Industry-Wide 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, Ohio Aerospace Institute, Cleveland, Ohio, October 6-7, 1994

National Aeronautics and Space Administration
Office of Management
Scientific and Technical Information Program
1995
Preface

This publication contains the presentations made at the Industry-wide Workshop on Computational Turbulence Modeling, which was hosted by ICOMP/LeRC, and took place on October 6-7, 1994 at the Ohio Aerospace Institute. The purpose of the workshop was to initiate the transfer of technology developed at Lewis Research Center (LeRC) to industry and to discuss the current status and the future needs of turbulence models in industrial CFD. To address the latter, a total of fourteen presentations were made by researchers from industry. CMOTT would like to thank all the workshop speakers for bringing to our attention a host of problems which are important to industry and for which they think CMOTT can be of help. We are prioritizing all the suggestions in order to incorporate them into the CMOTT work plan.

One unanimous recommendation of the workshop participants was to make the workshop an annual event. This first workshop grew out of the recommendations by the peer review committee of the LeRC turbulence modeling program, held in September of 1993. It could have not successfully transpired without the help and guidance of Dr. Chander Prakash (GE-Aircraft Engines), Dr. Munir Sindir (RocketDyne), and Dr. Saadat Syed (Pratt & Whitney), and for this CMOTT would like to thank them.
WORKSHOP ORGANIZING COMMITTEE

Industries

C. Prakash, General Electric  
M. Sindir, Rocketdyne  
S. Syed, Pratt & Whitney

Universities

J.Y. Chen, University of California, Berkeley  
J.L. Lumley, Cornell University

National Aeronautics and Space Administration

L.A. Povinelli, Chairman  
R. Mankbadi, Lewis Research Center  
D.R. Reddy, Lewis Research Center  
P. Richardson, Headquarters  
R.J. Shaw, Lewis Research Center

Institute for Computational Mechanics in Propulsion

T. Keith, Ohio Aerospace Institute  
A. Shabbir, Center for Modeling of Turbulence and Transition  
T.-H. Shih, Center for Modeling of Turbulence and Transition
TABLE OF CONTENTS

TURBULENCE PROGRAM FOR PROPULSION SYSTEMS
Tsan-Hsing Shih, Institute for Computational Mechanics in Propulsion and Center for
Modeling of Turbulence and Transition, NASA Lewis Research Center .......................... 1

TURBULENCE MODEL DEVELOPMENT AND APPLICATION AT LOCKHEED FORT
WORTH COMPANY
Brian R. Smith, CFD Group, Lockheed Fort Worth Company ........................................... 29

A SUMMARY OF COMPUTATIONAL EXPERIENCE AT GE AIRCRAFT ENGINES
FOR COMPLEX TURBULENT FLOWS IN GAS TURBINES
R. Zerkle and C. Prakash, GE Aircraft Engines ................................................................. 39

THE APPLICABILITY OF TURBULENCE MODELS TO AERODYNAMIC AND PROPULSION
FLOWFIELDS AT McDonnell Douglas Aerospace
Linda D. Kral, John A. Ladd, and Mori Mani, McDonnell Douglas Aerospace .................. 47

EXPERIENCE WITH k-e TURBULENCE MODELS FOR HEAT TRANSFER
COMPUTATIONS IN ROTATING
Prabhat Tekriwal, GE Corporate Research and Development ........................................ 65

TURBULENCE MODELS FOR GAS TURBINE COMBUSTORS
Andreja Brankovic, CFD Group, Pratt & Whitney ......................................................... 79

COMBUSTION SYSTEM CFD MODELING AT GE AIRCRAFT ENGINES
D. Burrus and H. Mongia, GE Aircraft Engines, and A. Tolpadi, S. Correa, and M. Braaten,
GE Corporate Research and Development ................................................................. 87

RECENT PROGRESS IN THE JOINT VELOCITY-SCALAR PDF METHOD
M.S. Anand, Allison Engine Company .................................................................................. 99

OVERVIEW OF TURBULENCE MODEL DEVELOPMENT AND APPLICATIONS AT
ROCKETDYNE
A.H. Hadid, E.D. Lynch, and M.M. Sindir, Rocketdyne Division, Rockwell
International ......................................................................................................................... 107

RECENT ADVANCES IN PDF MODELING OF TURBULENT REACTING FLOWS
A.D. Leonard and F. Dai, CFD Research Corporation ...................................................... 119

EXPERIENCE WITH TURBULENCE INTERACTION AND TURBULENCE-CHEMISTRY
MODELS AT FLUENT INC.
D. Choudhury, S.E. Kim, D.P. Tselepidakis, and M. Missaghi, Fluent Inc ......................... 131

EXPERIENCES WITH TWO-EQUATION TURBULENCE MODELS
Ashok K. Singhal, Yong G. Lai, and Ram K. Avva, CFD Research Corporation .............. 143

PROGRESS IN SIMULATING INDUSTRIAL FLOWS USING TWO-EQUATION MODELS:
CAN MORE BE ACHIEVED WITH FURTHER RESEARCH?
Vahé Haroutunian, Fluid Dynamics International, Inc. .................................................. 155
TURBULENCE PROGRAM FOR PROPULSION SYSTEMS

Tsan-Hsing Shih
Institute for Computational Mechanics in Propulsion and Center for Modeling of Turbulence and Transition
NASA Lewis Research Center
Cleveland, Ohio

BACKGROUND

• CMOTT group at LeRC has been in existence for about 4 years. In the first 3 years, its main activities were in developing and validating turbulence and combustion models for propulsion systems, in an effort to remove the deficiencies of the existing models. Two workshops on computational turbulence modeling were held at LeRC (1991, 1993).

• A peer review of turbulence modeling activities at LeRC was held in September, 1993. Seven peers (GE, P&W, RocketDyne, Cornell, Berkeley and NASA Ames) conducted the peer review. The objective of the peer review was to assess the turbulence program at LeRC/CMOTT and to suggest the future direction of turbulence modeling activities for propulsion systems.

• Important messages from the peer review:

  ◊ "LeRC should spend substantial effort being responsive to industry’s current pressing perceived needs; this involves extensive discussion with industry during every phase of model development, analysis of industry’s problems, goal oriented model development, evaluation of models relative to industry’s intended application ..."

  ◊ "LeRC has an obligation not only to respond to industry’s requests for help, but to play an autonomous, independent leadership role in providing models of the highest quality, ..., which can be employed not only by the aircraft gas turbine and rocket industries but also by other industries ..."

  ◊ "In the present financial climate, industry does not have the resources to undertake model development and evaluation. LeRC’s help in this regard via the creation of its turbulence modeling effort, is, therefore, welcome from the industry’s standpoint."

  ◊ "It is important to work with the industry to evaluate the models and rank-order them by performance and cost in order to identify the most appropriate models for particular situations."

  ◊ Many other useful suggestions and comments including collaboration with industry, joint programs, industry-wide workshop ...
PROGRAM GOALS AT CMOTT

- Develop reliable turbulence (including bypass transition) and combustion models for complex flows in propulsion systems
- Integrate developed models into deliverable CFD tools for propulsion systems in collaboration with industry.

PROGRAM APPROACH

- Develop turbulence and combustion modules for industry customers
- Industry collaboration and technology transfer
- Model development for propulsion systems
  - One-point moment closures for non-reacting flows
  - Scalar PDF method for turbulent reacting flows
  - Validation of existing and newly developed models
Development of Turbulence and Combustion Modules

- Objective
  - Build a quick and efficient vehicle for technology transfer to industry

- The features of the turbulence module:
  - It contains various turbulence models from which users can choose the appropriate model for flows of interest
  - It is self-contained, i.e., it contains its own solver for turbulence model equations
  - It can be easily linked to industry's CFD codes

- Turbulence module for NPARC code has been developed, tested, and is ready to be released
  - The models built-in at the present time:
    - Mixing length, Chien $k - \varepsilon$, CMOTT $k - \varepsilon$ models
  - The model to be built-in:
    - CMOTT algebraic Reynolds stress, Reynolds stress transport equation models and other models based on the request from industries.
  - Built-in robust, realizable numerical solver for model equations.

- General turbulence modules
  - Can be used for both compressible and incompressible flows.
  - Interface programs for different industry CFD codes
  - Built-in models will be periodically updated.
Nozzle exit height = 0.122 in.

Fig. 3. Schematic of ejector nozzle test case.
THROAT HEIGHT ($H_T$) = 44 mm

Strong-shock case

(a) Top wall

(b) Bottom wall

Exp
Chien
Shih-Lumley
Collaboration with Industry and Technology Transfer

- Joint research programs with industry
  - Preliminary programs with engine companies and others have been initiated (GE, P&W, RocketDyne, Naval Research Laboratories)
  - Develop further joint research programs related to the industry's projects
- Industry-wide workshops will be a regular program (once every two years)
  - Release Lewis turbulence and combustion modules to industries
  - Discuss the needs of industry

Models developed at CMOTT

1. Isotropic eddy viscosity models
2. Reynolds stress & scalar flux algebraic equation models
3. Second moment transport equation models
4. Multiple-scale models for compressible turbulent flows
5. Bypass transition models
6. PDF models for turbulent reacting flows

PROGRAM SUMMARY
Isotropic eddy viscosity models

- Objective
  - To examine the deficiencies of existing models
  - To develop better eddy viscosity models

- Current status of existing $k - \varepsilon$ eddy viscosity models

\[-\bar{u}_i u_j = \nu_T (U_{i,j} + U_{j,i}) - \frac{2}{3} k \delta_{ij}, \quad \nu_T = C_\mu f_\mu \frac{k^2}{\varepsilon}\]

\[\frac{Dk}{Dt} = T^{(k)} + P^{(k)} - \varepsilon + W.C., \quad \frac{De}{Dt} = T^{(e)} + P^{(e)} - D^{(e)} + W.C.\]

- They are not tensorially invariant due to $f_\mu(y^+), W.C.(y^+)$
- Model constants are not consistent for flows with and without wall
- Normal stresses may violate realizability
- Do not work very well for flows with pressure gradients

- Development of a Galilean-, tensorially invariant, realizable, $k - \varepsilon$ model
  - New damping function $f_\mu(k/S\nu)$ is proposed to remove the dependence on $y$
  - New dissipation $\varepsilon$ equation is introduced to give better response to pressure gradients
  - Consistent model coefficients for all flows
  - Realizability of the normal stresses is guaranteed
  - Modified wall function for cases with pressure gradients
- CMOTT $k - \varepsilon$ eddy viscosity model

$$-u_i u_j = \nu T(U_{i,j} - U_{j,i}) - \frac{2}{3} k \delta_{ij}, \quad \nu T = C_\mu f_\mu \frac{k^2}{\varepsilon}$$

$$\frac{Dk}{Dt} = T_k + P_k - \varepsilon$$

$$\frac{D\varepsilon}{Dt} = T_\varepsilon + C_1 f_1 S \varepsilon - C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + f_\phi \Phi$$

- $f_\mu, f_1, f_\phi$ are functions of $R = k/S\nu$, which is tensorially invariant
- $C_\mu = \frac{1}{A_0 + A_\ast U^* k/\varepsilon}$, which ensures realizability for normal stresses
- $\Phi$ represents the effect of inhomogeneity

$$\Phi = b_1 \nabla k \nabla k + b_2 \frac{k^2}{\varepsilon} \nabla S \nabla k + b_3 \frac{k^4}{\varepsilon^2} \nabla S \nabla S$$

- Validation

Flows:

- Channel flows
- Boundary layer flows with and without pressure gradients
- Planar jet, round jet and mixing layer
- Backward-facing step flows
- Complex flows related to industrial applications

Models:

- Launder-Sharma, Lam-Bremhorst, Chien, Nagano-Hishida, ...
- $k - \omega$ model (Wilcox)
- CMOTT $k - \varepsilon$ model
Present model with the modified wall function
Spreading Rate of Free Shear Flows

<table>
<thead>
<tr>
<th></th>
<th>exp.</th>
<th>st. $k - \epsilon$</th>
<th>Chien $k - \omega$</th>
<th>CMOTT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Planar Jet</td>
<td>0.10-0.11</td>
<td>0.108</td>
<td>0.098</td>
<td>0.14*</td>
</tr>
<tr>
<td>Round Jet</td>
<td>0.085-0.095</td>
<td>0.116</td>
<td>0.104</td>
<td>0.32*</td>
</tr>
<tr>
<td>Mixing Layer</td>
<td>0.13-0.17</td>
<td>0.152</td>
<td>0.152</td>
<td>0.16*</td>
</tr>
</tbody>
</table>

Planar Jet

\[
\frac{U}{U_0} = f(y/x)
\]

Round Jet

\[
\frac{U}{U_0} = g(y/x)
\]
Algebraic Reynolds stress models

• Objective
  ◦ To examine the deficiencies of existing ARS models
  ◦ To develop better ARS models

• Current status of ARS models
  ◦ Second-order closure based ARS models (Rodi, 1980)

$$\frac{u_i u_j}{k} (P - \varepsilon) = -u_i u_k U_{j,k} - u_j u_k U_{i,k} - \frac{1}{\rho} \left( \frac{\partial u_j}{\partial x} \right) + \frac{\partial u_i}{\partial x} - 2 \nu u_i, k u_j, k$$

Comments:
  * Assumption: $u_i, u_j / k = \text{Const.}, \left( u_i u_j u_k \right), k = (k u_i), i = 0$
  * Numerical difficulties

◊ Pope's explicit ARS model (2-D flows), Taulbee's ARS model (3-D), Gatski and Speziale's ARS model

◊ Other methods: RNG, DIA and invariant theory
• General constitutive relations from invariant theory

\[
\overline{u_i u_j} = \frac{2}{3} k \delta_{ij} + 2a_2 \frac{K^2}{\varepsilon} (U_{i,j} + U_{j,i} - \frac{2}{3} U_{i,i} \delta_{ij}) + 2a_4 \frac{K^3}{\varepsilon^2} (U_{i,j}^2 + U_{j,i}^2 - \frac{2}{3} \Pi_1 \delta_{ij}) \\
+ 2a_6 \frac{K^3}{\varepsilon^2} (U_{i,k} U_{j,k} - \frac{1}{3} \Pi_2 \delta_{ij}) + 2a_7 \frac{K^3}{\varepsilon^2} (U_{k,i} U_{k,j} - \frac{1}{3} \Pi_2 \delta_{ij}) \\
+ 2a_8 \frac{K^4}{\varepsilon^3} (U_{i,k} U_{j,k}^2 + U_{j,k} U_{i,k} - \frac{2}{3} \Pi_3 \delta_{ij}) + 2a_{10} \frac{K^4}{\varepsilon^3} (U_{k,i} U_{k,j}^2 + U_{k,j} U_{k,i} - \frac{2}{3} \Pi_3 \delta_{ij}) \\
+ 2a_{12} \frac{K^5}{\varepsilon^4} (U_{i,k} U_{j,k}^2 - \frac{1}{3} \Pi_4 \delta_{ij}) + 2a_{13} \frac{K^5}{\varepsilon^4} (U_{k,i} U_{k,j}^2 - \frac{1}{3} \Pi_4 \delta_{ij}) \\
+ 2a_{14} \frac{K^5}{\varepsilon^4} (U_{i,k} U_{i,k} U_{j,i} + U_{j,k} U_{i,k} U_{i,i} - \frac{2}{3} \Pi_5 \delta_{ij}) \\
+ 2a_{16} \frac{K^6}{\varepsilon^5} (U_{i,k} U_{i,k} U_{j,i}^2 + U_{i,k} U_{i,k} U_{i,i}^2 - \frac{2}{3} \Pi_6 \delta_{ij}) \\
+ 2a_{18} \frac{K^7}{\varepsilon^6} (U_{i,k} U_{i,k} U_{i,m} U_{j,m} + U_{j,k} U_{i,k} U_{i,m} U_{i,m} - \frac{2}{3} \Pi_7 \delta_{ij})
\]

• RDT and realizability constraints (Reynolds, Lumley)

• CMOTT algebraic Reynolds stress model

\[
\overline{u_i u_j} = \frac{2}{3} k \delta_{ij} - C_\mu \frac{k^2}{\varepsilon} 2 S_{ij}^* + 2C_2 \frac{k^3}{\varepsilon^2} (-S_{ik}^* \Omega_{kj}^* + \Omega_{ik}^* S_{kj}^*)
\]

\[
k_{,t} + U_{j,k,j} = [\nu + \frac{\nu_t}{\sigma_k}] k_{,j}, j - \overline{u_i u_j} U_{i,j} - \varepsilon.
\]

\[
\varepsilon_{,t} + U_{j,\varepsilon,j} = [\nu + \frac{\nu_t}{\sigma_\varepsilon}] \varepsilon_{,j}, j - C_{e1} \frac{\varepsilon}{k} \overline{u_i u_j} U_{i,j} - C_{e2} \frac{\varepsilon^2}{k}
\]

where

\[
C_\mu = \frac{1}{A_0 + A_{\mu} \frac{U_{,k}}{\varepsilon}}, \quad C_2 = \frac{\sqrt{1 - 9C_\mu^2 (S_{,k})^2}}{C_0 + 6 S_{,k} \Omega_{,k}^*} \\
\nu_t = C_\mu \frac{k^2}{\varepsilon}, \quad A_0 = 6.5, \quad C_0 = 1.0 \\
C_{e1} = 1.44, \quad C_{e2} = 1.92, \quad \sigma_k = 1, \quad \sigma_\varepsilon = 1.3
\]
• Validation

◊ Rotating homogeneous shear flows
◊ Backward-facing step flows
◊ Confined jets
◊ Complex flows related to industrial applications

Configuration of rotating homogeneous shear flow

Evolution of turbulent kinetic energy with time.

