta_t}{k_s} + 2.568 \right) \right]^{-2} \]

WHERE

\[ \theta_t(s) = \left[ \frac{0.0156}{V_e^{4.11}} \int_s^t V_e^{3.86} ds \right]^{0.8} + \theta_t(s_tr) \]

ICE ACCRETION MODELING

CONVECTIVE HEAT TRANSFER MODEL USED FOR ICE GROWTH

LAMINAR

\[ h_t(s) = 0.296 \frac{\lambda}{V_e} \left[ V_e^{-2.88} \int_0^s V_e^{1.88} ds \right]^{-1/2} \]

TURBULENT

\[ h_t(s) = SpV_e c_p \left[ \frac{c_f/2}{Pr_t + \frac{c_f/2}{(1/St_k)}} \right] \rho V_e c_p \]
ICE ACCRETION MODELING

CONVECTIVE HEAT TRANSFER MODEL USED FOR ICE GROWTH

ROUGHNESS STANTON NUMBER

\[ St_k = 1.16 \left( \frac{V_r k_s}{V} \right)^{-0.2} \]

AND

\[ V_r = V_e \sqrt{\frac{c_f}{2}} \]

ICE ACCRETION MODELING

ICE ROUGHNESS CHARACTERIZATION

SAND-GRAIN ROUGHNESS

ACTUAL ICE ROUGHNESS
ICE ACCRETION MODELING

PLANS

• EXPERIMENTS TO CHARACTERIZE ICE ROUGHNESS GEOMETRIES AT A VARIETY OF ICING CONDITIONS

• EXPERIMENTS TO CHARACTERIZE VELOCITY FIELD OVER REAL AND ARTIFICIAL ICE ROUGHNESS GEOMETRIES

• EXPERIMENTS TO MEASURE HEAT TRANSFER OVER REAL AND ARTIFICIAL ICE ROUGHNESS GEOMETRIES

• DEVELOPMENT OF MODIFIED COMPUTATIONAL MODEL BASED ON THESE EXPERIMENTS

ICED AIRFOIL AERODYNAMICS

NACA 0012 ICING CONDITIONS

$\alpha = 4^\circ$  $V = 130$ mph
$d = 20\, \mu m$ $LWC = 2.1\, g/m^3$
$T = 18^\circ F$

SHEAR LAYER

RECIRCULATION REGION

5 MINUTE ICE GROWTH
ICED AIRFOIL AERODYNAMICS
BALDWIN-LOMAX TURBULENCE MODEL

**Inner Layer**
\[ \mu_t - l^2 |u_y - v_x| \]
\[ l = \text{mixing length} \]

**Outer Layer**
\[ \mu_t \sim F_{max} Y_{max} \]
\[ F(y) = y|\alpha| (1 - \exp((-y^*)/A)) \]

ICED AIRFOIL AERODYNAMICS
BALDWIN-LOMAX TURBULENCE MODEL

NORMAL B.L. F PROFILE  
RECIRCULATION REGION  
F PROFILE

![Graphs showing normal boundary layer and recirculation region profiles](image)
\[ \mu_t = \rho l^2 \omega \]

WHERE

\[ \frac{y + \Delta y}{y} < C_1 \quad l(y) = \kappa \frac{C_1}{C_2} y^* \left( 1 - \left( 1 - \frac{y + \Delta y}{y} \right)^{\frac{C_2}{C_1}} \right) \left( 1 - e^{-\left(\frac{y + \Delta y}{y} / A^*\right)} \right) \]

AND WHERE

\[ \frac{y + \Delta y}{y^*} > C_1 \quad l(y) = \kappa \frac{C_1}{C_2} y^* \]

\[ y^* = \frac{v}{u^*} = \frac{v}{\sqrt{\tau_w / \rho_w}} \]
THE CEBECI-CHANG ROUGHNESS MODEL IS ADDED TO THE TURBULENCE MODEL

\[
\Delta y^+ = \begin{cases} 
0.9 \left[ k_s^+ - k_s^+ \exp \left( \frac{-k_s^+}{6} \right) \right] & 5 < k_s^+ \leq 70 \\
0.7 (k_s^+)^{0.58} & 70 \leq k_s^+ \leq 2000 
\end{cases}
\]

WHERE,

\[
\Delta y^+ = (\Delta y) (u_\tau/\nu) \quad \text{and} \quad k_s^+ = k_s (u_\tau/\nu)
\]
ICED AIRFOIL AERODYNAMICS
MML TURBULENCE MODEL

$C_D$ vs. $\alpha$

- $\circ$ Bragg, [96]
- $\triangle$ ARCID, MML model
- $\square$ ARCID, B-L model

ICED AIRFOIL AERODYNAMICS
MML TURBULENCE MODEL

STRUCTURED GRID FOR ARTIFICIAL ICE SHAPE
ICED AIRFOIL AERODYNAMICS
MML TURBULENCE MODEL

UNSTRUCTURED GRID FOR ARTIFICIAL ICE SHAPE

ICED AIRFOIL AERODYNAMICS
MML TURBULENCE MODEL

STRUCTURED GRID MACH NUMBER CONTOURS
CONCLUDING REMARKS

- Turbulence modeling plays a role in ice growth prediction and in performance evaluation.
- New modeling is required for the large roughness elements of a typical ice accretion.
- An experimental program is currently underway to develop a database for creation of such a model.
- An alternate algebraic turbulence model has been used to evaluate performance degradation due to icing.
- The MML model has been used in an unstructured grid Navier-Stokes code to calculate flow over an artificial ice shape.
Page intentionally left blank
Applied RNG Algebraic Turbulence Model for Three-Dimensional Turbomachinery Flows

K. R. Kirtley

Sverdrup Technology, Inc. LeRC Group
and
Cambridge Hydrodynamics, Inc.
What is RNG?