— — : present model; — — : SKE; • : LES
Table Comparison of the reattachment points

<table>
<thead>
<tr>
<th>Case</th>
<th>measurement</th>
<th>SKE</th>
<th>PRESENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>DS</td>
<td>6.1</td>
<td>4.99</td>
<td>5.82</td>
</tr>
<tr>
<td>KKJ</td>
<td>7 ± 0.5</td>
<td>6.35</td>
<td>7.35</td>
</tr>
</tbody>
</table>

---: present model; ---: SKE; •: EXP
Scalar turbulence model

- Objective

To improve the predictive capability of current scalar turbulence ($\overline{\theta^2} - \varepsilon_\theta$) models

- A new scalar flux constitutive relation
- A new scalar dissipation rate model equation

\[
\begin{align*}
\overline{u_i \theta} &= -C_\lambda \frac{k^2}{\epsilon} \left( \frac{2}{r} \right) (1/2) \Theta_{,i} + C \frac{k^2}{2r} (1/2) (a_2 U_{i,j} + a_3 U_{j,i}) \Theta_{,j} \\
U_j \frac{\partial \overline{\theta^2}}{\partial x_j} &= \frac{\alpha T}{\sigma_x} \frac{\partial \theta^2}{\partial x_j} - 2 \overline{u_i \theta} \frac{\partial \theta}{\partial x_i} - 2 \varepsilon_\theta \\
U_j \frac{\partial \varepsilon_\theta}{\partial x_j} &= \frac{\alpha T}{\sigma_x} \varepsilon_\theta - C_{\theta 1} \varepsilon_\theta S + C_{\theta 2} \sqrt{\frac{\varepsilon_\theta \epsilon}{P_T}} S_T - C_{\theta 3} \frac{\theta \epsilon}{k} \\
C_\lambda &= \frac{(2 + 2r + 0.5r^2)}{26 + 3.2\eta^2 + 2r^2} \\
S_T &= \sqrt{\Theta_{,i} \Theta_{,i}}, \quad S = \sqrt{2 S_{ij} S_{ij}}, \quad \eta = S k/\epsilon, \quad \xi = \frac{k}{\epsilon} (\frac{S_{ij}}{S_{ij}})^{1/2} S_T, \quad r = \frac{2k}{\epsilon} \frac{S_{ij}}{S_{ij}} \\
C_{\theta 1} &= C_1 - 0.13, \quad C_{\theta 2} = 0.63, \quad C_{\theta 3} = C_2 - 1, \quad \sigma_x = 1.0, \quad \sigma_\phi = 1.8
\end{align*}
\]
Flat plate boundary layer with constant surface temperature
• Validation

◊ Homogeneous turbulence subjected to $\partial \Theta/\partial y$
◊ Homogeneous turbulence subjected to $\partial U/\partial y$, $\partial \Theta/\partial y$
◊ Flat plate boundary layer with constant surface temperature

• Work in progress

◊ Model assessment for different scalar boundary conditions
◊ Model extension for integration to the wall
Second Order Closure Models

\[ \frac{D\overline{u_i u_j}}{Dt} = T_{ij} + P_{ij} + \Pi_{ij}^{\text{Rapid}} + \Pi_{ij}^{\text{Return}} - \frac{2}{3} \varepsilon \delta_{ij} \]

- Objective
  - To assess existing models
  - To find the direction of improving closure models

- Basic model forms
  \[ \Pi_{ij}^{\text{Rapid}} = F_{ij}(S_{ij}, \overline{u_i u_j}), \]
  \[ \Pi_{ij}^{\text{Return}} = F_{ij}(\overline{u_i u_j}, \nu, k, \varepsilon), \]
  \[ T_{ij} = F_{ij}( (\overline{u_i u_j})_k, k, \varepsilon) \]

- General comments on second order closures:
  - The model, \( \Pi_{ij}^{\text{Rapid}} \), is relatively well developed compared with other terms
  - The model, \( \Pi_{ij}^{\text{Return}} \), is least developed
  - A Galilean and tensorially invariant second order closure model has not been well developed yet
  - All models have large errors near the wall, especially in the buffer layer; therefore, for engineering application, the wall function approach is suggested at the present time
- Application of realizability to IP and LRR models
Multiple scale $k$-$\varepsilon$ model

- **Objective:**
  - To consider the effect of a non-equilibrium energy spectrum on eddy viscosity for compressible turbulence

- **Approach:**
  - Use multiple scale concept introduced by

  - **Large-Scale**
    \[
    \overline{\rho} \frac{D\tilde{k}_p}{Dt} = \frac{\partial}{\partial y} \left[ (\bar{\mu} + \frac{\mu_T}{\sigma_{\varepsilon_p}}) \frac{\partial \tilde{k}_p}{\partial y} \right] + \mu_T \left( \frac{\partial \tilde{u}}{\partial y} \right)^2 - \overline{\rho} \tilde{\varepsilon}_p + fc_1
    \]
    \[
    \overline{\rho} \frac{D\tilde{\varepsilon}_p}{Dt} = \frac{\partial}{\partial y} \left[ (\bar{\mu} + \frac{\mu_T}{\sigma_{\varepsilon_p}}) \frac{\partial \tilde{\varepsilon}_p}{\partial y} \right] + C_{p_1} \frac{\tilde{\varepsilon}_p}{k_p} \mu_T \left( \frac{\partial \tilde{u}}{\partial y} \right)^2 - C_{p_2} \overline{\rho} \tilde{\varepsilon}_p^2 \]
  - $fc_1$ - exchanges between the turbulent kinetic energy and internal energy
  - $fc_2$ - increased spectral energy transfer due to compressibility effects

  - **Small Scale**
    \[
    \overline{\rho} \frac{D\tilde{k}_i}{Dt} = \frac{\partial}{\partial y} \left[ (\bar{\mu} + \frac{\mu_T}{\sigma_{\varepsilon_i}}) \frac{\partial \tilde{k}_i}{\partial y} \right] + \overline{\rho} \tilde{\varepsilon}_i - \tilde{\varepsilon}_i
    \]
    \[
    \overline{\rho} \frac{D\tilde{\varepsilon}_i}{Dt} = \frac{\partial}{\partial y} \left[ (\bar{\mu} + \frac{\mu_T}{\sigma_{\varepsilon_i}}) \frac{\partial \tilde{\varepsilon}_i}{\partial y} \right] + C_{t_1} \tilde{\varepsilon}_i \tilde{\varepsilon}_p \frac{\tilde{\varepsilon}_i}{k_t} - C_{t_2} \overline{\rho} \tilde{\varepsilon}_i^2
    \]

  - **Eddy Viscosity**
    \[
    \mu_T \approx \overline{\rho} u l \approx \overline{\rho} \left( \tilde{k}_p + \tilde{k}_i \right)^{1.5} \left( \frac{\tilde{k}_p + \tilde{k}_i}{\tilde{\varepsilon}_p} \right)^{1.5}
    \]
Model Evaluation

- Turbulent Shear Flow

- Shock/Turbulent-Boundary-Layer Interactions
  - transonic flow

- supersonic flow

Compressible Turbulent Shear Flow
Flow over a Bump—Bachalo and Johnson (1979)

Shock Reflection—Reda et al. (1973-1977)
Bypass transition models

- Objective:
  - Develop transition models for flows with free stream turbulence

- Approach:
  - Using $K-\varepsilon$ model as the base model
  - Introduce effective intermittency to either the eddy viscosity or the $k-\varepsilon$ model equations
PDF modeling of turbulent reacting flows

- Objective:
  - Develop models that can accurately simulate finite chemical reactions in turbulent flows.
  - Develop and validate independent PDF models.
  - Technology transfer.

- Approach:
  - Joint pdf for scalar compositions.
  - Moment closure schemes for velocity field.
  - Develop hybrid solver consisting of Monte Carlo method and finite-difference/finite-volume method.
TURBULENCE MODEL DEVELOPMENT AND APPLICATION AT
LOCKHEED FORT WORTH COMPANY

Brian R. Smith
CFD Group
Lockheed Fort Worth Company
Fort Worth, Texas

Broad Range of Flow Problems of Interest

Wide Range of Flow Conditions:
Subsonic – Hypersonic
Internal – External – Store Separation
Cruise – High Angle of Attack

Flows phenomena of Interest:
Inlets/Diffusers
Streamwise Curvature
Shock/BL Interactions
Rectangular Duct – Circular

Leading Edge Separation – Cowl Lips
Separation Induced Unstart

Nozzles
Entrainment
Round – Rectangular Duct
High Speed Shear Layers

Film cooling, Liners, Vanes
Swirl

External Aerodynamics
Vortex
Leading Edge Separation
Shock/BL Interactions

3D Boundary Layers
Wakes

The CFD Environment at Lockheed Fort Worth Company

Most codes developed or highly modified in house
General grid generation and solvers for diverse applications
Structured and unstructured solvers
Computational efficiency important
• Complex geometries, many gridpoints
• Large arrays of flow conditions
Requirements for Turbulence Models

Turbulence Modeling Priorities for Industrial Application

- Validation
  - High accuracy for attached flows
  - Reasonable accuracy for all flows
  - High confidence level
- Computational efficiency
- Robust for complex geometries
- Transitional modeling capability

To obtain acceptable accuracy, propulsion flows demand more sophisticated turbulence models than do external aerodynamic flows

The k - kl and k - I Two Equation Turbulence Models

Advantages of using kl or I instead of $\varepsilon$ or $\omega$

kl and I equations are easier to resolve numerically than $\varepsilon$ equation

Dissipation Length Scale is an integral length scale
- Can derive equation for volume integral of two point correlation function.
- Theoretical $\varepsilon$ equation is dominated by small scales

k - kl and k - I agree better with compressible boundary layer data than does $k - \varepsilon$

Disadvantage - current formulation requires calculation of distance to walls

**k - kl model**
- Includes unique, consistent wall function
- Accurate for transonic flows

**k - I model**
- Derived from k - kl model - identical in high Re turbulence
- Near wall model simulates $k$ in viscous sublayer

30
The $k - k_l$ Model Wall Function

Wall layer model derived from and consistent with the $k - k_l$ model

- Assume convection in momentum, energy and turbulent kinetic energy equations to be negligible
- Boundary layer approximation

Match velocity, $k$ and $l$ at first grid point in Navier - Stokes solution

First grid point can be in viscous sublayer, buffer or logarithmic region

Boundary conditions on $k$ and $l$ simple for $k - k_l$ model

Advantages of wall functions

- Reduces number of necessary grid points
- Reduces number of iterations to converge steady state solution 60 - 90%

Wall Functions are Accurate for Separated Flow Applications

Axial Symmetric Bump, Transonic Flow Experiment

<table>
<thead>
<tr>
<th>$C_p$</th>
<th>$X/C$</th>
</tr>
</thead>
<tbody>
<tr>
<td>-1.0</td>
<td>0.5</td>
</tr>
<tr>
<td>-0.75</td>
<td>0.75</td>
</tr>
<tr>
<td>-0.5</td>
<td>1.0</td>
</tr>
<tr>
<td>-0.25</td>
<td>1.25</td>
</tr>
<tr>
<td>0.0</td>
<td>1.5</td>
</tr>
</tbody>
</table>

Experimental Data

- Fine Grid, $y^+ = 1.5$
- Coarse Grid, $y^+ = 40$

$M = 0.925$
$M = 0.975$
$M = 0.80$

Velocity profiles with and without wall functions

<table>
<thead>
<tr>
<th>$Y(cm)$</th>
<th>$U/U_o$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>0.25</td>
</tr>
<tr>
<td>1.0</td>
<td>0.50</td>
</tr>
<tr>
<td>1.5</td>
<td>0.75</td>
</tr>
</tbody>
</table>

Experimental Data

- Fine Grid, $y^+ = 1.5$
- Coarse Grid, $y^+ = 40$

$X_C = 0.25$
$X_C = 0.50$
The k – 1 Model with Near Wall Model

k\(\ell\) equation is transformed exactly to an l equation

Advantages of k – 1 formulation

- \(l\) is linear near wall, \(k\ell\) nonlinear and very small
- Near wall damping terms disappear
- Production term drops out with current choice of constants

k – 1 model includes:

- Transitional flow modeling
- Compressibility corrections

Modeling of details of k profile near wall important for hypersonic flows

- Magnitude of normal stress term comparable to static pressure
- Near wall density variations large

\(l\) Equation Much Easier to Resolve than \(\varepsilon\) Equation

\(\varepsilon\) equation requires fine grid from wall to \(y^*\) of 20 to resolve peak

- Exclusion of near wall viscous dissipation term aggravates problem
- Logarithmic region, \(\varepsilon \approx 1/y\)

\(l\) equation is nearly linear near wall - much less sensitive to grid resolution
Resolution Study with $k - \varepsilon$ and $k - f$ Models

Sample Applications:

Mach 8 Shock Wave Turbulent Boundary Layer Interactions

F-16 Inlet Derivative, Isolated Duct Study

Multi-slot Ejector

F110 Nozzle Drag Reduction Study
The \( k-\Omega \) Model Predicts Turbulent Shock - Wave Boundary Layer Interaction Well

Mach 8, 10 Degree Wedge Generator

2D case, Separated Flow
Afterbody/Nozzle Pressure Distributions Match Test Data
Mach 0.6

Upper Centerline

Cp
-0.6 -0.5 -0.4 -0.3 -0.2 -0.1 0 0.1 0.2 0.3 0.4
F.S. (ft)

72 Degrees

Cp
-0.6 -0.5 -0.4 -0.3 -0.2 -0.1 0 0.1 0.2 0.3 0.4
F.S. (ft)

Nozzle Surface Cp
At nose looking forward

Lower Centerline

Cp
-0.6 -0.5 -0.4 -0.3 -0.2 -0.1 0 0.1 0.2 0.3 0.4
F.S. (ft)
Good Predictions of Multi-Slot Ejector Obtained with k-kl Model

NPR = 14
P_e/P_t

Mach Contours

k-kl Model Predicts Entrainment Effects Near Slots

Velocity vectors colored by Mach Number

Mach Contours
Summary

Computationally efficient k-I and k-kl models have been developed and implemented at Lockheed Fort Worth Company.

Many years of experience applying two equation turbulence models to complex 3D flows for design and analysis.
A SUMMARY OF COMPUTATIONAL EXPERIENCE AT GE AIRCRAFT ENGINES FOR
COMPLEX TURBULENT FLOWS IN GAS TURBINES

R. Zerkle and C. Prakash
GE Aircraft Engines
Cincinnati, Ohio

CONTENTS:

• INTRODUCTION
• 2–D BOUNDARY LAYER CODE WITH LRN TURBULENCE MODEL
• 3–D NAVIER–STOKES CODE WITH WALL FUNCTIONS
• 3–D NAVIER–STOKES CODE WITH LRN TURBULENCE MODEL
• FILM COOLING SIMULATION
• TURBULATED PASSAGE SIMULATION
• OVERALL CONCLUSIONS
• LIST OF REFERENCES

INTRODUCTION:

• Indications are that the standard k–ε turbulence model together with
  standard wall functions are adequate for CFD simulations in cavities
  away from the primary gaspath of a gas turbine engine.
• However, CFD simulations in the primary gaspath and in blade cooling
  passages require more advanced turbulence models.
• Therefore, this presentation will summarize some CFD experience at
  GEAE only for flows in the primary gaspath of a gas turbine engine and
  in turbine blade cooling passages.
2-D BOUNDARY LAYER CODE WITH LOW REYNOLDS NUMBER (LRN) TURBULENCE MODEL:

- The STAN5 B.L. code was modified to include the LRN $k-\varepsilon$ turbulence model of Lam & Bremhorst as described by Zerkle & Lounsbury [1].
- Includes the following near-wall effects:
  - High freestream turbulence
  - Axial pressure gradient
  - Onset of transition
  - Relaminarization
  - Wall roughness
  - Wall curvature
- Used to compute heat transfer coefficient distributions on turbine airfoil external surfaces.
- Primary limitation:
  - It's a 2-D code in a 3-D environment.

3-D NAVIER-STOKES CODE WITH WALL FUNCTIONS:

- Time-marching finite-volume formulation of the Reynolds-averaged Navier-Stokes equations as described by Turner & Jennions [2,3].
- Includes:
  - Explicit Runge-Kutta flow solver
  - Implicit formulation of the standard $k-\varepsilon$ turbulence model
  - Standard wall functions
  - Transonic flow effects
- Used to simulate high speed flows in turbomachinery passages.
- Limitations:
  - Lacks near-wall physics of the 2-D boundary layer code.
  - For example, lack of boundary layer transition leads to overprediction of loss for some turbomachinery airfoil passages containing significant regions of transitional flow.
3-D NAVIER–STOKES CODE WITH LOW REYNOLDS NUMBER (LRN) TURBULENCE MODEL:

• The LRN $k-\varepsilon$ turbulence model of Lam & Bremhorst was implemented in the 3-D Navier–Stokes code as described by Dailey, Jennions and Orkwis [4].

• Addition of the LRN turbulence model improved the prediction of loss for transitional flows.

• Primary limitation:
  - The need for a very fine grid in the near-wall region leads to excessive run times which renders the code impractical for design applications at this time.

FILM COOLING SIMULATION:

• Film cooling at the surface of an HP turbine airfoil is crucial to its life.

• Improvement of the film cooling process would significantly improve turbine performance by reducing the need for cooling air flow.

• CFD simulation could facilitate film cooling development by reducing the need for expensive cascade testing and, more importantly, by giving greater insight into the associated flow physics.

• A CFD simulation of film—cooling tests, which were carried out at the Univ. of Texas by Professors Crawford & Bogard, and their students, is described by Leylek & Zerkle [5].

• These tests are of special interest because the ranges of film cooling parameters are consistent with those typically found in gas turbine airfoil applications.

• The objective was to validate a CFD model of film cooling by comparing numerical and experimental results.
FILM COOLING SIMULATION (CONT'D):

- The model includes:
  - A 3-D, fully-elliptic, Navier–Stokes solution of the coupled flow in the plenum, film hole, and cross-stream regions.
  - An exact representation of the inclined, round, film-hole geometry using a highly-orthogonalized fine grid mesh.
  - The standard $k-\varepsilon$ turbulence model with standard wall functions.

---

**Figure 1.** Essential features of experimental film cooling configuration showing overall extent of computation domain and coordinate system.
FILM COOLING SIMULATION (CONT'D):

Summary of Results:
- The flowfield is dominated by a strong three-way coupling between the plenum, film-hole, and cross-stream regions.
- Flow within the film hole is extremely complex, with counter-rotating vortices and local jetting effects.
- A comparison of computed and experimental film effectiveness on the plate surface indicates that the simulated coolant jet is not spreading as fast as experimental results.

Conclusions:
- There is excellent qualitative agreement between the numerical and experimental results.
- However, the lack of lateral spreading of the coolant is caused by the inability of the k-ε turbulence model to cope with non-uniform rates of diffusion in different directions.
- Improved accuracy requires an anisotropic turbulence model.

Figure 14. Lateral variation of adiabatic effectiveness from computations and experiments for M=0.5 at five streamwise stations.
TURBULATED PASSAGE SIMULATION:

- Modern high-performance turbine blades are cooled by internal radially-rotating serpentine passages.
- The airflowing through these passages is exposed to very large Coriolis and centripetal body forces which induce strong secondary flows and buoyant effects.
- These effects tend to increase heat transfer coefficient on the trailing face of an up-pass, but decrease it on the leading face.
- Turbulators are added to the passage walls in order to enhance their cooling effectiveness.
- The primary objective of blade cooling development is to determine turbulator and passage configurations which can influence the secondary flows to achieve a uniformly high heat transfer coefficient, but within pressure-drop constraints.
- Rotating-passage rig tests are expensive, and it is very difficult to achieve high-quality data in the range of engine operating parameters.

TURBULATED PASSAGE SIMULATION (CONT'D):

- Therefore, CFD could facilitate blade cooling development by simulating new cooling configurations at real engine operating conditions.
- An exploratory investigation of CFD simulation in turbulated blade cooling passages is described by Prakash & Zerkle [6].
- Conclusions are:
  - The flow fields in turbulated blade cooling passages are very complex, and desired accuracy requires advanced turbulence models.
  - An LRN model is needed near turbulated walls in the case of low passage Reynolds number.
  - An anisotropic turbulent model is needed in the case of large blockage ratio (rib height to passage diameter).
  - Practical LRN and anisotropic models are not yet available.
### Blockage Ratio

<table>
<thead>
<tr>
<th>Blockage Ratio</th>
<th>Anisotropic and Multiple Length Scale Effects</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Low Blockage Ratio</strong></td>
<td></td>
</tr>
<tr>
<td>Low Reynolds Number</td>
<td>Need a low Reynolds number model</td>
</tr>
<tr>
<td></td>
<td>May not need a Reynolds stress model; i.e., an isotropic model may suffice</td>
</tr>
<tr>
<td>High Reynolds Number</td>
<td>May not need a low Reynolds number model</td>
</tr>
<tr>
<td></td>
<td>May not need a Reynolds stress model</td>
</tr>
<tr>
<td><strong>High Blockage Ratio</strong></td>
<td></td>
</tr>
<tr>
<td>Low Reynolds Number</td>
<td>Need a low Reynolds number model</td>
</tr>
<tr>
<td></td>
<td>May need a Reynolds stress model</td>
</tr>
<tr>
<td>High Reynolds Number</td>
<td>May not need a low Reynolds number model</td>
</tr>
<tr>
<td></td>
<td>May need a Reynolds stress model</td>
</tr>
</tbody>
</table>

**Present Computations:**
- Model (Isotropic) with 
- Wall Functions
OVERALL CONCLUSIONS:

- Application of the standard k-ε turbulence model with wall function is not adequate for accurate CFD simulation of aerodynamic performance and heat transfer in the primary gas path of a gas turbine engine.

- New models are required in the near-wall region which include more physics than wall functions. The two-layer modeling approach appears attractive because of its computational economy.

- In addition, improved CFD simulation of film cooling and turbine blade internal cooling passages will require anisotropic turbulence models.

- New turbulence models must be practical in order to have a significant impact on the engine design process.

- A coordinated turbulence modeling effort between NASA centers would be beneficial to the gas turbine industry.

LIST OF REFERENCES:


Objective

- Evaluate turbulence models for integrated aircraft components such as the forebody, wing, inlet, diffuser, nozzle, and afterbody

Approach

- Integrate turbulence models into existing Navier–Stokes program maintaining zonal philosophy
- Introduce corrections to baseline turbulence models to account for additional affects such as compressibility or separation
- Develop algorithmic improvements for better numerical stability and robustness
- Compare the strengths and weaknesses of turbulence models
- Determine applicability of algebraic, one-equation, and two-equation turbulence models for typical complex flows and geometries
Turbulence Modeling Capabilities

- **Algebraic Models**
  - Cebeci-Smith boundary layer model
  - Baldwin–Lomax boundary layer model
  - P. D. Thomas shear layer model

- **One-Equation Models**
  - Baldwin–Barth
  - Spalart–Allmaras

- **Two-Equation Models**
  - High Reynolds number $k - \epsilon$
  - Low Reynolds number $k - \epsilon$ (Jones–Lauder, Speziale, Chien, Lam–Bremhorst, So, and Huang–Coakley)
  - Wilcox $k - \omega$
  - Menter baseline and shear-stress transport blended $k - \omega/k - \epsilon$

Navier–Stokes Time–Dependent Algorithm
NASTD

- **Euler/Navier–Stokes Equations**
  - Laminar or Turbulent
  - Ideal Gas, Thermally Perfect Air, Equilibrium or Nonequilibrium Chemistry

- **Finite Volume Formulation**
  - Roe and Coakley Flux Difference Split Schemes, Optional TVD Schemes

- **Solution Update Procedure**
  - Approximate Factorization
  - Runge–Kutta Time Stepping
  - Iterative Space Marching (PNS)

- **Geometric Capabilities/Generalizations**
  - Zonal Capabilities and Flexible Boundary Conditions
  - Grid Sequencing
  - Overlapping Grids

- **Turbulence Models**
  - Cebeci-Smith, Baldwin–Lomax and P. D. Thomas Algebraic Models
  - Baldwin–Barth and Spalart–Allmaras One-Equation Models
  - Six Low Reynolds Number $k - \epsilon$ Models
  - $k - \omega$ and Menter blended $k - \omega/k - \epsilon$ Models
Selected Applications

- Transonic Supercritical Airfoil
- Three-Element High-Lift System
- Single Slot 2-D Ejector Nozzle
- Confluent Mixer
- Highly Offset 3-D Diffuser

Modifications to Production Term

Default calculation of production:

\[ P_k = \frac{\bar{\mu}_t}{Re} \left[ \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)^2 - \frac{2}{3} \left( \frac{\partial \bar{u}_k}{\partial x_i} \right)^2 \right] - \frac{2}{3} \bar{\rho} k \frac{\partial \bar{u}_k}{\partial x_k} \]

Vorticity used in production:

\[ P_k^* = \frac{\bar{\mu}_t}{Re} |\omega|^2 \]

Production limiter used:

\[ P_k^* = \min(P_k, 20D_k) = \min(P_k, 20c_2 \rho k Re) \]
Effect of Production Limiter for the Chien $k-\varepsilon$ Model

RAE Airfoil Analysis, Turbulent Viscosity Contours
Mach = 0.725, $\alpha = 2.55$ deg., Re = 6.5 Million

RAE Airfoil Analysis
$M_\infty = 0.725, \alpha = 2.55^\circ, Re = 6.5$ Million
Effect of Freestream Turbulence Level on Surface Pressure
Chien $k-\varepsilon$ Turbulence Model
RAE Airfoil Analysis
$M_\infty = 0.725, \alpha = 2.55^\circ, Re = 6.5 \text{ Million}$

Production Limiter Used
Chien $k-\epsilon$ Turbulence Model

RAE Airfoil Analysis, Turbulent Viscosity Contours
Mach = 0.725, $\alpha = 2.55$ deg., Re = 6.5 Million
RAE Airfoil Analysis, Mach Contours
Mach = 0.725, \( \alpha = 2.55 \) deg., Re = 6.5 Million

Effect of Turbulence Model on Surface Pressure

\( M_\infty = 0.725, \alpha = 2.55^\circ, Re = 6.5 \) Million
RAE Airfoil Analysis

$M_\infty = 0.725$, $\alpha = 2.55^\circ$, $Re = 6.5$ Million

Effect of Turbulence Model on Surface Pressure
NASTD Solution of MDA Three-Element High-Lift System

$M = 0.2, \text{ AOA} = 16.21$ Baldwin-Barth

More Accurate Solutions Have Been Obtained With One-Equation Spalart-Allmaras Turbulence Model

Skin Friction Coefficients on the upper Surfaces

Velocity Profile at Station 1 on the Wing ($M = 0.2, \alpha = 16.21$)
Four-Zone Grid for an Ejector Nozzle

Single Slot Ejector Analysis
NPR=14, Pts/Ptp=.34
Mach Number Contours from Several Turbulence Models

P.D. Thomas
Baldwin-Barth
Chien
Spalart-Allmaras
Single Slot Ejector Analysis

\[ NPR = 14, \frac{P_{ts}}{P_{tp}} = 0.34 \]

Eddy Viscosity from Several Turbulence Models

<table>
<thead>
<tr>
<th>Model</th>
<th>( W_s/W_p )</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>0.1010</td>
<td>-</td>
</tr>
<tr>
<td>Thomas/Baldwin-Lomax</td>
<td>0.1108</td>
<td>+9.7</td>
</tr>
<tr>
<td>Baldwin-Barth</td>
<td>0.1129</td>
<td>+11.8</td>
</tr>
<tr>
<td>Spalart-Allmaras</td>
<td>0.1146</td>
<td>+13.5</td>
</tr>
<tr>
<td>Chien ( k - \epsilon )</td>
<td>0.1168</td>
<td>+15.6</td>
</tr>
<tr>
<td>Jones-Lauder ( k - \epsilon )</td>
<td>0.1126</td>
<td>+11.5</td>
</tr>
<tr>
<td>Speziale ( k - \epsilon )</td>
<td>0.1127</td>
<td>+11.6</td>
</tr>
<tr>
<td>So ( k - \epsilon )</td>
<td>0.1148</td>
<td>+13.7</td>
</tr>
<tr>
<td>Huang-Coakley ( k - \epsilon )</td>
<td>0.1112</td>
<td>+10.1</td>
</tr>
</tbody>
</table>
Single Slot Ejector Nozzle

Surface Static Pressure Comparison with Experimental Data

$NPR = 14.0, \frac{P_t}{P_p} = 0.34$

Offset Diffuser Analysis

$Ae/At=1.6, L/D=4.5$, Design Pressure Ratio

Surface Pressure and Computational Mesh
Offset Diffuser Analysis
Ae/At=1.6, L/D=4.5, Design Pressure Ratio

Comparison of Engine Face Total Pressures

Baldwin-Lomax  Baldwin-Barth  Spalart-Allmaras

Pt/Pto

1.0  .95  .90  .85  .80

Experiment  Chien
Offset Diffuser Analysis
Lower Centerline Surface Static Pressure
$A_e/A_t = 1.6, L/D = 4.5$, Design Pressure Ratio

Offset Diffuser Analysis
Upper Centerline Surface Static Pressure
$A_e/A_t = 1.6, L/D = 4.5$, Design Pressure Ratio
Three-Dimensional Highly Offset Diffuser