RNG

1) Decompose velocity into low and high wave number components
2) Use perturbation theory to eliminate high wave number bands then renormalize spectrum and repeat to infrared cutoff
3) Correlations disappear through mode elimination procedure
4) Closure is automatic
5) Evaluate coefficients from high Re limit (fixed point) of perturbation expansion

Reynold’s Averaging

1) Decompose velocity into mean and fluctuating components
2) Ensemble (time) average over entire spectrum
3) Correlations arise
4) Model correlations (Boussinesq)
5) Evaluate coefficients from generic flow data, e.g., wake, jet, etc.

Self Consistent

Problem Dependent

Result — effective viscosity

RNG-Based Algebraic Model

\[ \nu = \nu_0 \left[ 1 + H \left( \frac{a}{\nu_0^3} \epsilon A_f^{-4} - C_c \right) \right]^{\frac{1}{3}} \]

\[ \epsilon = P = \nu S \]

where

\[ S = \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \frac{\partial u_i}{\partial x_j} \]

\( A_f = \) Infrared cutoff = top of the Inertial range resolvable < \( A_f < \) modelled

\( a = .1186 \) from -5/3 spectral law

\( C_c = 200 \) from \( a \) and the ultraviolet dissipation range cutoff
How do you go from Fourier space to physical space?

why $2\pi$ of course.

Integral scale,

$$L_f^i = \frac{2\pi}{\Lambda_f}$$

But we need a mixing length, therefore

Assume $E = C_K \epsilon^{\frac{2}{3}} \Lambda^{-\frac{2}{3}}$

$$C_K = 1.6075$$

Integration from $L_f$ to infinity gives

$$k = \frac{3}{2} C_K \Lambda_f^{-\frac{2}{3}} \epsilon^{\frac{2}{3}}$$

Assume in high Re limit

$$\nu = L_f^2 S^{\frac{1}{2}} = C_K \frac{k^2}{\epsilon}$$

Then

$$L_f = \left( \frac{9}{4} C_K \epsilon^{\frac{2}{3}} \Lambda_f^{-1} \right) = \frac{a^4}{2\pi} L_f^i$$

Giving

$$\nu = \nu_0 \left[ 1 + H \left( \frac{\epsilon}{\nu_0^3} L_f^4 - C_c \right) \right]^{\frac{1}{3}}.$$

Attributes of the RNG-Based Model

1) The Heaviside function mimics:
   a) Near wall damping
      $$\nu = \nu_0$$ for all $y' \leq \frac{C_c^4}{\kappa} = 9.2$
   b) Intermittency
      $$\nu = \nu_0$$ when $\epsilon \leq \frac{C_c \nu_0^3}{L_f^4}$
      $\epsilon \to 0$ in the outer flow
   c) Transition

      See Next Slide

2) Energy-Based - Non-equilibrium effects can be included through clever manipulation of the dissipation rate
A neat little analysis (following a slight detour)
Turbulence Length Scales

In boundary layers

\[ L_f = C_\mu \delta \tanh \left( \frac{\kappa n}{C_\mu \delta} \right) \]

\( \kappa = 0.272 \) and \( C_\mu = 0.0845 \) from RNG

In wake region use Raj & Lakshminarayana correlation for cascade wakes

\[ L_f = \min (\kappa s, C_w b) \]

\[ b = \delta_0 + c C_d^{1/2} 1.35 \left( \frac{s}{c} + 0.02 \right)^{0.58} \]

\( C_d \) = coefficient of drag (assume 0.015)
\( c \) = local chord
\( \delta_0 \) = ave. of s.s and p.s. trailing edge boundary layer thickness
\( C_w = 0.169 \) from wake behind a circular cylinder
Flow Development in Rotor Passage

Flow Development in Rotor Passage, Contours of $U_{\text{total}}/U_{\text{tip}}$

PSU Low Speed Compressor

Streamwise Velocity

Transverse Velocity

360
Figure 1
Total Pressure Loss Coefficient

Measured (Langston et al. 76-GT-50)

Comcomputed w/ Alg. RG Mixing Length

mid-span

hub

mid-span

hub

mid-span

hub

363
Hub Surface Static Pressure

Measured (Langston et al. 76-GT-50) Computed w/ Algebraic RG Mixing Length

![Diagram of Hub Surface Static Pressure]
Conclusions

- RNG based algebraic model is a viable alternative to traditional algebraic models for complex flows.

- RNG model mimics transitions and near-wall damping through the Heaviside function.

- Good boundary layer profiles, loss profiles, pressure loadings can be achieved for viscous dominated flow.

- Stanton number predictions are not yet adequate when using the "plain vanilla" form of the model.

- Work is needed to include non-equilibrium effects.
Page intentionally left blank
Applied k–ε and Baldwin–Lomax Turbulence models for S–ducts

G.J. Harloff, Sverdrup Technology, Inc.
LeRC Group, Brook Park, Ohio

September 16, 1993

Experimental contributions to paper by

B.A. Reichert, NASA Lewis Research Center
Cleveland, Ohio
S.R. Wellborn, Iowa State University
Ames, Iowa

Objectives

Provide CFD validation data with secondary flows
Predict and compare with experimental flow fields
Assess algebraic turbulence model
Compare algebraic and K - ε turbulence model results
Introduction

Diffusing S-ducts common on aircraft: 727, L-1011, F-16, F-18
S-duct fabricated and tested

CFD: 3-D full Navier-Stokes, two turbulence models
Research extends previous PNS and FNS studies
Overall features qualitatively correct: total pressure velocity vectors, exit vortices
Boundary layer separation details generally not correct
Turbulence models and/or grid resolution usually cited for lack of agreement

Code Selected: PARC3D

Originally ARC3D - Pulliam and Steger
Modified for internal flows - Cooper

Full 3-D Navier-Stokes equations in Reynolds (mass) average form

Beam-Warming approximate factorization
Multi-block for computer efficiency

Low Reynolds number $K - \varepsilon$ turbulence model of Speziale employed - Nichols
Algebraic Baldwin-Lomax turbulence model
Boundary Conditions

No slip at wall

Total pressure and temperature specified at inlet

Static pressure specified at exit

Symmetry about x - z plane

$k = 0$ on wall

\[ \varepsilon = \frac{2u}{\rho} \left[ \left. \frac{\partial \sqrt{k}}{\partial y} \right|_{\text{wall}} \right]^2 \]