$A_e/A_t = 1.6$, $L/D = 4.5$, Design Pressure Ratio

Comparison of Engine Face Parameters

<table>
<thead>
<tr>
<th>Model</th>
<th>$P_{avg}/P_{\infty}$</th>
<th>$P_{min}/P_{\infty}$</th>
<th>$P_{max} - P_{min}$/ $P_{avg}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>0.958</td>
<td>0.890</td>
<td>0.114</td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>0.936</td>
<td>0.708</td>
<td>0.292</td>
</tr>
<tr>
<td>Baldwin-Barth</td>
<td>0.944</td>
<td>0.735</td>
<td>0.265</td>
</tr>
<tr>
<td>Spalart-Allmaras</td>
<td>0.955</td>
<td>0.860</td>
<td>0.140</td>
</tr>
<tr>
<td>Chien k - $\epsilon$</td>
<td>0.970</td>
<td>0.894</td>
<td>0.106</td>
</tr>
<tr>
<td>Jones-Launder k - $\epsilon$</td>
<td>0.966</td>
<td>0.896</td>
<td>0.104</td>
</tr>
<tr>
<td>So k - $\epsilon$</td>
<td>0.975</td>
<td>0.888</td>
<td>0.112</td>
</tr>
</tbody>
</table>

G.E. Confluent Mixer

Surface Grid and Predicted Temperature Variations

Baldwin-Barth Model Prediction

Every other grid point shown for clarity
Centerline Eddy Viscosity Contours

\[ \frac{\mu_t}{\mu_\infty} \]

Chien k-epsilon

Baldwin-Barth

Spalart-Allamaras

Centerline Temperature Contours

\[ T(\text{F}) \]

Chien k-epsilon

Baldwin-Barth

Spalart-Allamaras
Comparison of Throat Total Temperatures

GE Slot Cooled Nozzle, Confluent Mixer

Surface Temperature Distributions,

\[ TGX = \frac{T_I - T_{\text{cold}}}{T_{\text{hot}} - T_{\text{cold}}} \]

---

To (F)

700.0
600.0
500.0
400.0
300.0
200.0
100.0

---

Baldwin-Barth Model
k-Epsilon Model
Spalart-Allamaras
Experiment (Average)
Summary of Turbulence Modeling at McDonnell Douglas Aerospace

- The one-equation models have replaced the algebraic models as the baseline turbulence models.
- The Spalart-Allmaras one-equation model consistently performs better than the Baldwin-Barth model, particularly in the log-layer and free shear layers. Also, the Spalart-Allmaras model in not grid dependent like the Baldwin-Barth model.
- No general turbulence model exists for all engineering applications.
- The Spalart-Allmaras one-equation model and the Chien $k - \epsilon$ models are the preferred turbulence models.
- Although the two-equation models often better predict the flowfield, they may take from two to five times the CPU time.
- Future directions are in further benchmarking the Menter blended $k - \omega/k - \epsilon$ and algorithmic improvements to reduce CPU time of two-equation model.
OUTLINE

• Geometry and flow configuration

• Effect of $y^+$ on heat transfer computations

• Standard and Extended $k$-$\varepsilon$ turbulence model results with wall function

• Low-Re model results (the Lam-Bremhorst model without wall function)

• A criterion for flow reversal in a radially rotating square duct

• Summary

Fig. 1-Illustration of geometry and physics of flow
TWO-EQUATION TURBULENCE MODELS

\[ \mu_t = f_\mu \, C_\mu \rho \, k \frac{\partial k}{\partial x} \]

\[ \frac{D(\rho k)}{Dt} = \frac{3}{\partial x_i} \left( P_k \frac{\partial k}{\partial x_i} \right) + \rho (G_k - \varepsilon) \]

\[ \frac{D(\rho e)}{Dt} = \frac{3}{\partial x_i} \left( P_\varepsilon \frac{\partial e}{\partial x_i} \right) + f_1 \, C_1 \frac{\varepsilon}{k} \rho G_k - f_2 \, C_2 \, \rho \, \frac{\varepsilon^2}{k} + C_3 \, \rho \, \frac{G_k^2}{k} \]

where \( G_k = \frac{\mu_t}{\rho} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \); \( C_\mu = 0.09 \)

Standard \( k-\varepsilon \) model:
\( P_k = 1.0, \ P_\varepsilon = 1.3, \ C_1 = 1.44, \ C_2 = 1.92, \ C_3 = 0.0, \ f_1 = 1.0, \ f_2 = 1.0, \) and \( f_{\varepsilon} = 1.0 \)

Extended \( k-\varepsilon \) model:
\( P_k = 0.89, \ P_\varepsilon = 1.15, \ C_1 = 1.15, \ C_2 = 1.9, \ C_3 = 0.25, \ f_1 = 1.0, \ f_2 = 1.0, \) and \( f_{\varepsilon} = 1.0 \)

Lam-Bremhorst low-Re model:
\( P_k = 1.0, \ P_\varepsilon = 1.3, \ C_1 = 1.44, \ C_2 = 1.92, \ C_3 = 0.0, \ f_1 = (1 + 0.05 F_\mu)^3, \ f_2 = 1 - e^{-R_k^2}, \)
and \( f_{\varepsilon} = (1 - e^{-0.0165 R_k}) \left( 1 + 20.5 / R_k \right) \), where \( R_k = k^{1/2} \rho / \mu \) and \( R_i = k^2 \rho / \mu \varepsilon \)

[Figure 2(a)-Effect of \( y^+ \) and grid size on \( Nu \) computation]
Fig. 2(b)-Effect of $y+$ and grid size on Nu computation

Fig. 2(c)-Effect of $y+$ on the cross-stream velocity near leading wall
Fig. 2(d)-Effect of $y^+$ on the cross-stream velocity near trailing wall

Fig. 2(a)-Comparison of model results with data (Morris & Ghavami-Nasr, 1991)
Fig. 3(b)-Comparison of model results with data (Morris & Ghavami-Nasr, 1991)

Fig. 4(a)-Comparison of model results with data on leading wall
Trailing Wall

Ro = 0.12
ΔT/Tw = 0.13
Re = 25000
R/d = 49

Fig. 4(b)-Comparison of model results with data on trailing wall

Leading Wall

Ro = 0.24
ΔT/Tw = 0.13
Re = 25000
R/d = 49

Fig. 4(c)-Comparison of model results with data on leading wall
Fig. 4(d)-Comparison of model results with data on trailing wall

Fig. 5-Comparison of the two model results at high Ro and high density ratio

Fig. 6(a)-Plots of Nu-ratio and k-ratio from the two models

Fig. 6(b)-Plots of Nu-ratio and k-ratio from the two models
Fig. 7(a)-Comparison on both leading and trailing walls

Fig. 7(b)-Comparison on both leading and trailing walls
Fig. 8(a) - Comparison of Eke and low-Re results on leading wall
(Re=10000, Ro=.088, Case B)

Fig. 8(b) - Comparison of Eke and low-Re results on leading wall
(Re=5000, Ro=.176, Case B)
Fig. 8(c)-Comparison of Eke and low-Re results on leading wall (Re=5000, Ro=.176, Case C)

Fig. 8(d)-Comparison of low-Re results with data (Re=2500, Ro=.352, Case B)
Table 1 Prediction of Flow Reversal Near the Leading Wall

<table>
<thead>
<tr>
<th>Ro</th>
<th>DT/Tw</th>
<th>RAl</th>
<th>Re</th>
<th>Gr/Re²</th>
<th>Flow Reversal ?</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.12</td>
<td>0.07</td>
<td>49</td>
<td>25000</td>
<td>0.05</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td>0.13</td>
<td></td>
<td></td>
<td>0.09</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td>0.23</td>
<td></td>
<td></td>
<td>0.16</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td>0.36</td>
<td></td>
<td></td>
<td>0.26</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td>0.48</td>
<td></td>
<td></td>
<td>0.34</td>
<td>Yes</td>
</tr>
<tr>
<td></td>
<td>0.07</td>
<td>196</td>
<td>300</td>
<td>0.20</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.30</td>
<td>Yes</td>
</tr>
</tbody>
</table>

| 0.24| 0.07  | 33  | 25000| 0.13   | No              |
|     | 0.13  | 49  |     | 0.20   | No              |
|     | 0.16  | 196 |     | 0.77   | Yes             |
|     | 0.23  | 300 |     | 1.18   | Yes             |
|     | 0.07  | 49  |     | 0.36   | Yes             |
|     | 0.13  |     |     | 0.45   | Yes             |
|     | 0.23  |     |     | 0.65   | Yes             |
|     | 0.07  |     |     | 0.20   | No              |
|     | 0.13  |     |     | 0.36   | Yes             |
|     | 0.23  |     |     | 0.65   | Yes             |

| 0.34| 0.13  | 49  | 25000| 0.73   | Yes             |
|     | 0.16  |     |     | 0.91   | Yes             |
|     | 0.23  |     |     | 1.30   | Yes             |

| 0.48| 0.13  | 49  | 25000| 1.45   | Yes             |

**SUMMARY**

1. Near-wall grid size has a significant effect on the heat transfer calculations when the "wall function" treatment is used. Numerical experiment on the data of Morris et al. (1991) suggests that a y+ value in the range of 12 to 42 or so yields more accurate results.

2. The extended k-ε turbulence model, while yielding heat transfer results virtually the same as those of standard k-ε model for low rotation-number flows, provides an improvement over the standard k-ε model by up to 15% or so in heat transfer predictions for high rotation number flows.

3. Wall-function k-ε models predict lower (than data) heat transfer at the trailing wall and higher at the leading wall. The need to properly represent the effect of rotation in the k-ε model equations is realized.

4. The low-Reynolds number model utilizes a large number of cells and the convergence rate is very slow in comparison to the high-Reynolds number model using wall function. It is difficult and expensive to obtain a well converged solution with the low-Re turbulence model.
5. The poor agreement of the low-Re model results with the data makes the low-Re model an unattractive choice for heat transfer computations in rotating radial outward flow at high Rotation number (> 0.24) and high-Reynolds number (25000).

6. The extended version of high-Reynolds number turbulence model in conjunction with wall function yields satisfactory results for flows with isothermal walls as well as uneven wall temperatures. The agreement is within 5-25% of the data with uneven wall temperatures for flows at Reynolds numbers 10000 or higher.

7. For flows at Reynolds number 5000 or lower, the low-Re model predictions are better, especially for the case of uneven wall temperature conditions.

8. The centrifugal buoyancy may cause a flow reversal near the leading wall depending upon the geometry and flow parameters such as rotation number, temperature ratio, mean radius ratio and Reynolds number. For the square-section channel considered here, a criterion of \( Bo = Gr/Re^2 \) higher than 0.3 is predicted to cause flow reversal near the leading wall for flows at Reynolds number up to 25000.

REFERENCES


TURBULENCE MODELS FOR GAS TURBINE COMBUSTORS

Andreja Brankovic
CFD Group
Pratt & Whitney
West Palm Beach, Florida

F100-PW-200 TURBOFAN ENGINE

PW4000

79

PRECEEDING PAGE BLANK NOT FILMED
GT COMBUSTOR FLOW PHYSICS

• Key issue is flame stabilization by means of recirculating flow of hot gases and chemically-active species to ensure continuous ignition of fresh reactants.

• Three main mechanisms: 1) axial swirling air jet associated with each fuel introduction; 2) sudden expansion of axial swirling jets; 3) blockage due to radial air jets downstream of fuel sources.
TURBULENCE MODELS SURVEYED

• Following models or modifications have been tested at P&W / UTRC using RANS solvers on building block flows:
  1. low-Re models (complex ducts);
  2. RSTM or SMC (complex ducts, swirling and non-swirling dump combustor);
  3. RNG (pipe, backstep, 180 deg duct);
  4. two-layer near-wall model (internal flows, heat transfer);
  5. realizable algebraic stress model (swirling dump combustor);
  6. compressible turbulence (shear layers, compression corner)
  7. steady vs. unsteady-state solver (bluff-body, compression corner)

• Major difficulty occurs with swirling flows, and failure to predict downstream velocity components.

SWIRLING FLOWS

• Benchmark-quality data set provided by Johnson-Roback co-annular combustor with swirl:

  ![](image)

  \( X = 10.2 \text{ cm} \)

  \[
  \frac{r}{R_o} \quad U, \text{ m/sec} \quad V, \text{ m/sec} \quad W, \text{ m/sec}
  \]

• Poor agreement of CFD and data highlights need for improved upstream BC specification (swirler geometry), 3-D, unsteady analysis. Even SMC models fail to reproduce downstream velocity profiles.
UNSTEADINESS AND FLOW FIELD RESOLUTION

- RANS solvers can predict flow coherence (vortex shedding) when run in an unsteady mode with small Δt.

- Same flow field computed in steady-state sense gives completely unusable results.

- Example: V-gutter flow, computed by Durbin (1994):

  ![Diagram of V-gutter flow](image)

  Ratio of characteristic frequencies (estimated)
  \[ \frac{f_T}{f_0} = 225 \]

- RANS solvers cannot predict flow oscillations at frequencies near characteristic turbulence frequency.

- Example: Unsteady comp. corner flow of Dolling and Or (1983):

  ![Diagram of comp. corner flow](image)

  Frequency ratio:
  \[ \frac{f_T}{f_0} = 7.7 \]

  - Separation bubble oscillations (at resonant frequency) not resolved by RANS solver.

  - Limitations of steady-state and unsteady-state RANS solvers set by flow characteristic time scales. True time-accurate solvers (LES, DNS) needed for prediction of all relevant phenomena.

82
TURBULENT COMBUSTION MODELING

- Eddy Dissipation Concept Model, together with reaction exclusion regions, capable of prediction gross flow features at near LBO conditions (Sturgess et al., 94-GT-433)

- EDC model, however, fails to predict flame attachment at rich conditions

```
K_{f_turb} = \int_0^b k_f(T) P(T) \, dT \\
K_{f_Lam} = \frac{k_f(T)}{k_f(\bar{T})} \\
T_{min} = \max(\bar{T} - \phi \sqrt{T''}, T_{low}) \\
T_{max} = \min(\bar{T} + \phi \sqrt{T''}, T_{high})
```

- Assumed-Pdf method of Girimaji (LaRC Workshop, 1991) used with non-equilibrium kinetics model.

- Example: \( N + O_2 \rightarrow NO + O \) in extended Zeldovich model

- Results dependent on \( T_{low}, T_{high}, \phi \), modeling of \( h\h \) transport equation, etc.

- More testing needed
PRESENT STATUS OF COMBUSTOR MODELING

- Corsair (Ryder, P&W) unstructured, unsteady flow solver

- Example: Time-dependent combustor flow using engineering boundary conditions, compressor exit to turbine inlet

- Code currently includes standard $k$-$\epsilon$ and EBU combustion model. Additional capabilities being added under "Subsonic Emissions and Combustor Design Code" program with NASA LeRC.

PRESENT STATUS OF COMBUSTOR MODELING

- Example: Structured flow solver solution of Task 200 LBO Research Combustor:

- $k$-$\epsilon$ turbulence model
- EBU combustion model for propane fuel
- 285,000 elements
PRESENT STATUS OF COMBUSTOR MODELING

- Example: Unstructured flow solver solution of Task 200
  LBO Research Combustor:

- k-ε turbulence model
- EBU combustion model for propane fuel
- Approx. 300,000 elements

TURBULENCE RESEARCH NEEDS

- Modelling: Applications / validations of currently available combustion models (β-pdf, Monte Carlo pdf, laminar flamelet) to complex combustor geometry with jet fuel kinetics.

- Flow Physics: Accurate numerical description of mechanisms responsible for flame holding, local extinction (LES, DNS); contrast cold flows with heat release flows.

Entrainment of unburned fuel in the recirculation region

85
CURRENT COMBUSTION SYSTEM CFD MODELING CAPABILITIES AT GEAE PROVIDED BY THE CONCERT CODE

KEY FEATURES INCLUDE:

- FINITE VOLUME, PRESSURE CORRECTION FORMULATION
- SECOND ORDER ACCURATE QUICK NUMERICS
- SINGLE STRUCTURED BODYFITTED GRID
- CONVENTIONAL K–E TURBULENCE MODEL WITH LOG WALL FUNCTIONS

AVAILABLE COMBUSTION MODELS INCLUDE:
- SINGLE SCALAR PRESUMED SHAPE PDF (FAST CHEMISTRY)
- TWO SCALAR PRESUMED SHAPE PDF (REACTION PROGRESS VARIABLE)
- TWO STEP EDDY BREAKUP (ARRHENIUS KINETICS)
- ZELDOVICH THERMAL NOx MECHANISM (FORWARD AND REVERSE REACTIONS)

BOTH 2D/AXISYMMETRIC AND FULLY 3D VERSIONS AVAILABLE AND IN DAY TO DAY USE

CURRENTLY HAVE A USER BASE OF OVER 20 ENGINEERS AT GEAE AND GE-CRD

TYPICALLY APPLIED TO PREDICT COMBUSTOR PERFORMANCE INCLUDING:
- EMISSIONS (CO, HC, AND THERMAL NOx), COMBUSTION EFFICIENCY
- EXIT GAS TEMPERATURE RADIAL PROFILE AND PATTERN
- GENERAL FLOW FIELD CHARACTERISTICS
CONCERT DEVELOPMENT HISTORY

EFFORT INITIATED IN 1983

INITIAL PRODUCTION VERSION RELEASED TO GEAE USERS IN 1987

FOCUSED TO PROVIDE HIGHLY PRODUCTIVE ENGINEERING ANALYSIS CAPABILITIES

- GRID GENERATION OPTIMIZED FOR THE SPECIFIC GEOMETRY FEATURES OF THE GAS TURBINE COMBUSTOR
- INCLUDES ROUND DILUTION HOLES, SWIRLER DISCHARGE, AND LINER SLOT FEATURES WITHIN THE GRID
- EASY INTRODUCTION OF INTERNAL BODIES OF COMPLEX GEOMETRY
- WORKSTATION BASED USER FRIENDLY PRE AND POST PROCESSING FUNCTIONS BUILT AROUND THE SOLVER.
- SOLVER HIGHLY OPTIMIZED FOR THE GEAE CRAY C-90 COMPUTER.

TYPICAL 3D MODEL OF A COMBUSTOR UTILIZING A MESH OF ~100,000 POINTS CAN BE GENERATED, RUN, AND POST PROCESSED WITHIN A SINGLE WORKING DAY!

HAS UNDERGONE CONTINUAL DEVELOPMENT TO IMPROVE AND ENHANCE MODELING CAPABILITIES

- CURRENTLY ON VERSION 3 RELEASE

CONCERT CFD MODELING PACKAGE PROVIDES DESIGN ENGINEERS WITH A COST AND TIME EFFECTIVE ANALYSIS TOOL THAT REDUCES DEPENDENCE ON COSTLY COMPONENT RIG TESTING.

COMBUSTION SYSTEM CFD MODELING IN ACTION AT GEAE

SECONDARY SWIRLER AIR IN

PURGE AIR

PRIMARY SWIRLER AIR IN

SWIRLCUP/SPIRAY MODELING
- RECIRCULATION STRENGTHENING
- FLOW FIELD CHARACTERISTICS
- SPRAY DROPLET TRAJECTORIES
- INPUTS FOR 3D COMBUSTOR MODEL

BLEED AIR

DIFFUSER FLOW MODELING
- Pb RECOVERIES AND Pb LOSSES
- FLOW FIELD CHARACTERISTICS

BLEED AIR

SPRAYBAR

FLAMEHOLDERS

AUGMENTOR MODELING
- FLOW FIELD CHARACTERISTICS/MIXING
- GAS TEMPERATURES AND PATTERNS
- EFFICIENCY

MAIN COMBUSTOR MODELING
- FLOW FIELD CHARACTERISTICS/MIXING
- GAS TEMPERATURES AND PATTERNS
- EMISSIONS/EFFICIENCY

FLOW FIELD CHARACTERISTICS_MIXING
- GAS TEMPERATURES AND PATTERNS
- EFFICIENCY

88
## MODELING APPLIED FOR DESIGNING ENGINE COMBUSTION SYSTEMS

<table>
<thead>
<tr>
<th>PRODUCTION ENGINES</th>
<th>DEMONSTRATOR ENGINES</th>
<th>ADVANCED ENGINES</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFM56-5B DUAL ANNULAR</td>
<td>YF120</td>
<td>A/F-X</td>
</tr>
<tr>
<td>GE90</td>
<td>F120</td>
<td>NASA/GE HSCT</td>
</tr>
<tr>
<td>CF6-80C LOW EMISSIONS</td>
<td>XTE45 IHPTET PHASE I DEMO</td>
<td>NASA ASI PRELIMINARY CONCEPTS</td>
</tr>
<tr>
<td>LM1600 DLE</td>
<td>XTE46 IHPTET PHASE II DEMO</td>
<td>DOE/GE ATS</td>
</tr>
<tr>
<td>LM2500 DLE</td>
<td></td>
<td></td>
</tr>
<tr>
<td>LM6000 DLE</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

## MODELING APPLIED TO IMPROVE FUNDAMENTAL UNDERSTANDING

- CFM56-3 and CFM56-5B NOx EMISSIONS CHARACTERISTICS DIFFERENCES
- CFM56-5A EXIT GAS TEMPERATURE PROFILE SHIFT
- F120 PATTERN FACTOR AND RADIAL PROFILE IMPROVEMENT
- LM2500 CO EMISSIONS REDUCTION EFFORT
- CF34 LINER COOLING MOD IMPACT ON CO EMISSIONS
- F110X AUGMENTOR MIXER, SPRAYBAR, FLAMEHOLDER INTERACTION OPTIMIZATION
- F110-400 AUGMENTOR EXHAUST DUCT LINER FAILURE AND FIX INVESTIGATION
CONCERT3D RESULTS FOR CURRENT PRODUCTION COMBUSTORS

CONCERT3D MODEL OF NASA/GE E3 COMBUSTOR

Calculated flow field in plane in line with swirl cups
CONCERT3D vs. RIG DATA COMPARISON FOR NASA/GE E3 COMBUSTOR

(EXIT GAS TEMPERATURE AVERAGED AND MAXIMUM RADIAL PROFILES)

(37/63 PILOT/MAIN STAGE FUEL SPLIT)

NON-DIMENSIONAL EXIT PASSAGE HEIGHT

NON-DIMENSIONAL TEMPERATURE

CONCERT3D MODEL SOLUTION

AVERAGED RADIAL PROFILE

MAXIMUM RADIAL PROFILE

CONCERT3D vs. RIG DATA COMPARISON FOR NASA/GE E3 COMBUSTOR

(NOx EMISSIONS)

MEASURED DATA

CONCERT3D SOLUTION

COMBUSTOR INLET PRESSURE - PSIA
GEAE CONCERT EXPERIENCE:

**CONCERT3D WITH PRESUMED SHAPE PDF/FAST CHEMISTRY MODEL AND THERMAL NOx MODEL DOES WELL AGAINST REAL ENGINE DATA**

**CONCERT3D WITH TWO STEP EDDY BREAKUP MODEL DOES NOT CONSISTENTLY DEMONSTRATE ACCEPTABLE AGREEMENT FOR [CO] AND [HC] EMISSIONS**

**OTHER PERFORMANCE ISSUES NOT AS WELL PREDICTED COMPARED TO PRESUMED SHAPE PDF/FAST CHEMISTRY APPROACH**

SHORTCOMINGS:

**TWO STEP EDDY BREAKUP MODEL NOT ADEQUATE FOR THE REQUIRED LEVEL OF PREDICTIVE ACCURACY**

**FAST CHEMISTRY CANNOT PREDICT [CO], [HC], AND IGNITION, BLOWOUT, AND RELIGHT**

**REQUIRES ACCURATE FINITE RATE CHEMISTRY REPRESENTATION AND MORE ACCURATE TURBULENCE-CHEMISTRY INTERACTION MODELING**

*GE HAS EMBARKED ON THE DEVELOPMENT OF IMPROVED CONCERT MODELING CAPABILITIES*

---

**HYBRID CONCERT CFD / MONTE-CARLO MODELING APPROACH**

**APPROACH ADOPTED FOR THE NEXT RELEASE OF COMBUSTION CFD MODELING CAPABILITY AT GEAE**

RETAINS;

- SINGLE STRUCTURED BODYFITTED GRID
- PRESSURE CORRECTION FINITE VOLUME FORMULATION
- K–E TURBULENCE MODELING WITH LOG WALL FUNCTIONS

INTRODUCES;

- MONTE-CARLO SCALAR PDF TO ADDRESS TURBULENT COMBUSTION
  - SINGLE ATTRIBUTE (CONSERVED SCALAR) FOR FAST CHEMISTRY
  - MULTIPLE SCALARS FOR FINITE RATE CHEMISTRY OF CH4 AND JETA FUELS BASED ON APPROPRIATE REDUCED MECHANISMS

**DEVELOPMENT HAS BEEN UNDERWAY SINCE 1992**

- 3D CODE DEVELOPMENT INITIATED IN MID YEAR 1993

---

92
HYBRID CONCERT CFD / MONTE-CARLO MODELING APPROACH

BETA TESTING INITIATED BEGINNING OF 1994

FOCUSED ON FAST CHEMISTRY CALCULATIONS AND OPTIMIZING COMPUTATIONAL EFFICIENCY

SIGNIFICANT IMPROVEMENT IN COMPUTATIONAL EFFICIENCY ACHIEVED

<table>
<thead>
<tr>
<th></th>
<th>TEST CASE 1</th>
<th>TEST CASE 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>NUMBER OF GRID POINTS</td>
<td>9,261</td>
<td>58,621</td>
</tr>
<tr>
<td>NUMBER OF M/C PARTICLES</td>
<td>216,000</td>
<td>1,500,000</td>
</tr>
<tr>
<td>CPU TIME (CRAY C-90 seconds)</td>
<td>83</td>
<td>5,400</td>
</tr>
<tr>
<td>CONCERT WITHOUT M/C</td>
<td>39,960</td>
<td>187,560</td>
</tr>
<tr>
<td>INITIAL HYBRID CONCERT /MC</td>
<td>1,770</td>
<td>41,400</td>
</tr>
<tr>
<td>OPTIMIZED VERSION</td>
<td>-95.6%</td>
<td>-77.9%</td>
</tr>
<tr>
<td>WALL CLOCK TIMES (seconds) UTILIZING CRAY MULTI-TASKING OPTION</td>
<td>1,500</td>
<td>29,520</td>
</tr>
</tbody>
</table>

RUN TIMES HAVE BEEN REDUCED TO THE POINT WHERE OVERNIGHT TURNAROUND TIMES FOR A TYPICAL 3D COMBUSTOR MODEL ARE POSSIBLE.

93
HYBRID CONCERT CFD / MONTE-CARLO MODELING APPROACH

(INITIAL 3D CALCULATION OF CFM56-3 COMBUSTOR WITH FAST CHEMISTRY)

CALCULATED FLOW FIELD IN PLANE IN LINE WITH INLET SWIRL CUPS

INITIAL CALCULATED RESULTS SHOW A TEMPERATURE FIELD THAT DOES NOT AGREE WELL WITH EXPECTED LEVELS. CALCULATION SHOWS CONSIDERABLY LESS DIFFUSION OF THE SCALAR FIELD (FUEL MIXTURE FRACTION) THAN OBSERVED FROM RIG DATA AND CONCERT CALCULATIONS PERFORMED USING THE PRESUMED SHAPE SCALAR PDF COMBUSTION MODELING APPROACH.

FUTURE WORK PLANNED

- PERFORM CALCULATIONS AGAINST A BENCHMARK REACTING FLOW EXPERIMENT WITH AVAILABLE TEST DATA
  - BLUFF BODY STABILIZED FLAME; (GULATI AND CORREA)

- SYSTEMATICALLY STUDY THE EFFECTS OF SCHMIDT NUMBER AND OTHER PARTICLE TRACKING PARAMETERS ON THE FAST CHEMISTRY SOLUTION TO IMPROVE AGREEMENT WITH THE DATA

- PERFORM 3D SINGLE AND DUAL ANNULAR COMBUSTOR CALCULATIONS AND COMPARE RESULTS WITH AVAILABLE GEAE DATA BASE

- IMPLEMENT REDUCED CHEMISTRY SCHEMES (MULTIPLE SCALARS) TO PERFORM FINITE RATE CHEMISTRY CALCULATIONS
  - PREDICT [CO], [HC], AND [NOx] EMISSIONS

- RELEASE CODE FOR PRODUCTION USE AT GEAE
  - FAST CHEMISTRY BY END OF FIRST QUARTER OF 1995
  - FINITE RATE CHEMISTRY BY END OF THIRD QUARTER OF 1995
FUTURE MODELING DIRECTIONS

FOCUSED ON IMPROVING THE PREDICTIVE ACCURACY FOR ALL KEY COMBUSTOR PERFORMANCE ISSUES TO LEVELS THAT WOULD ELIMINATE THE NEED FOR COMPONENT RIG DEVELOPMENT TESTING

1970's / 1980's

YEARS

1990's

MONTHS

20??

WEEKS

FUTURE MODELING DIRECTIONS

INDUSTRY WILL LOOK INCREASINGLY TO THE ACADEMIC COMMUNITY (UNIVERSITIES AND NATIONAL LABS) TO DEVELOP THE NEEDED MODELING IMPROVEMENTS

INDUSTRY MUST PROVIDE THE GUIDANCE AS TO WHAT IS NEEDED

FUTURE GENERATION MODELS MUST;

- PROVIDE MORE RIGOROUS REPRESENTATION OF COMPLEX PHYSICAL PROCESSES
- BE COST EFFECTIVE AS A ROUTINE APPLIED DESIGN/ANALYSIS TOOL
- RETAIN USER FRIENDLY CHARACTERISTICS
- PROVIDE THE LEVEL OF ACCURACY AND CAPABILITIES DEMANDED OF IT

COMPUTING PLATFORM CAPABILITIES ARE ADVANCING AT A RAPID PACE

THE PRACTICALITY OF ADVANCED MODELS IN INDUSTRY MAY NOT BE TOO FAR INTO THE FUTURE

TIME TO START NOW ON DEVELOPMENT OF THE ADVANCED MODELS OF THE FUTURE INTO PRACTICAL TOOLS TO HAVE THEM READY FOR USE WHEN THE REQUIRED COMPUTING PLATFORMS BECOME AVAILABLE IN INDUSTRY
The need:
* Prandtl-mixing-length models require knowledge of distance from nearby walls AND between walls (eg Nikuradze formula)
* Many low-Re models require the distance from nearby walls
* In spaces "cluttered" with solids (eg electronics cooling), calculation of distances and gaps has, in the past, been time-consuming.

The solution:
* This contribution computes WDIS and WGAP (the required quantities) by solving: \( \text{div grad } L = -1 \)

Outline of the theory

Obviously L values which satisfy this equation will be proportional to the distance from the wall at points which are close to it. The question is: what is the proportionality constant?

The constant depends also on the distance across the inter-solid space, which however is the other unknown which it is desired to determine.

The practice adopted by the author is to deduce both the required quantities, WDIS the distance from the wall, and WGAP the distance between walls (whatever these quantities may mean in "cluttered spaces"), from the an algebraic function of the local values of L and its gradient.

The results

The formula employed gives exact results for situations where WDIS and WGAP have unequivocal meanings, namely for the space between two parallel plates or within a long circular-sectioned pipe; and it gives plausible results for more complex cases.

The equation for L, with the appropriate boundary conditions, is of course very easy to solve by numerical means; so WDIS and WGAP can be quickly computed before the flow simulation starts.

The use of the method is illustrated by a PHOENICS calculation for a geometry involving two boxes, a connecting arc, an inlet and an outlet. It was performed by I Poliakov and S Semin, of CHAM, to whom the author's thanks are due.
The need:
* In "cluttered" regions, the between-solid distances are too often too small for fine-grid resolution.
* Reynolds numbers are usually low, at least in some places.
* A model is needed which gives plausible results in these circumstances AND fits experimental data for better-studied ones.