$k, \varepsilon$: zeroth-order extrapolation at inlet, outlet, and centerline

Nomenclature

\[ C_{p0} = \frac{p_0 - p_{\text{wall}}}{p_{0_{\text{cl}}} - p_{\text{wall}}} \]

\[ C_p = \frac{p - p_{\text{wall}}}{p_{0_{\text{cl}}} - p_{\text{wall}}} \]

\[ u^+ = \frac{u}{u^*} \]

\[ y^+ = \frac{u^* y}{v} \]

\[ u^* = \sqrt{\frac{t_w}{\rho_w}} \]

\[ c_f = \frac{t_w}{2 \rho \frac{U^2}{\text{cl}}} \]
Duct Test Conditions and Geometry

\[ \text{Re} = 2.6 \times 10^6 \]

\[ M_1 = 0.6 \]

\[ D_1, D_2 = 8.04, 9.90 \text{ in} \]

\[ \frac{A_2}{A_1} = 1.52 \]

\[ R = 40.2 \text{ in} \]

\[ \theta_{\text{max}} / 2 = 30^\circ \]

measurement planes \( S/D_1 = -0.5, 5.73 \)

upstream and downstream pipes =3.75D_1

Grid for S-duct

O Grid in 3 blocks
- 32 x 71 x 53
- 69 x 71 x 53
- 32 x 71 x 53

H Grid in center
- 129 x 11 x 15
The geometry of the diffusing S-duct

Total pressure contours at $s/D_1 = -0.5$
Transverse velocity components at $s/D_1 = 5.73$

Axial Mach number contours at $s/D_1 = 5.73$
Boundary layer wall coordinate plots
at $s/D_1 = -0.5$

Boundary layer wall coordinate plots
at $s/D_1 = 5.73$
Axial skin friction coefficient

Streamlines near the S-duct surface
Conclusions

Computed flow fields agree reasonably well with experimental flow field

K-\(\varepsilon\) turbulence model better predicts pressure field than algebraic model

Both models underpredict length, angular extent and axial location of boundary layer separation

Possible causes include: inappropriate artificial/computed viscosity, especially in separated region

Improvements are needed to account for strong secondary flows with separation
Proteus Experience with the Modified MML Turbulence Model

Julianne Conley
NASA Lewis Research Center

The Workshop on Computational Turbulence Modeling
NASA Lewis Research Center
September 15-16, 1993
Overview

- Background
- Modification of MML Turbulence Model
- Test Cases
- Concluding Remarks

Background

- Based on original MML model of Potapczuk (1989)
- Simple model for flows where Baldwin Lomax model falls short
- Based on Prandtl's mixing length theory:

\[ \mu_i = \rho \rho^2 |\omega| \]
• MML model:

\[ l = \kappa \frac{C_1}{C_2} y^* \left(1 - \left(1 - \frac{y^+}{C_1}\right)\left(1 - e^{-\left(\frac{y^+}{A^*}\right)}\right)\right), \quad y^* < C_1 \]

\[ l = \kappa \frac{C_1}{C_2} y^*, \quad y^* > C_1 \]

\[ y^* = \frac{\mu}{\rho |u'|} \]

\[ y^+ = \frac{y}{y^*} \]

Modification of the MML Turbulence Model

**Step 1:** Get good estimate of \( \tau_w \)

**Step 2:** Evaluate and modify MML for zero pressure gradient boundary layer flows

**Step 3:** Modify model for an adverse pressure gradient flow

**Step 4:** Combine all features into one general model (MMLPG)
Step 1: Get good estimate of $\tau_w$

$$\tau_w = \mu \frac{\partial u}{\partial y}$$

Global approach -- use momentum equation and 2 interior grid points

$$\frac{\partial \tau}{\partial y} = \frac{\partial p}{\partial x}$$

$$T = ay^2 + by + c$$

$$a = \frac{\tau_1 - \tau_2 - b(y_1 - y_2)}{y_1^2 - y_2^2}, \quad b = \frac{\partial p}{\partial x}$$

$$c = \frac{1}{2} \left( \tau_1 + \tau_2 - a(y_1^2 + y_2^2) - b(y_1 + y_2) \right)$$

$$\Rightarrow \tau_w = c$$

$$\tau = \mu_{total} \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)$$

Use local average of $\tau_w$ to avoid large values of $y^*$ near separated regions:

$$|\tau^*_d| = 0.1|\tau_{w-1}| + 0.2|\tau_{w-1}| + 0.4|\tau_w| + 0.2|\tau_{w+1}| + 0.1|\tau_{w+1}|$$
**Step 2:** Evaluate and modify MML for zero pressure gradient boundary layer flows

Outer length scale: \[ l = \kappa C_1 y^*, \quad y^+ > C_1 \]

Doesn't allow boundary layer thickness to grow properly.

**Velocity Defect Profiles for Flow over a Flat Plate**

![Velocity Defect Profiles](image)

**Turbulent Viscosity for Flow over a Flat Plate**

![Turbulent Viscosity](image)
Found optimum $C_1$ at each $Re_x$:

$$l_{inner}^+ = \kappa y^+ \left( 1 - e^{-\frac{y^+}{\Lambda_k^+}} \right), \quad y^+ < C_1$$

$$l_{cap}^+ = \kappa C_1 \left( 1 - e^{-\frac{C_1}{\Lambda_k^+}} \right), \quad y^+ > C_1$$

Found $l_{cap}^+ = fcn (c_f)$

$$l_{cap}^+ = 1860 - (6.20 \times 10^5) c_f$$

$$l^+ = \min (l_{inner}^+, l_{cap}^+)$$

---

**Velocity Defect for Flow over a Flat Plate**

![Graph showing velocity defect for different Reynolds numbers and experimental data points.](image)

382
Step 3: Modifications for adverse pressure gradient flow

**Benchmark test cases:** Equilibrium boundary layer flows
(P. Bradshaw, 1966)

\[ U_\infty \propto x^a, \quad \beta = \frac{\delta_1 \partial p}{\tau_w \partial x} = constant \]

\[ a = 0, -0.15, -0.255 \quad \beta = 0, 1, 5 \]