The solution:
* The LVEL model of PHOENICS gets local effective viscosities from the analytical nu-versus-up relation which fits the laminar, transitional & full-turbulent ranges very well. Only local velocity and WDIS (wall distance) are needed.

The u-plus versus y-plus formula of Spalding (1961) is employed namely:
\[
y^+ = u^+ + \frac{1}{E} \left[ \exp(Ku^+) - 1 - Ku^+ - (Ku^+)^{2/2} \right. \\
\left. - (Ku^+)^{3/6} - (Ku^+)^{4/24} \right]
\]
which implies the formula for dimensionless effective viscosity:
\[
v^+ = 1 + \frac{K}{E} \left[ \exp(Ku^+) - 1 - Ku^+ - (Ku^+)^{2/2} \right.
\left. - (Ku^+)^{3/6} \right]
\]
With the wall-distance and the velocity known at every point, the effective viscosity can also be computed at every point.

The method is valid for the whole range of Reynolds numbers; but it is best supplemented by a low-Re "v+-collapse" formula.

The LVEL model gives the well-known experimental results for simple circumstances, such as flow in pipes and between parallel plates; and it gives plausible results for more complex cases.

The use of the method is illustrated by a PHOENICS calculation of the flow and heat transfer in a small part of a large space cluttered with solids which participate in the heat-transfer process.

The method is the only plausible and practicable one known to the author for handling heat transfer in electronics-cooling problems, because of the excessive grid-fineness requirements of low-Reynolds-number k-epsilon extensions.
RECENT PROGRESS IN THE JOINT VELOCITY-SCALAR PDF METHOD

M.S. Anand
Allison Engine Company
Indianapolis, Indiana

- TURBULENCE
- REACTION (treatment, kinetic schemes, emissions)
- TURBULENCE/CHEMISTRY INTERACTIONS
- ATOMIZATION
- SPRAY EVAPORATION

SIMULATION ISSUES:
- NUMERICS (accuracy, convergence)
- GEOMETRY (body-fitted grids, unstructured grids)
- COMPUTATIONAL RESOURCES (Time, Storage)
JOINT VELOCITY-SCALAR PDF METHOD

SIGNIFICANT MILESTONES AND RECENT PROGRESS

- 2-D and 3-D time dependent flows (with finite-volume method)

- Stochastic dissipation model development and validation

- 2-D Elliptic flows (mean pressure algorithm), swirling flows
  (Anand et al. 1989, 1993)

- Spray treatment
  (Anand 1990)

- Manifold methods for reaction kinetics

- Solve Poisson equation for mean pressure:
  \[
  \frac{\partial^2 <p>}{\partial x_i \partial x_j} = -\frac{\partial^2}{\partial x_i \partial x_j} <pU>_{ij}
  \]

- Satisfy continuity by solving for velocity correction potential, velocity correction:
  \[
  \frac{\partial^2 \phi}{\partial x_i \partial x_j} = -\frac{\partial}{\partial x_i} <pU>_{ij} \quad \Delta U_i = \frac{1}{<p>} \frac{\partial \phi}{\partial x_i}
  \]

- Solution algorithm is consistent with B-spline representation of mean fields

- Same discretized form: \( A \hat{\phi} = b \)

- \( A \) is a banded matrix, constant and same for both \(<p>\) and \( \phi \)

- LU decomposition only once

- Special band solver economizes storage and computational effort

- Judicious implementation of the algorithm results in significant economy in computer resource requirement
TURBULENT COMBUSTION MODELING ISSUES
(FOR GAS TURBINE COMBUSTORS)

- Most promising method for turbulent reacting flows

**ATTRIBUTES OF DIFFERENT PDF METHODS**

<table>
<thead>
<tr>
<th>Method</th>
<th>Attributes</th>
<th>Limitations/shortcomings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Joint PDF of $\phi$</td>
<td>Reaction treated exactly</td>
<td>Assumes gradient-diffusion, Does not give velocity field</td>
</tr>
<tr>
<td>Joint PDF of $U$ and $\phi$</td>
<td>Reaction exact, Convection (mean and turbulent) exact, Variable-density effects exact</td>
<td>Needs $\varepsilon$ equation (or equivalent)</td>
</tr>
<tr>
<td>Joint PDF of $U$, $\phi$, and $\omega$</td>
<td>... In addition Provides complete closure, Treats turbulent streams of different scales, Can account for effects of large scale structures</td>
<td>Turbulence/chemistry interactions not fully simulated</td>
</tr>
</tbody>
</table>

**PDF CALCULATIONS FOR A RECIRCULATING FLOW**
(Anand et al. 1989)

[Diagram showing PDF calculations for a recirculating flow]

- Backward-facing step
- Pronchick and Kline (1983)

Storage: 1.3 Mwords
CPU Time: 6 min, Cray YMP
STOCHASTIC DISSIPATION MODEL

- Provides complete closure of the PDF equation (joint velocity-frequency-scalar)
- More realistic than a mean dissipation model. Dissipation (rather, turbulent frequency) is also a random variable and included in the joint PDF.
- Treats multiple scales in the flow
- Accounts for internal intermittency
- Accounts for effects of large scale structures, and influence of origin and history of the fluid particles

\[
\begin{align*}
\dot{\omega}^* &= -\omega^* \langle \omega \rangle (S_{\omega} + C_v \Omega) \, dt + \langle \omega^2 \rangle h \, dt + \omega^* (2C_x \langle \omega \rangle \sigma^2)^{1/2} \, dW \\
\end{align*}
\]

- SWIRLING FLOWS

- No theoretical limitations
- Additional production terms due to non-zero mean swirl velocity
- Additional terms in calculating the mean pressure (or mean pressure gradients)
  - Boundary layer flows:
    > radial pressure gradient
    > axial pressure gradient also included
  - Elliptic flows
    > additional terms in the Poisson equation for pressure

- Validation of the stochastic dissipation model and first calculation of swirling flows with the joint PDF method (Anand et al. 1993)
JOINT PDF CALCULATIONS FOR SWIRLING FLOWS

COMPARISON WITH REYNOLDS-STRESS MODEL RESULTS AND ASSESSMENT OF GRADIENT DIFFUSION MODELING

RS MODEL:
\[
\langle u'v' \rangle = C_s \frac{k}{\langle \varepsilon \rangle} \frac{2 \langle u'^2 \rangle}{\partial r}
\]

\[ C_s = 0.22 \]
SPRAY CALCULATIONS
(Anand 1990)

- Advanced spray models (stochastic Lagrangian, Monte Carlo) naturally compatible with the joint PDF method
- Assumptions about turbulent kinetic energy partition avoided
- Effects of gas phase turbulence structure (velocity cross-correlation) included

105 micron glass beads, NASA HOST C data

Computed profiles of normalized turbulent kinetic energy of air compared against data.

REDUCED KINETICS / MANIFOLD METHODS

- Low dimensional manifold methods (ILDM, TGLDM)
  - Given detailed kinetics, they provide low-dimensional description (e.g., 1-D, 2-D, 3-D) in multidimensional composition/scalar space
  - Use dynamical systems theory to determine the low dim. manifold
  - Avoid ad hoc assumptions, e.g., partial equilibrium of some of the reactions
  - Implications for ignition and lean blow-off
- Not fuel specific like conventional reduced kinetic schemes

Perfectly Stirred Reactor (Pope & Maas 1993)

Laminar Premixed Flame (Maas & Pope 1994)
PARALLEL PROCESSING

- Objective: Turnaround time of 1 day or less for 3-D combustor calculations

- Particle partitioning, domain decomposition (multigrid, multi-block)

- Preliminary results for 2-D flow with particle partitioning (Pope 1994)
  - 16 nodes, 128 MB each, IBM SP1
  - 12.8 million particles (800,000 per processor)
  - 50 time steps
  - 44 minutes/processor (45 minutes clock time)

Extrapolation to 3-D combustor calculations
- 6.5 hours clock time with 32 processor SP1

JOINT PDF FOCUS AREAS

- 3-D Flows, Improved solution algorithms

- Parallel processing

- Reduced kinetics / Low Dimensional Manifolds

- Evaporating / reacting sprays

- Emphasis on emissions and performance predictions
TURBULENCE MODELING REQUIREMENTS, DEVELOPMENT PHILOSOPHY AND APPROACH

• REQUIREMENTS
  • TURBULENCE MODELING IS A KEY ENABLING TECHNOLOGY FOR ALL PROPULSION RELATED CFD ACTIVITIES
  • FACTORS TO CONSIDER INCLUDE ACCURACY, CONSISTENCY, COMPUTATIONAL COST, AND EASE OF USE
  • TURBULENCE MODELS THAT CAN NOT BE INCLUDED IN PRODUCTION GRADE CFD CODES ARE OF LIMITED VALUE TO INDUSTRY

• PHILOSOPHY
  • BASIC MODEL DEVELOPMENT IS BEST LEFT TO SPECIALIZED "CENTERS OF EXCELLENCE"
  • VARIOUS CLASSES OF MODELS NEED TO BE SUPPORTED SINCE NO SINGLE UNIVERSAL MODEL IS SHOWN TO EXIST
  • ESTABLISHING THE RANGE OF APPLICABILITY, ACCURACY, AND THE COMPUTATIONAL COST OF THE MODELS IS ESSENTIAL

TURBULENCE MODELING REQUIREMENTS, DEVELOPMENT PHILOSOPHY AND APPROACH (Cont.)

• APPROACH
  • IDENTIFY KEY "CENTERS OF EXCELLENCE" AND ESTABLISH COLLABORATIVE RELATIONSHIP
  • ACQUIRE MODELS AND ASSESS PERFORMANCE FOR THE INTENDED CLASS OF APPLICATIONS
  • DELINEATE MODEL DEFICIENCIES AND INITIATE EFFORT TO REDUCE THEM
  • DEVELOP MODELS INTO STAND-ALONE MODULES
  • INCLUDE MODULES IN PRODUCTION CODES AND ESTABLISH BASELINE FOR APPLICATIONS
TWO MAJOR AREAS OF CONCENTRATION

- HIGH SPEED TURBULENCE MODELING (LEAD DR. DOUG LYNCH)
  - FOCUSED ON HIGH SPEED (M>1) PROPULSION (ROCKET AND AIRBREATHING) AND AERODYNAMICS
  - EMPHASIS ON 2-EQUATION PHENOMENOLOGICAL MODELS WITH NASA ARC AND LARC AS KEY TECHNOLOGY PARTNERS
  - LES WORK IN PLANNING STAGES WITH CTR

- LOW SPEED TURBULENCE MODELING (LEAD DR. ALI HADID)
  - FOCUSED ON LOW SPEED (M<1) AND ROTATING FLOW APPLICATIONS
  - EMPHASIS ON REYNOLDS STRESS PHENOMENOLOGICAL MODELS IN COLLABORATION WITH UMIST, ICOMP, CTR, AND UAH
  - LES WORK INITIATED WITH CTR

HIGH SPEED TURBULENCE MODELING

- EMPHASIS IS ON THE DEVELOPMENT OF ENGINEERING TURBULENCE MODELS FOR
  - HIGH SPEED AIRBREATHING PROPULSION SYSTEMS
  - THRUST CHAMBERS
  - VEHICLE AERODYNAMICS

- APPROACH TAKEN IS BASED ON 2-EQUATION MODELS
  - DIFFERENT CLASSES OF 2-EQUATION MODELS STUDIED
    - k-ε
    - k-ω
    - POINTWISE R
  - COMPRESSIBILITY EFFECTS AND TURBULENCE-CHEMISTRY INTERACTIONS MAJOR MODEL UPGRADE THRUSTS
  - COMPRESSIBILITY MODIFICATIONS FROM ARC
  - TURBULENCE-CHEMISTRY INTERACTION MODELS FROM LARC
  - USA AND GASP SERVE AS NUMERICAL PLATFORM
    - GASP - CHIEN, LAM-BREMHOST k-ε, k-ω
    - USA - VARIETY OF k-ε, k-ω
COMPRESSIBILITY EFFECTS

- MIXING LAYER SPREADING REDUCED AT HIGH MACH NUMBERS
- INCREASE DISSIPATION RATE OF K
- DEFINE Ck0 AS A FUNCTION OF TURBULENT MACH NUMBER (PK/P)
- ZEMAN MODIFICATION (1990)
- MODIFICATIONS OF ZEMAN AND SARKAR NOT RECOMMENDED
- HEAT TRANSFER OVER PREDICTED NEAR SHOCK WAVES
- LIMIT TURBULENT LENGTH SCALE $L_t$ TO $\min \left( \frac{k^{3/2}}{\epsilon}, \frac{K_Y}{C_{mu}^{3/4}} \right)$ (VUONG AND COAKLEY, 1987)
- SEPARATION UNDERPREDICTED IN RAPID COMPRESSION OR STRAIN REGIONS
- INCREASE $\alpha_0$ OR $\alpha_0^+$ UNDER RAPID COMPRESSION (VUONG AND COAKLEY)
- HEAT TRANSFER OVER PREDICTED FOR VERY COLD WALLS $T_w/T_\infty < 0.1$ (COAKLEY)
- CEBECI-SMITH ~ 60%, $k_\omega$ ~ 40%, $q_\omega$ ~ 10%, $k_\varepsilon$ ~ 30%

TURBULENCE MODELS ADAPTED TO USA CODE

<table>
<thead>
<tr>
<th>ALGEBRAIC</th>
<th>DAMPING WALL LOCAL</th>
<th>BOUNDARY CONDITIONS</th>
<th>TRANSITION MODEL</th>
<th>COMPRESSIBILITY EFFECTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Myong-Kasagi</td>
<td>X</td>
<td>X</td>
<td>$k = 0$</td>
<td>X</td>
</tr>
<tr>
<td>2. Chen (1987)</td>
<td>X</td>
<td>X</td>
<td>$k = 0$</td>
<td>X</td>
</tr>
<tr>
<td>3. Jones-Lauder (1972)</td>
<td>X</td>
<td>X</td>
<td>$k = 0$</td>
<td>X</td>
</tr>
<tr>
<td>4. Launder-Sharma (1974)</td>
<td>X</td>
<td>X</td>
<td>$k = 0$</td>
<td>X</td>
</tr>
<tr>
<td>5. Huang-Cookley (1982)</td>
<td>X</td>
<td>X</td>
<td>$k = 0$</td>
<td>X</td>
</tr>
<tr>
<td>6. Speziale-So-Zhang (1990)</td>
<td>X</td>
<td>X</td>
<td>$k = 0$</td>
<td>X</td>
</tr>
<tr>
<td>7. Lam-Bremhorst (1981)</td>
<td>X</td>
<td>X</td>
<td>$k = 0$</td>
<td>X</td>
</tr>
<tr>
<td>8. High Re</td>
<td>X</td>
<td>X</td>
<td>Wall Function</td>
<td>X</td>
</tr>
<tr>
<td>9.</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1. High Re Wilcox (1991)</td>
<td>-</td>
<td>-</td>
<td>$k = 0$</td>
<td>X</td>
</tr>
<tr>
<td>2. Low Re Wilcox (1993)</td>
<td>-</td>
<td>-</td>
<td>$k = 0$</td>
<td>X</td>
</tr>
<tr>
<td>9.</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>COAKLEY (1987)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>One-Equation (Goldberg - Two-Time Scale 1995)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>One-Equation Ry (Goldberg 1993, 1994)</td>
<td>X</td>
<td></td>
<td>$k = 0$</td>
<td>X</td>
</tr>
</tbody>
</table>
M = 8.2 FLAT PLATE FLOW
• CHIEN k-ε MODEL WITH RAPID COMPRESSION AND LENGTH SCALE COMPRESSIBILITY MODIFICATIONS

VELOCITY PROFILE

TURBULENT VISCOSITY

REF: G.T. COLEMAN AND J.L. STOLLERY, JFM 56: 741, "HEAT TRANSFER FROM A HYPERSOニック TURBULENT FLOW AT A WEDGE COMPRESSION CORNER"