**Note:**
- \( \frac{l_{cap}}{\delta} = 0.08 \)
- \( C_t \frac{\delta}{\delta} = 0.4 \)
- Slope increases with increasing pressure gradient
Combining with Blending Function:

\[ l^+ = \kappa C_1 \frac{C_1}{C_2} \left( 1 - \frac{y^+}{C_1} \right) \left( 1 - e^{-\frac{y^+}{C_1}} \right), \quad y^+ < C_1 \]

\[ l^+ = \kappa C_3 C_2 y^+, \quad y^+ > C_1 \]

\[ C_1 = 0.4 C_4, \quad C_2 = 5 C_3 \kappa \]

\[ C_3 = fcn(\beta), \quad C_4 = fcn(\beta, c_f) \]

\[ C_6 = C_5 + C_6 c_f \]

<table>
<thead>
<tr>
<th>Strength of pressure gradient</th>
<th>$\beta$</th>
<th>$C_3$</th>
<th>$C_5$</th>
<th>$C_6$</th>
</tr>
</thead>
<tbody>
<tr>
<td>zero</td>
<td>0</td>
<td>1.00</td>
<td>23,300</td>
<td>-7.75x10^5</td>
</tr>
<tr>
<td>mild</td>
<td>1</td>
<td>1.25</td>
<td>30,100</td>
<td>-1.16x10^7</td>
</tr>
<tr>
<td>strong</td>
<td>5</td>
<td>1.53</td>
<td>33,800</td>
<td>-2.09x10^7</td>
</tr>
</tbody>
</table>

Velocity Defect Profiles – Adverse Pressure Gradient Flow

Zero

Mild

Strong

- $Re = 10^6$
- $Re = 1.2 m$
- $Re = 1.7 m$
- $Re = 2.1 m$
- Bradshaw Data

- $Re = 10 x 10^6$
- $Re = 1.2 m$
- $Re = 1.7 m$
- $Re = 2.1 m$
- Bradshaw Data

- $Re = 12 x 10^6$
- $Re = 1.2 m$
- $Re = 1.7 m$
- $Re = 2.1 m$
- Bradshaw Data

- Schütz-Grzegor

- Bradshaw Data

384
Step 4: Combine to get one general model, MMLPG

\[
\begin{align*}
C_3 &= 1.0 \\
C_5 &= 23,300. \\
C_6 &= -7.75 \times 10^6 \\
\end{align*}
\]

\[
\begin{align*}
C_3 &= 1.0 + 0.307 \beta - 0.0391 \beta^2 \\
C_5 &= 23,200 + 8560 \beta - 1230 \beta^2 \\
C_6 &= -7.75 \times 10^6 - 4.51 \times 10^6 \beta + 386,000 \beta^2 \\
\end{align*}
\]

\[
\begin{align*}
C_3 &= 1.52 \\
C_5 &= 33,900 \\
C_6 &= -20,900 \\
\end{align*}
\]

\[\beta < 0.0\]

\[0.0 \leq \beta \leq 5.34\]

\[\beta > 5.34\]
Static Pressure Distribution -- Weak Shock Case, R=0.82

Test Cases

Sajben Transonic Diffuser Flows

Computational Grid

81x51
Concluding Remarks

- Developed MMLPG for adverse pressure gradient flows
- Shown that MMLPG successfully computes boundary layer flows and transonic diffuser flows
- Future work: continue validation modifications for separated flow
Turbulence Model Experiences for a Round-to-Rectangular Transition Duct

Workshop on Computational Turbulence Modeling
September 15–16, 1993

Charles E. Towne
NASA Lewis Research Center
Cleveland, OH

Outline

• Proteus code description
• Geometry and flow conditions
• Numerical details
• Turbulence model modifications
• Convergence history
• Computed results and comparison with experimental data
• Concluding remarks
Proteus Navier-Stokes Code

- Reynolds-averaged, unsteady, compressible Navier-Stokes equations
- Strong conservation-law form
- Fully-coupled ADI solution procedure, Beam-Warming generalized time differencing
- Second-order central differencing for spatial derivatives
- Implicit steady/unsteady boundary conditions
- Convection and diffusion terms linearized using second-order Taylor series expansion
- Generalized nonorthogonal body-fitted coordinates
- Baldwin-Lomax and Chien $k-\varepsilon$ turbulence models

- 2-D, axisymmetric w/wo swirl, or 3-D flow
- Thin-layer and Euler options
- Wide variety of boundary conditions
- Constant stagnation enthalpy option
- Constant-coefficient or adaptive artificial viscosity
- First- or second-order time differencing
- Variety of time step selection methods
- Output files for CONTOUR and PLOT3D
- Highly vectorized for Cray computers
- Extensively commented
- Three-volume documentation set
Circular-to-Rectangular Transition Duct

Flow Conditions

- $M_{ref} = \bar{M}_{inlet} = 0.2$ to simulate incompressible experiment
- $Re_{ref} = (\rho u R / \mu)_{inlet} = 195,000$
- Constant $c_p$ and $c_v$, with $\gamma = 1.4$
- Constant $\mu_I$ at 519 °R
- Constant stagnation enthalpy
Total Pressure Contours

Boundary and Initial Conditions

- Boundary conditions

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Conditions Specified</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>( p_T = (p_T)_{exp}, u_x = 0, v = w = 0 )</td>
</tr>
<tr>
<td>Exit</td>
<td>( m = m_{exp}, u_x = v_x = w_x = 0 )</td>
</tr>
<tr>
<td>Outer wall</td>
<td>( p_r = 0, u = v = w = 0 )</td>
</tr>
<tr>
<td>Centerline</td>
<td>( (p, u, v, w)<em>{CL} = ave (p, u, v, w)</em>{CL+1} )</td>
</tr>
<tr>
<td>( \theta = 0 )</td>
<td>( p_r = u_r = w_r = 0, v = 0 )</td>
</tr>
<tr>
<td>( \theta = 90 )</td>
<td>( p_r = u_r = v_r = 0, w = 0 )</td>
</tr>
</tbody>
</table>

- Initial conditions

\( p = p_{\infty}, u = (u_{\exp})_{\infty}, v = w = 0 \)
Numerics

- $61(x) \times 31(\theta) \times 50(r)$ and $61 \times 31 \times 99$ meshes
- Local time step, with:

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Time Level</th>
<th>CFL</th>
</tr>
</thead>
<tbody>
<tr>
<td>$61 \times 31 \times 50$</td>
<td>1 - 1001</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>1001 - 2001</td>
<td>10</td>
</tr>
<tr>
<td></td>
<td>2001 - 3001</td>
<td>10</td>
</tr>
<tr>
<td></td>
<td>3001 - 4001</td>
<td>20</td>
</tr>
<tr>
<td>$61 \times 31 \times 99$</td>
<td>1 - 1501</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>1501 - 3001</td>
<td>10</td>
</tr>
<tr>
<td></td>
<td>3001 - 4501</td>
<td>10</td>
</tr>
<tr>
<td></td>
<td>4501 - 6001</td>
<td>15</td>
</tr>
<tr>
<td></td>
<td>6001 - 7501</td>
<td>15</td>
</tr>
</tbody>
</table>

- Cray Y-MP CPU time — just under $5 \times 10^{-5}$ sec/grid point/time step, or 5.2 and 18.8 hours for the two cases

Computational Mesh
Baldwin-Lomax Turbulence Model

Inner region
• $\mu_i = \rho l^2 \Delta \omega / R e_{ref}$

Outer region
• $\mu_i = K C_{cp} p F_{Kleb} F_{wake} R e_{ref}$

where $F_{wake} = (y_n)_{max} F_{max}, F_{max} = \max \{y_n \Omega(1 - e^{-y^2/A^2})\}$
and $(y_n)_{max}$ is the corresponding $y_n$

Model based on boundary layer flows, without accounting for secondary flows and streamwise vorticity

At each streamwise station, the search region for $F_{max}$ was limited to the value of $(y_n)_{max}$ in the $y = 0$ plane

Rationale — to prevent the secondary vortices, at the "ends" of the cross-section, from affecting the $\mu_i$ values

Effect of Turbulence Model Modification

![Diagram showing effect of turbulence model modification](image)
Effect of Turbulence Model Modification

![Diagram showing the effect of turbulence model modification with two models: Modified $\mu_t$ model and Original $\mu_t$ model. The graph illustrates the differences in turbulent viscosity and flow characteristics at different stations.]

395
Effect of Turbulence Model Modification

Mass Flow Rate Convergence
Convergence History

Peripheral Static Pressure Distribution
TRANSITION DUCT FLOW

Centerline Static Pressure Distribution

![Graph showing centerline static pressure distribution with data points and lines indicating analysis and experiment results.]

Secondary Velocity Vectors

![Images showing secondary velocity vectors at Station 3 and Station 4 with analysis and experiment data superimposed.]

398
Secondary Velocity Vectors

Total Pressure Contours
Total Pressure Contours

Station 5
Analysis
Experiment

Station 6
Analysis
Experiment

Total Pressure Profiles Along Semi-Minor Axis

Station 1
4
5
6

Total pressure coefficient, \( \frac{p_T - p_0}{\rho_0 U^2} \)
Total Pressure Profiles Along Semi-Major Axis

Concluding Remarks

- The Proteus 3-D Navier-Stokes code has been used to compute the flow through a round-to-rectangular transition duct.
- The search region for $F_{\text{max}}$ in the Baldwin-Lomax turbulence model was limited to prevent excessively large $\mu_i$ values.
- The code correctly captures the basic physics — the generation of secondary flows and the resulting distortion.
- Agreement between computed and experimental results is good through the transition section.
- The computed secondary flows are overly damped in the downstream straight section.
- Further work is needed to investigate the effects of mesh resolution and turbulence model.
Proteus experience with three different types of turbulence models

Trong T. Bui
NASA Lewis Research Center
Cleveland, Ohio

Contents

1. Overview of turbulence models used
2. Practical issues
3. Suggestions
Overview of turbulence models used

1. Algebraic model
   - Baldwin-Lomax (BL)

2. One-equation model
   - Baldwin-Barth one-equation (BB)

3. Two-equation models
   - Chien k-ԑ (CH)
   - Launder-Sharma k-ԑ (LS)

Contents

1. Overview of turbulence models used

2. Practical issues
   - CPU time requirement
   - Initialization of multi-equation models
   - y* dependency
   - Compressibility corrections
   - Where the turbulence models work well, and where they do not work too well

3. Comments
**CPU time requirement**

Not a major issue in using multi-equation turbulence models:

(CPU time in sec/iter/grid point)

<table>
<thead>
<tr>
<th>Turbulence models</th>
<th>Incomp. flat plate</th>
<th>Comp. flat plate</th>
<th>S-duct</th>
<th>3-D shock / B.L. inter.</th>
</tr>
</thead>
<tbody>
<tr>
<td>B.- B.</td>
<td>6.650E-5</td>
<td>9.436E-5</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>L.- S.</td>
<td>8.667E-5</td>
<td>11.77E-5</td>
<td>N/A</td>
<td>N/A</td>
</tr>
</tbody>
</table>

- For flat plates, the Chien k-ε model takes only about 10% more CPU time than the B.- L. model.
- For 3-D S-duct and shock/B.L. inter., the Chien k-ε model takes about 25% more CPU time than the B.- L. model.

**Initialization of multi-equation models**

Initial profiles of turbulence variables are needed to start the time marching process. The following steps have been found to work well:

1. $\mu_t$ is computed using the BL model.
2. BB model – $k^2/\varepsilon$ is computed using $\mu_t$ and the BB formula.
3. k-ε models – $\varepsilon$ is calculated from the local equilibrium assumption, and $k$ is calculated using $\varepsilon$, $\mu_t$, and the CH damping function $f_{\mu}$. 