MACH 7.05 FLOW OVER AXISYMMETRIC FLARE
CHIEN k-ε MODEL WITH RAPID COMPRESSION AND LENGTH SCALE COMPRESSIBILITY MODIFICATIONS

AXISYMMETRIC FLARE
WALL PRESSURE FOR AXISYMMETRIC FLARE

MACH 7.05 FLOW OVER AXISYMMETRIC FLARE
CHIEN k-ω MODEL WITH RAPID COMPRESSION AND LENGTH SCALE
COMPRESSIBILITY MODIFICATIONS

WALL HEAT TRANSFER FOR AXISYMMETRIC FLARE

MACH 8.6 FLOW OVER COLD WALL WEDGE

M = 8.6
Tw/Taw = 0.065

THREE STUDIES

1. CHIEN k-ε MODEL WITH RAPID COMPRESSION AND LENGTH SCALE
   CORRECTIONS AND WITH AND WITHOUT MIXING LAYER TREATMENT
2. HIGH-Re k-ω MODEL WITH VARIOUS AIR CHEMISTRY MODELS
3. BALDWIN-LOMAX TURBULENCE MODEL USING WALL AND LOCAL
   DAMPING
MACH 8.6 FLOW OVER COLD WALL WEDGE
CHIEN k-ε MODEL WITH AND WITHOUT MIXING LAYER TREATMENT

HEAT FLUX CALCULATIONS

MACH 8.6 FLOW OVER COLD WALL WEDGE
HIGH-Re k-ω MODEL WITH VARIOUS AIR CHEMISTRY MODELS

HEAT FLUX CALCULATIONS
LOW SPEED TURBULENCE MODELING

• EMPHASIS IS ON THE DEVELOPMENT OF ENGINEERING TURBULENCE MODELS FOR
  • ROTATING MACHINERY
  • FLOW IN DUCTS AND MANIFOLDS
  • REACTING FLOWS

• APPROACH TAKEN IS TO
  1. SYSTEMATICALLY ASSESS EXISTING PHENOMENOLOGICAL MODELS USING COMMON NAVIER-STOKES SOLVER
  2. IDENTIFY, DEVELOP AND VALIDATE MODEL UPGRADES COMMENSURATE WITH OBSERVED FLOW PHYSICS
  3. DEVELOP SELF-CONTAINED TURBULENCE MODEL DECKS (MODULES) THAT CAN BE INTEGRATED WITH NAVIER-STOKES SOLVERS
  4. PROVIDE GUIDANCE TO EXPERIMENTAL AND THEORETICAL RESEARCH IN TURBULENCE MODELING FOR ENGINEERING APPLICATIONS
TURBULENCE MODELS BEING ASSESSED

PHENOMENOLOGICAL SINGLE POINT CLOSURE MODELS

- SINGLE-SCALE
- MULTI-SCALE

2-EQUATION MODELS
- K-ε (SKEM)
- ALGEBRAIC STRESS MODELS (ASM)
- REYNOLDS STRESS MODELS (RSM)

NEAR-WALL TREATMENTS INCLUDE (WHERE APPROPRIATE) WALL FUNCTIONS, MULTILAYER MODELS, AND LOW-REYNOLDS NUMBER APPROXIMATIONS

TURBULENCE MODEL DECK STRUCTURE AND INTEGRATION WITH NAVIER-STOKES SOLVER

- PREPROCESSOR
  1. GRID
  2. BOUNDARY CONDITION FLAGS
  3. FLOW PROPERTIES
  4. INITIAL CONDITIONS

- NAVIER-STOKES SOLVER
  - INPUT TO TURBULENCE DECK
  - MEAN VELOCITY \( U_1 \)
  - OUTPUT FROM TURBULENCE DECK
  - ITERATION LOOP

- TURBULENCE MODEL DECK
  - SELF-CONTAINED DECK WITH BUILT-IN SOLVER
  - 2 LEVELS OF MODELING
    1. EDDY VISCOSITY MODELS
       - 1. SKEM
    2. REYNOLDS STRESS MODELS
       - 1. ASM
       - 2. RSM
PROJECT WELL UNDERWAY

• TEAM
  - MODELS PROVIDED BY UMIST, LERC/ICOMP, ARC/CTR
  - MODULE DEVELOPMENT BY ROCKETFYNE
  - MODULE TESTING BY ROCKETFYNE (REACT, USA) AND UAH (MAST)
  - MODEL UPGRADES BY ROCKETFYNE, UMIST, ARC/CTR
  - APPLICATION BY ROCKETFYNE TO TURBOPUMP COMPONENT (E.G. IMPELLER) ANALYSIS

• 2-D MODULES COMPLETED, TESTED, AND RELEASED
  - SINGLE SCALE k-ε
  - MULTI SCALE k-ε
  - ASM
  - RSM

• 3-D MODULE DEVELOPMENT IN PROGRESS

NONLINEAR ALGEBRAIC-STRESS MODEL
VORTEX SHEDDING FROM RECTANGULAR CYLINDERS (DURAO, et al)

![Particle Streaklines](#)

**PARTICLE STREAKLINES**

**MEAN AXIAL VELOCITY ALONG CENTERLINE**

**MEAN KINETIC ENERGY ALONG CENTERLINE**

![Graphs](#)
ROTATION MODIFIED $k$-$\epsilon$ MODEL
BACKWARD FACING STEP (DRIVER AND SEEGMILLER)
STREAMLINE CONTOURS

MEAN AXIAL VELOCITY AT $X/M=4$
RADIAL TURBULENT INTENSITY ($\overline{\nu'\nu'})$

ALGEBRAIC STRESS MODEL
CONFINED COAXIAL SWIRLING JET FLOW (ROBACK AND JOHNSON)
GEOMETRY
STREAMLINE CONTOURS
DECAY OF MEAN AXIAL CENTERLINE VELOCITY
RADIAL PROFILES OF $\overline{\nu'\nu'}$
CONCLUDING REMARKS

- PROGRAMS (BOTH COMMERCIAL AND GOVERNMENT) EMPLOY NEW TECHNOLOGY ONLY WHEN IT PROVIDES "ADDED VALUE"
  - REDUCED DEVELOPMENT COST
  - INCREASED RELIABILITY AND PERFORMANCE
  - ENHANCED MANUFACTURABILITY
- THE NEW TECHNOLOGY WE OFFER IS THE COMPUTATIONAL ENGINEERING TOOLS FOR PRODUCT DESIGN AND ANALYSIS
- THESE TOOLS ARE THE END PRODUCT FOR ALL ENABLING TECHNOLOGY DEVELOPMENT
  - PRE- AND POST PROCESSING
  - ALGORITHMS AND NUMERICAL PLATFORMS
  - PHYSICAL MODELS (E.G. TURBULENCE AND CHEMISTRY)
- FAILURE OF ANY ENABLING TECHNOLOGY JEOPARDIZES THE PERFORMANCE (VALUE) OF THE TOOL

NOW MORE THAN EVER, THERE IS A NEED FOR CLOSER COLLABORATION AND COOPERATION BETWEEN GOVERNMENT, INDUSTRY, AND RESEARCH INSTITUTIONS TO ENSURE MAINTENANCE OF COUNTRY'S TECHNOLOGY BASE
ACKNOWLEDGEMENTS

- NASA LeRC - Phase II SBIR
  Technical Monitor: David Fricker

- Pratt & Whitney: Dr. Geoff Sturgess

- Wright Laboratories: Mr. Dale Shouse

MOTIVATION
Accurate and Efficient Prediction of Emissions

1. Accurate Prediction of Emissions From Combustion Devices Requires Treatment of Finite-Rate Kinetics

2. The Effect of Turbulent Fluctuations In Velocity, Energy, Composition, etc. on Finite-Rate Chemical Kinetics Must be Modeled
TURBULENCE/CHEMISTRY INTERACTIONS

Possible Approaches

- Neglect Fluctuations
  + Simple
  - Ignores Effect of Turbulence

- Eddy Break Up
  + Simple
  - Assumes Fast Chemistry
  - Mean Density, Temperature Must Still Be Modeled

- Prescribed PDF
  + Efficient
  - Limited to Fast Chemistry or Single Step Reaction

- Composition PDF
  + Finite-Rate Multi-Step Kinetics
  - Expensive
  - Gradient Diffusion

- Velocity-Composition PDF
  + More Accurate
  - More Expensive

PARTICLE REPRESENTATION

A Solution Method for a Large Number of Independent Variables

- Computational Requirements Increases Exponentially With Dimensions for Finite Difference Methods

- Computational Requirements Increase Linearly With Dimensions for Monte Carlo Methods
COMPOSITION PDF SOLUTION
Stochastic Lagrangian Particle Simulation

Particle Composition and Position Changed to Model Transport of Joint PDF

- Mean Convection
  - Move Particles Between Cells

- Chemical Reactions
  - Lookup Table Holds Composition Change

- Turbulent Diffusion
  - Exchange Particles Between Cells

- Molecular Mixing
  - Particle Interaction Changes Composition

COUPLING
PDF Solution is Separate Module

\[ \begin{align*} 
\text{CFD-ACE} & \quad \text{Monte Carlo PDF} \\
\tilde{u}, \tilde{v}, \tilde{w}, \tilde{p} & \quad \tilde{u}, \tilde{v}, \tilde{w}, k, \varepsilon \\
k, \varepsilon & \quad f (\psi_1, \ldots, \psi_n) \\
\rho & \quad \bar{\rho} 
\end{align*} \]
CHEMICAL KINETICS

Reduced Models are Used

Hydrogen: \[ 2H_2 + O_2 \rightleftharpoons 2H_2O \]

CO: \[ CO + H_2O \rightleftharpoons CO_2 + H_2 \]
\[ 2H_2 + O_2 \rightleftharpoons 2H_2O \]

Methane: \[ CH_4 + 2H + 2H_2O \rightarrow CO + 4H_2 \]
\[ CO + H_2O \rightleftharpoons CO_2 + H_2 \]
\[ 2H_2 + O_2 \rightarrow 2H_2O \]
\[ 3H_2 + O_2 \rightleftharpoons 2H_2O + 2H \]

Hydrocarbon: \[ C_nH_{2n+2} + \left(\frac{n}{2}\right)O_2 \rightarrow n CO + (n + 1)H_2 \]
\[ C_nH_{2n+2} + nH_2O \rightarrow n CO + (2n + 1)H_2 \]
\[ CO + H_2O \rightleftharpoons CO_2 + H_2 \]
\[ 2H_2 + O_2 \rightleftharpoons 2H_2O \]

Thermal NO: \[ N_2 + O \rightleftharpoons NO + N \]
\[ N + O_2 \rightleftharpoons NO + O \]
\[ N + OH \rightleftharpoons NO + H \]

RESULTS TO BE PRESENTED

- Jet Diffusion Flame (Hydrogen with Helium Dilution)
- Bluff Body Stabilized Flame \((H_2/CO)\)
- Piloted Jet Diffusion Flame (Methane)
- Generic Gas Turbine Combustor \((Propane)\)
HYDROGEN JET DIFFUSION FLAME
Illustration of Experiment at Sandia National Lab

$Re = 10^4$

Fuel

100% $H_2$
80% $H_2$, 20% He
60% $H_2$, 40% He

60% HYDROGEN FLAME
Scatter Plots of Mixture Fraction and NO Mole Fraction

PDF Results

Experim Data
UNDILUTED HYDROGEN FLAME
Conditional Averaged NO Mole Fraction

HYDROGEN DIFFUSION FLAME
Dilution Effects on Emissions Index
BLUFF BODY STABILIZED DIFFUSION FLAME
Illustration of Experiment of Correa and Gulati

Inlet Air Flow
- 27.5% CO
- 32.3% H2
- 40.2% N2

BLUFF BODY STABILIZED DIFFUSION FLAME
Composition PDF Predicts Mean Values as well as Velocity-Composition PDF
### PILOTED JET DIFFUSION FLAME
Illustration of Experiment of Masri et.al.

#### Experimental Conditions

<table>
<thead>
<tr>
<th>Flame</th>
<th>Fuel Jet Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>L</td>
<td>41 m/s</td>
</tr>
<tr>
<td>B</td>
<td>48 m/s</td>
</tr>
<tr>
<td>M</td>
<td>55 m/s</td>
</tr>
</tbody>
</table>

---

### PILOTED JET DIFFUSION FLAME
Good Agreement with Experimental Data

![Diagram of pilot jet diffusion flame with experimental data at x/D = 20 and x/D = 30]
PILOTTED JET DIFFUSION FLAME
More Accurate Prediction with Monte Carlo PDF

No PDF          Prescribed PDF          Monte Carlo PDF

Flame B at x = 20D

GENERIC GAS TURBINE COMBUSTOR
Pratt & Whitney Four-Nozzle Sector
Combustor Tested at Wright Laboratory

67,840 Cells
MONTE CARLO PDF COMBUSTOR CALCULATION
Stochastic Particle Traces

VERTICAL PLANE THROUGH CENTER OF FUEL INJECTOR
Mean CO Mass Fraction Countours
RUN TIME AND MEMORY

3D Combustor Calculation
(68,000 cells)

<table>
<thead>
<tr>
<th>Method</th>
<th>CPU Time</th>
<th>Memory</th>
</tr>
</thead>
<tbody>
<tr>
<td>Conventional CFD</td>
<td>20 hours</td>
<td>80 MBytes</td>
</tr>
<tr>
<td>Monte Carlo PDF</td>
<td>100 hours</td>
<td>120 MBytes</td>
</tr>
<tr>
<td>Parallel PDF (Projected)</td>
<td>25 hours</td>
<td>30 MBytes</td>
</tr>
<tr>
<td></td>
<td>25 hours</td>
<td>30 MBytes</td>
</tr>
<tr>
<td></td>
<td>25 hours</td>
<td>30 MBytes</td>
</tr>
<tr>
<td></td>
<td>25 hours</td>
<td>30 MBytes</td>
</tr>
</tbody>
</table>

CPU Time for IBM RS/6000 Model 560

CONCLUSIONS

- Monte Carlo PDF Solution Successfully Coupled with Existing Finite Volume Code
  - Minor Changes to Finite-Volume Code
  - Can be Coupled with Other Codes

- PDF Solution Method Applied to Turbulent Reacting Flows
  - Good Agreement with Data for 2D Case
  - Demonstration of 3D Elliptic Flow

- PDF Methods Must be Run on Parallel Machines for Practical Use
OUTLINE OF TALK

• Part I: Turbulence Modeling
  – Challenges in Turbulence Modeling
  – Desirable Attributes of Turbulence Models
  – Turbulence Models in FLUENT
  – Examples using FLUENT
• Part II: Combustion Modeling
  – Turbulence-Chemistry Interaction
  – FLUENT Equilibrium Model
• Concluding Remarks

PART I:
Turbulence Modeling and Industrial Flows

• Many industrial flows are turbulent; certainly in the markets that two of our codes, FLUENT and RAMPANT, are focused in.
• Turbulence augments rates of mass, momentum and heat transfer, often by orders of magnitude.
• Most combustion processes involve turbulence and often depend on it.
• Choice of turbulence model dictates the accuracy of CFD predictions.
• There is still a large gap between the state-of-the-art and users' expectations and needs.
Challenges in Turbulence Modeling

- Modeling the correlations: $\rho \overline{u_i u_j}$ and $\rho \overline{u_i \phi}$.
  - Closures based on the "eddy-viscosity" concept (industry's most popular choice)
  - Closures based on transport equations (RSM)

- Modeling an additional transport equation for a scalar quantity to fix the state of turbulence.
  - Most popular choice: the kinetic energy dissipation rate, $\varepsilon$.
  - However, this equation is derived by continuum mechanics-based phenomenological considerations and intuition.

- Modeling of the viscosity-affected, near-wall laminar sublayer.
  - Most popular choice: "Wall-functions" that bridge the turbulent field to the solid wall.
  - However, assumptions involved are not always right.

Desirable Attributes of Turbulence Models in Commercial CFD Codes

- Accuracy and Universality
  - The range of applicability should be as broad as possible.
  - Applicable to complex geometries and unstructured meshes.