---

NASA Lewis Research Center
Cleveland, Ohio

INTERNAL FLUID MECHANICS DIVISION

---

405
Automatic Initial k Profile for Incompressible Flat Plate TBL

Initial k profile

Converged k profile

Initial k Profile for Turbulent S-Duct Flow

Initial k profile

Converged k profile
**Initial k Profile for Glancing Shock Wave/TBL Interaction**

---

**Y**\(^{+}\) dependency

\(y^{+}\) is needed to feed the Van Driest style damping functions in the near wall region. Not desirable for two main reasons:

1. Geometrical considerations:
   - Need to identify the nearest solid walls. This is difficult for complex 3-D geometries with multiple solid boundaries.
   - Need to compute distances to solid walls. This is difficult to do for unstructured grids.
   - What is \(y^{+}\) in base/wake flows?

2. Ambiguity in the definition of \(y^{+}\) for variable-property flows:

   \[
   y^{+}_{h} = \left( \frac{y_{h}}{V_{1}} \right) \left( \frac{\tau_{w} Re_{r}}{\rho_{1}} \right)
   \]
   - Local or wall values for \(V_{1}\) and \(\rho_{1}\)?

However, models that use \(y^{+}\) appear to be more stable numerically than the ones that do not.
Effects of $y^+$ Computation, CH

\[ x' = \left( \frac{C_F}{C_L} \right) \frac{Re}{Pr} \]

\[ x' = \left( \frac{C_F}{C_L} \right) \frac{Re}{Pr} \]

\[ \pm 10\% \text{ deviation from Van Driest II} \]

NASA Lewis Research Center
INTERNAL FLUID MECHANICS DIVISION

Cleveland, Ohio

Effects of Compressibility Corrections, CH

\[ x' = \left( \frac{C_F}{C_L} \right) \frac{Re}{Pr} \]

\[ x' = \left( \frac{C_F}{C_L} \right) \frac{Re}{Pr} \]

\[ \pm 10\% \text{ deviation from Van Driest II} \]

NASA Lewis Research Center
INTERNAL FLUID MECHANICS DIVISION

Cleveland, Ohio
Where the turbulence models work well

Attached, thin layer flows with little or no pressure gradients
Where they don't work too well

Separated/Reattached flows and flows with large pressure gradients

Comments

• Two-equation k-ε models can be used without serious CPU time penalty. However, no real gain in predictive capability compared with algebraic formulations.

• To improve the predictive capability, need to extend the two-equation formulations with algebraic Reynolds stress models or anisotropic k-ε models.

• y* dependency limits the generality of turbulence models, and it should be removed if possible (preferably without significant cost in the stability and accuracy of the models).

• Models are needed for the Reynolds heat flux terms.
Several Examples Where Turbulence Models Fail in Inlet Flow Field Analysis

Bernhard H. Anderson
NASA Lewis Research Center
Cleveland, OH
Inlet Flow Field Analysis
Computational Uncertainties

- Turbulence Modeling for 3D Inlet Flow Fields
  (1) Flows Approaching Separation
  (2) Strength of Secondary Flow Field
  (3) 3D Flow Predictions of Vortex Liftoff
  (4) Influence of Vortex-Boundary Layer Interactions

- Vortex Generator Modeling
  (1) Representation of Generator Vorticity Field
  (2) Relationship Between Generator and Vorticity Field

Inlet Flow Field Studies
Goals and Objectives

To advance the understanding, the prediction, and the control of intake distortion, and to study the basic interactions that influence this design problem.

- To develop an understanding of and predictive capability for the aerodynamic properties of intake distortion and its management.
- To establish a set of design guidelines to maximize the effectiveness of vortex flow control for the management of intake distortion.
Inlet Flow Field Benchmark Data Sets
Turbulence and Vortex Generator Modeling

- Fraser Flow A, Stanford Conference 1968
- 727/JT8D-100 S-Duct Confirmation Experiment, 1973
- Univ. Washington TD410 Transition Duct Experiment, 1990
- M2129 Imperial College Laser-Doppler Experiment, 1990
- DRA-Bedford Experiments on the M2129 Intake S-Duct
  (2) DRA Surface Pressure and Engine Face Experiment, 1989
  (3) DRA Phase 1B Hot-Wire Flow Experiment, 1990
  (4) DRA Phase 2 Yawmeter Flow Experiment, 1991
  (5) DRA Phase 3 Vortex Generator Experiment, 1992
- TD118 Bi-Furcated Transition Duct Experiment, 1994

Reduced Navier-Stokes Analysis
RNS3D Computer Code

- Velocity decomposition approach, Briley and McDonald (1979 & 1984)
  (1) Conservation form of the vorticity transport equation
  (2) Mass flow conservation, \( m = \int p u_dA = constant \)
- Non-orthogonal coordinate system, Levy, Briley and McDonald (1983)
- Arbitrary geometry gridfile description, Anderson (1990)
  (1) Reccluster existing gridfile mesh distribution
  (2) Redefine the centerline space curve
  (3) Alter cross-sectional duct shape
- McDonald Camarata turbulence model

415
Full Navier-Stokes Analysis
PARC3D Computer Code

• Originally developed by Pulliam & Steger as AIR3D (1980)
  (1) Conservation form of the governing equations
  (2) Beam & Warming approximate factorization algorithm
  (3) Central differencing within a curvilinear system
• Addition of Jameson artificial dissipation by Pulliam, ARC3D (1981)
• Developed for internal flow by Cooper, PARC3D (1987)
  (1) Baldwin-Lomax turbulence model
  (2) Diewert approximation to turbulence model in the reverse flow region of flow field

Fraser Flow A, Stanford Conference 1968
Geometry and Mesh Definition
Fraser Flow A, Stanford Conference 1968
Comparison of Turbulence Models

\( nz = 49, y^+ = 1.17 \)

---

(a) Incompressible Shape Factor (HI) Distribution

(b) Wall Skin Friction \((C_f)\) Distribution
Fraser Flow A, Stanford Conference 1968
Effect of $y^+$ on Flow Field Solution

(a) Incompressible Shape Factor (HI) Distribution

(b) Wall Skin Friction ($C_f$) Distribution

RNS Analysis, McDonald-Camarata Model

Fraser Flow A, Stanford Conference 1968
Effect of Mesh Resolution on Flow Field Solution

(a) Incompressible Shape Factor (HI) Distribution

(b) Wall Skin Friction ($C_f$) Distribution

RNS Analysis, McDonald-Camarata Model
Fraser Flow A, Stanford Conference 1968
Conclusions

(1) The current generation of turbulence models were unable to predict the complete state of the diffuser boundary layer approaching flow separation.

(2) Both near wall and mesh resolution separately played an important role in accurate solutions to wall skin friction distribution in flows characterized as "approaching separation", but had little effect on the solution for the incompressible shape factor development.

(3) It is important that grid independent solution be demonstrated before judgements about the turbulence models be stated, and that the complete state of the wall boundary layer be considered within this evaluation.

727/JT8D-100 Inlet S-Duct
Geometry and Mesh Definition
727/JT8D-100 Inlet S-Duct
Engine Face Flow Field
Generator Config. 12

Experiment
With Engine Dome

Analysis
Without Engine Dome

RNS Analysis, McDonald-Camarata Model

727/JT8D-100 Inlet S-Duct
Engine Face Ring Distortion Characteristics
Generator Config. 12

Radial Distortion
60° Sector Circumferential Distortion

RNS Analysis, McDonald-Camarata Model
(1) The current turbulence models predict the overall performance level of vortex generator installation remarkably well, although much of the detailed flow structure was not resolved.

(2) Turbulence models in 3D inlet flow field analysis can also be evaluated on the basis of standard engine performance parameters, such as radial and circumferential ring distortion descriptors, which provide a sensitive discriminator measuring the state of the overall compressor face flow field.
Univ. Tennessee Diffusing S-Duct
Topology of Vortex Liftoff

Analysis

Univ. Tennessee Diffusing S-Duct
Topography of Vortex Liftoff

$y/\delta_{t} = 0.03$

$y/\delta_{t} = 0.30$

RNS Analysis, McDonald-Camarata Model
Univ. Tennessee Diffusing S-Duct
Total Pressure Coefficient Contours
Without Vortex Generators

Experiment

Analysis

RNS Analysis, McDonald-Camarata Model

Univ. Tennessee Diffusing S-Duct
Total Pressure Coefficient Contours
With Vortex Generators

Experiment

Analysis

RNS Analysis, McDonald-Camarata Model

423
Univ. Tennessee Diffusing S-Duct
Effect of $y^+$ on Engine Face Flow Field
Total Pressure Coefficient Contours

$y^+ = 8.5$

$y^+ = 0.5$

RNS Analysis, McDonald-Camarata Model

Univ. Tennessee Diffusing S-Duct
Effect of $y^+$ on Circumferential Distortion

$y^+ = 0.5$

$y^+ = 8.5$

RNS Analysis, McDonald-Camarata Model
**Univ. Tennessee Diffusing S-Duct**

Conclusions

(1) Initial value space marching 3D RNS procedures adequately described the topological and topographical features of 3D flow separation associated with vortex liftoff.

(2) The current turbulence models predicted the overall structure of vortex generator installation remarkably well, although much of the detailed flow structure was not resolved.

(3) Adequate near wall resolution was necessary to obtain an accurate solution of the phenomena of vortex lift-off.

(4) Circumferential ring distortion is a sensitive discriminator in measuring the state of the engine face flow field.

---

**Univ. Washington TD410 Transition Duct**

Geometry and Mesh Definition

---

425
Univ. Washington TD410 Transition Duct
Case Definitions

<table>
<thead>
<tr>
<th>Case</th>
<th>Grid</th>
<th>Total</th>
<th>CPU (min)</th>
<th>Y⁺</th>
</tr>
</thead>
<tbody>
<tr>
<td>td410.§</td>
<td>199 x 121 x 521</td>
<td>12,545,159</td>
<td>149.0</td>
<td>0.565</td>
</tr>
<tr>
<td>td410.1</td>
<td>99 x 121 x 521</td>
<td>6,241,059</td>
<td>73.4</td>
<td>0.623</td>
</tr>
<tr>
<td>td410.3</td>
<td>49 x 121 x 521</td>
<td>2,851,849</td>
<td>36.0</td>
<td>0.568</td>
</tr>
<tr>
<td>td411.1</td>
<td>99 x 91 x 521</td>
<td>4,693,689</td>
<td>57.1</td>
<td>0.623</td>
</tr>
<tr>
<td>td412.§</td>
<td>199 x 61 x 521</td>
<td>6,324,419</td>
<td>72.9</td>
<td>0.565</td>
</tr>
<tr>
<td>td412.1</td>
<td>99 x 61 x 521</td>
<td>3,146,319</td>
<td>38.2</td>
<td>0.623</td>
</tr>
<tr>
<td>td412.3</td>
<td>49 x 61 x 521</td>
<td>1,557,269</td>
<td>18.7</td>
<td>0.568</td>
</tr>
<tr>
<td>td412.4</td>
<td>49 x 61 x 401</td>
<td>1,198,589</td>
<td>14.7</td>
<td>0.568</td>
</tr>
</tbody>
</table>

Univ. Washington TD410 Transition Duct
Effect of Radial Grid Resolution
Velocity Profiles

(a) Velocity Profiles Along the Minor Axis Surface

(b) Velocity Profiles Along the Major Axis Surface

RNS Analysis, McDonald-Camarata Model
**Univ. Washington TD410 Transition Duct**

**Effect of Radial Grid Resolution**

**Wall Skin Friction**

\[ \text{Data: } \begin{align*} n_z = 199, & \quad y^* = 0.565 \\ n_z = 99, & \quad y^* = 0.623 \\ n_z = 49, & \quad y^* = 0.568 \end{align*} \]

![Graph of Skin Friction](image)

(a) Skin Friction \( C_f \) along the Minor Axis Wall

(b) Skin Friction \( C_f \) along the Major Axis Wall

RNS Analysis, McDonald-Camarata Model

---

**Univ. Washington TD410 Transition Duct**

**Comparison with Experimental Data**

**Velocity Contours**

![Velocity Contours](image)

**Analysis**

**Experiment**

RNS Analysis, McDonald-Camarata Model

427
Univ. Washington TD410 Transition Duct
Conclusions

(1) The current generation of turbulence models predicted the overall development of vortex formation reasonably well, although there were important discrepancies which could not be explained as inadequate near wall or mesh resolution.

(2) Radial mesh resolution had the largest impact in the region along the major axis where the vortex pair was formed.

(3) It is important that grid independent solution be demonstrated before judgements about the turbulence models be stated.

(4) Fully 3D grid independent solutions were achieved with a Reduced Navier-Stokes analysis.

DRA M2129 Diffusing Inlet S-Duct
Geometry and Mesh Definition
DRA M2129 Diffusing Inlet S-Duct
Performance Characteristics

(a) Engine Face Total Pressure Recovery

(b) Engine Face $DC_40$ Distortion

DRA M2129 Diffusing Inlet S-Duct
Separation Characteristics

429
DRA M2129 Inlet S-Duct
Wall Static Pressure Distribution
AGARD Test Case 3.1

(a) \( \theta = 0.0^\circ \) Surface Element.

(b) \( \theta = 180.0^\circ \) Surface Element.

DRA M2129 Diffusing Inlet S-Duct
Engine Face Distortion Characteristics
AGARD Test Case 3.1

(a) Radial Ring Distortion

(b) 60\(^\circ\)-Sector Circumferential Distortion
DRA M2129 Inlet S-Duct

Conclusions

(1) Both Full Navier-Stokes (FNS) and Reduce Navier-Stokes (RNS) analyses adequately describe the overall flow physics of vortex liftoff, but consistently predict the location of liftoff further downstream in the duct inlet than was indicated by data.

(2) The current generation of turbulence models were unable to described the influence of separation on the main pressure field for "strong" vortex liftoff interactions.

(3) The current generation of mixing length turbulence models give remarkable good performance results, while the existing discrepancies between data and analysis can be attributed primarily to the over prediction of the liftoff location.

TD118 Bi-Furcated Transition Duct

Geometry and Mesh Definition
TD118 Bi-Furcated Transition S-Duct
Research Objectives

- To demonstrate diffuser duct technology advancement by using CFD to design a "conventionally shorter" transitioning S-duct configuration for application towards high speed inlet systems.
- To develop a computational protocol whereby turbulence model evaluations can be made between different computer codes.
- To develop a benchmark data set to evaluate CFD analysis and turbulence models, which cover fundamental flow phenomena as well as overall flow field physics as determined by standard engine performance parameters.

TD118 Bi-Furcated Transition S-Duct
Effect of Turbulence Model on Inlet Performance

<table>
<thead>
<tr>
<th>Analysis</th>
<th>Turbulence Model</th>
<th>$P_{ave}/P_0$</th>
<th>$DH$</th>
<th>$DC_{60}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>RNS3D</td>
<td>McDonald-Camarata</td>
<td>0.955</td>
<td>0.167</td>
<td>0.116</td>
</tr>
<tr>
<td>PARC3D</td>
<td>Baldwin-Lomax</td>
<td>0.959</td>
<td>0.163</td>
<td>0.135</td>
</tr>
<tr>
<td>PARC3D</td>
<td>P.D. Thomas</td>
<td>0.970</td>
<td>0.153</td>
<td>0.073</td>
</tr>
<tr>
<td>PARC3D</td>
<td>Launder-Spaulding</td>
<td>0.975</td>
<td>0.150</td>
<td>0.177</td>
</tr>
</tbody>
</table>

$DH = (P_{t_{max}} - P_{t_{min}})/P_{t_{ave}}$

$DC_{60} = (P_{t_{ave}} - P_{t_{min,60}})/q_{ave}$
TD118 Bi-Furcated Transition S-Duct
Effect of Turbulence Model on Engine Face Flow Field

(a) RNS Analysis, McDonald-Camarata Model
(b) FNS Analysis, Baldwin-Lomax Model
(c) FNS Analysis, P. D. Thomas Model
(d) FNS Analysis, Launder-Spalding ($k - \varepsilon$) Model

TD118 Bi-Furcated Transition S-Duct
Effect of Turbulence Model on Wall Boundary Layer
$\theta = 90.0^\circ$ Surface Element

(a) Incompressible Shape Factor $HI$ Distribution.
(b) Wall Skin Friction $C_f$ Distribution.
Inlet Flow Field Analysis

Concluding Remarks

(1) Difficulties in complex 3D flow fields often arise because fundamental 2D aerodynamic interactions have not been adequately resolved.

(2) Near wall ($y^+$) and radial mesh resolution ($nz$) play an important role in fundamental 2D and complex 3D flow field analysis.

(3) Judgements about turbulent models should not be stated until grid independent solutions have been established.

(4) Adequateness of turbulence models in inlet flow field analysis should also be made on the basis of fundamental performance parameters used to quantify the "goodness" of the flow entering the engine.
The purpose of this meeting was to discuss the current status and future development of turbulence modeling in computational fluid dynamics for aerospace propulsion systems. Various turbulence models have been developed and applied to different turbulent flows over the past several decades and it is becoming more and more urgent to assess their performance in various complex situations. In order to help users in selecting and implementing appropriate models in their engineering calculations, it is important to identify the capabilities as well as the deficiencies of these models. This also benefits turbulence modelers by permitting them to further improve upon the existing models. This workshop was designed for exchanging ideas and enhancing collaboration between different groups in the Lewis community who are using turbulence models in propulsion related CFD. In this respect this workshop will help the Lewis goal of excelling in propulsion related research. This meeting had seven sessions for presentations and one panel discussion over a period of two days. Each presentation session was assigned to one or two branches (or groups) to present their turbulence related research work. Each group was asked to address at least the following points: current status of turbulence model applications and developments in the research; progress and existing problems; requests about turbulence modeling. The panel discussion session was designed for organizing committee members to answer management and technical questions from the audience and to make concluding remarks.