- Economy
  - Mathematically simple.
  - Memory and CPU requirements should be moderate and affordable (model formulation and grid distribution requirements).

- Robustness
  - Model should be able to solve a wide range of problems with little or no convergence problems.
  - Computationally efficient (fast execution speed and uses memory sparingly).
Turbulence Models in FLUENT

- \( k-\varepsilon \) model adequate for simple flows with no significant strain rates.
- RNG \( k-\varepsilon \) model for separated flows, flows with large streamline curvature, swirling flows, or flows with significant strain rates.
- RSM recommended for swirling flows or highly anisotropic flows.

\( k-\varepsilon \) Model: Some Comments

- Well-tested, used for over 20 years, limitations well understood.
- It forms a good compromise between universality and economy of use for many engineering problems.
- Subject to the inherent limitations of the Boussinesq's hypothesis, i.e., isotropic eddy-viscosity and Newtonian closure (gradient-diffusion model).
- Many assumptions are introduced in deriving the modeled equations for the turbulent quantities, particularly the \( \varepsilon \)-equation, making their fidelity limited.
- The constants in the modeled equations are calibrated against simple benchmark experiments.
- As a result, the \( k-\varepsilon \) model performs poorly in flows with curvature, swirl, rotation, separated flows, low-Reynolds number flows, strongly anisotropic flows, etc.
Renormalization Group (RNG) Based $k-\varepsilon$ Model

- Basic theory and derivation are described in Yakhot and Orszag (1986). Further details and applications are in Yakhot, Orszag, Thangam, Speziale, and Gatski (1992), Speziale and Thangam (1992).
- First introduced in a commercial code, FLUENT, in 1992.
- The RNG method is essentially a scale-elimination technique that can be applicable to the Navier-Stokes and other scalar transport equations as well.
- Removal of successively large scales leads to differential transport equation models and associated formula for quantities such as the turbulent Prandtl/Schmidt number.
- The basic form of the RNG-based $k-\varepsilon$ equations remains largely the same with the standard $k-\varepsilon$ model. But, the constants in the model equations are derived explicitly from theory.
- The $\varepsilon$-equation ends up with an additional source term, a strain-dependent term.
- The RNG model can be integrated directly to a solid wall without using ad hoc damping functions or damping terms used in many near-wall models.
- High-Re form of the turbulence kinetic energy and dissipation rate equations derived by RNG procedure are:

\[
\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = P_k - \varepsilon + \frac{\partial}{\partial x_i} \left( \frac{\nu_r}{\sigma_k} \frac{\partial k}{\partial x_i} \right)
\]

\[
\frac{\partial \varepsilon}{\partial t} + U_i \frac{\partial \varepsilon}{\partial x_i} = 1.42 \frac{\varepsilon}{k} P_k - 1.68 \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_i} \left( \frac{\nu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right) - \mathcal{R}
\]

where:

- $\sigma_k = \sigma_\varepsilon = 0.7179$
- $P_k = 2\nu_r S_{ij} S_{ij}$ is the kinetic energy production

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right)
\]

is the mean rate of strain tensor

- $\nu_r = C_\mu \frac{k^2}{\varepsilon}$
- $\mathcal{R} = \frac{C_{\omega} \eta^4 (1 - \frac{\eta}{\eta_0})}{1 + \beta \eta^\gamma} \frac{\varepsilon^2}{k}$
- $\eta = S k / \varepsilon, \; S = (2 S_{ij} S_{ij})^{1/2}$
- $\eta_0 = 4.38, \; \beta = 0.015$
RNG-Based $k$-$\varepsilon$ Model (Cont'd)

- In the low-Re RNG model, a differential relationship exists between $\frac{k}{\varepsilon}$ and $\nu_{\text{eff}}$ (Yakhot and Orszag, 1986).
- The turbulent Prandtl/Schmidt number is no longer a constant, and computed from relationships relating the local value of the number to the viscosity ratio (Yakhot and Orszag, 1986).
- In these relations, as $\tilde{\nu} \to 1$, $\alpha \to \alpha_0$ (the low-Re limit) and as $\tilde{\nu} \to \infty$, $\sigma = \alpha^{-1} \to 0.7179$ (the high-Re limit). Here:
  - $\tilde{\nu} = \frac{\nu_{\text{eff}}}{\nu_0}$, where $\nu_{\text{eff}} = \nu_0 + \nu_T$
  - $\alpha = \text{inverse turbulent Prandtl number} (\sigma^{-1})$
  - $\alpha_0 = \text{inverse molecular Prandtl number} (\sigma_0^{-1})$
- In the low-Re regions, $\sigma_k$ and $\sigma_\varepsilon$ are obtained similarly from the Prandtl number relationships, with $\alpha_0 = 1.0$.
- The relationships ensure that in the high-Re number part of the flow where $\tilde{\nu} >> 1$:
  $$\nu_{\text{eff}} = \nu_T = 0.085 \frac{k^2}{\varepsilon}$$
  and the effective viscosity varies smoothly from the molecular viscosity to the turbulent viscosity.
- The low-Re eddy-viscosity formula does not explicitly involve any geometric length scale, i.e., the distance from a solid wall used in the damping functions adopted by most low-Re near-wall models, which is a very convenient feature for calculations for complex three-dimensional geometries.
- In collaboration with the originators of the RNG model, Drs. Yakhot and Orszag, the model has been extended to account for the effects of compressibility, swirl, rotation, and premixed combustion.
- The RNG-based $k$-$\varepsilon$ model also works well with conventional and enhanced (non-equilibrium) wall functions available in Fluent Inc.'s codes.
The Reynolds-Stress Model in FLUENT

- RSM solves transport equations for the Reynolds stresses: $\overline{u_iu_j}$ (4 equations in 2D problems, 6 equations in 3D problems).
- RSM is the level of modeling that has a well established track record of out-performing eddy-viscosity models in complex flows.
- It is computationally more expensive and more inclined to divergence and stability problems.
- The simple and most widely tested form of the Launder, Reece and Rodi (1975) form is used.
- The interpolation technique for co-located grids of Rhie and Chow (1983) is used.
- It offers the best choice for highly anisotropic flows.

Example 1:
Circle-to-Rectangle Transition Duct

- $Re_D = 3.9 \times 10^5$.
- Solution Domain.
  - Upstream Inlet Boundary: $x/D = -1.0$
  - Downstream Exit Boundary: $x/D = 8.0$
  - A Quadrant of the duct modeled.
Turbulent Flow in a Transition Duct

Station 6 (x/R = 8.0)

Calc. (RNG/k-ε)
Calc. (RSM/Soti. & Patel)
Experiment (David O. Davis)

Contours of computed streamwise velocity (RNG-based k-ε model)
Example 2:
Cyclone Sparator

- Measured by Qing (1983).
- RSM is used.
- Cylindrical $55 \times 23 \times 41$ grid.

![Cyclone Sparator Diagram]

![Axial Velocity Graph]

**FLUENT predictions**

**experiment**

Radial Distance: $(R-r)/R$
Example 3:
180° Bend of Square-Cross Section

- Solution Domain
  - Upstream Boundary: 5.0\(D_H\) from the start of the bend
  - Downstream Boundary: 5.0\(D_H\) from the end of the bend
  - A symmetric half of the duct modeled.

- Mesh
  - Orthogonal 101 × 47 × 27
  - Distance from the wall ≈ 0.01\(D_H\)
Part II:
Combustion Modeling

- With environmental awareness, legislations on combustion-generated pollutants such as $NO_x$, $SO_x$, carbon monoxide, soot, unburnt hydrocarbons, etc. have become increasingly tougher.
- Combustion simulation in industrial applications can help us to design combustors with higher efficiencies and lower pollutant emissions.
- The combustion process involves some of the most complex phenomena such as chemistry, multiphase flow, turbulence, heat transfer and the interaction between these phenomena.
- Here we focus on gaseous combustion in which the reactants may be mixed or non-mixed prior to flowing into the combustor.
Turbulence-Chemistry Interaction

- Accurate simulation of turbulent combustion requires a thorough assessment of the way turbulence and chemistry interact. The reaction rate and flame structure primarily depend on this interaction.

- In turbulent flames, chemical rates can be significantly different than those in laminar flames (sometimes several orders of magnitude), and the mean chemical rate is not the same as the rate calculated based on mean values of the various scalars:

\[ r(\bar{\theta}_1, \bar{\theta}_2, ...) \neq r(\bar{\theta}_1, \bar{\theta}_2, ...) \]

- Turbulent-chemistry interaction is best characterized by the Damkohler number which is the ratio of characteristic flow time to chemical reaction time:

\[ Da = \frac{\tau}{\tau_c} = \frac{\tau_l \bar{\theta}}{\tau_l \bar{\theta}} \]

- When \( Da \ll 1 \) chemical reactions are orders of magnitude slower than turbulent mixing and the influence of turbulence on reaction can be neglected.

- When \( Da >> 1 \) chemical reactions are very fast and hence combustion is controlled by turbulent mixing.

- At high \( Da \) we can exploit the laminar flame concept: turbulent flame is comprised of an array of laminar flames (flamelets). Hence chemical rate expressions can be those obtained in laminar flames and the effect of turbulence can be characterized through the probability density function (pdf).

- For turbulent diffusion flame, the pdf is usually expressed in terms of a scalar which can best characterize mixing, e.g., the mixture fraction. Since the rate of reaction is much higher than the mixing rate, we can assume that the reaction system is at equilibrium. The effect of turbulence is simply felt by the fluctuations in the mixture fraction. The mean value of any scalar in the flame is simply:

\[ \bar{\theta} = \int \theta(\xi)P(\xi)d\xi \]

- For turbulent premixed flames the pdf is usually expressed in terms of a scalar which can best characterize the reaction progress, e.g., normalized temperature:

\[ \bar{r} = \int r(c)p(c)dc \]
**FLUENT Equilibrium Model**

- For turbulent diffusion flames we use a two-moment beta pdf and equilibrium data to calculate various thermo-chemical scalars in the flame.

- To obtain equilibrium data we use the popular CHEMKIN library of SANDIA, fully interfaced with our codes. CHEMKIN contains data on all important gaseous fuels, combustion intermediates and products as well as their properties.

- We obtain the mean mixture fraction and its variance from their respective conservation equations:

\[
\frac{\partial}{\partial x_i} (\rho u_i \xi) = \frac{\partial}{\partial x_i} (\mu \frac{\partial \xi}{\partial x_i})
\]

\[
\frac{\partial}{\partial x_i} (\rho u_i \xi^2) = \frac{\partial}{\partial x_i} (\rho \xi \frac{\partial \xi}{\partial x_i}) + \frac{2\mu}{\alpha} \left( \frac{\partial \xi}{\partial x_i} \right)^2 - C_d \rho \xi \xi^2
\]

- To save computational time we calculate the integrals before the CFD calculations.

**Concluding Remarks**

- As of now, we provide our users with three turbulence models:
  - the "conventional" k-ε model,
  - the ReNormalization Group model,
  - the Reynolds-Stress Model.

- The Renormalization group k-ε model has broadened the range of applicability of two-equation turbulence models.

- The Reynolds-stress model has proved useful for strongly anisotropic flows such as those encountered in cyclones, swirlers and combustors.

- Issues remain, such as near-wall closure, with all classes of models.

- Collaborative research with ICOMP will not only serve to further quantify applicability of turbulence models but may bring to market new ideas in the field of turbulence modeling for industrial flows.
EXPERIENCES WITH TWO-EQUATION TURBULENCE MODELS

Ashok K. Singhal, Yong G. Lai, and Ram K. Avva
CFD Research Corporation
Huntsville, Alabama

N95-27894

OUTLINE

- Introduction to CFDRC
- Experiences with 2-Equation Models
  - Models Used
  - Numerical Difficulties
  - Validation and Applications
  - Strengths & Weaknesses
- Answers to Three Questions (Posed by Workshop Organizing Committee)
  1. What Are Your Customers Telling You?
  2. What Are You Doing In-House?
  3. How Can NASA-CMOTT Help?

INTRODUCTION TO CFDRC

- Young and Energetic (Turbulent) Organization, Dedicated to the Continuous Process of Advancement and Effective Transfer of CFD Technology

- TWO TYPES OF COMPLEMENTARY ACTIVITIES:
  - PROJECTS
  - SOFTWARE
INTRODUCTION TO CFDRC (Continued)

- Objective User of Turbulence Models
  (0, 1, and 2 Equation Models, RSM and LES)

- Humble Developer, e.g. Monte Carlo Joint Scalar PDF

- Active Participant in Recent Small Eddies of Turbulence, e.g.
  - National Workshops at: NASA MSFC, LeRC/CMOTT, etc. 1987-1994
  - ASME/Fluids Engineering Division, Biathlon, Lake Tahoe, June 1994

TWO-EQUATION MODELS USED

- Standard $k-\varepsilon$ Model (Launder & Spalding, 1974)
- Low-Re $k-\varepsilon$ Model (Chien, 1982)
- Extended $k-\varepsilon$ Model (Chen & Kim, 1987)
- Multiscale $k-\varepsilon$ Model (Kim & Chen, 1988)
- RNG-Based $k-\varepsilon$ Model (Yakhot et. al. 1993)
- 2-Layer $k-\varepsilon$ Model (Rodi, 1991)
- $k-\varepsilon^{++}$ Models
- $k-\omega$ Model (Wilcox, 1991)

++ Models with Corrections for: Curvature, Rotation, Buoyancy, Compressibility, etc.
NUMERICAL DIFFICULTIES

- Positivity of $k$ & $\varepsilon$ (or $\omega$) is not guaranteed in iterative algorithms
- Strong nonlinearity of source terms and coupling causes numerical difficulties
- Inappropriate specifications of $\varepsilon$ (or $\omega$) at boundaries or in initial conditions may also cause divergence
- Non-orthogonality of grids adds to difficulties
- Non-smooth change over for two-layer model hinders convergence

VALIDATIONS PERFORMED

- Channel and Pipe Flows
- Backward-Facing Step
- Turnaround Duct
- Swirl-Flow Combustor
- Rotating Disk Cavities
- Boundary Layers
- Jets, Wakes, and Mixing Layers
- Periodic wakes behind bluff bodies

Examples of successes and failures
1) Flow Around a Square Cylinder; 2) $180^\circ$ Square Duct; 3) S-Shaped Annular Diffuser; 4) Dump Combustor; 5) Backward Facing Step
FLOW AROUND A SQUARE CYLINDER

Strouhal Number

Strouhal Number = \( \frac{fH}{U_o} \)

\( f \) = Frequency of Vortex Shedding

\( H \) = Obstacle Height

\( U_o \) = Freestream Velocity

<table>
<thead>
<tr>
<th>Model/Expt.</th>
<th>Time Period</th>
<th>Strouhal Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Expt.</td>
<td>7.25</td>
<td>0.138</td>
</tr>
<tr>
<td>Standard k-( \epsilon )</td>
<td>7.1</td>
<td>0.141</td>
</tr>
<tr>
<td>2-Layer k-( \epsilon )</td>
<td>7.1</td>
<td>0.141</td>
</tr>
<tr>
<td>RNG k-( \epsilon )</td>
<td>7.6</td>
<td>0.132</td>
</tr>
</tbody>
</table>

Notes:


2. Computations with CFD-ACE
   - Inlet: 78H Upstream; Outlet: 22H Downstream
   - Grid: 120 x 80
   - Time Steps: Over 70 Per Time Period


FLOW AROUND A SQUARE CYLINDER

Instantaneous Streamlines

Mid-Cycle

End of Cycle
FLOW IN A 180° SQUARE DUCT

Computational Domain

Static Pressure Along Duct Walls

• Experiment by Chang, Humphrey and Modavi (1983)
• Computations Done with CFD-ACE on a 40x40x20 Grid


FLOW IN A 180° SQUARE DUCT

Mean Axial Velocity at \( \theta = 3^\circ \)

\[ \begin{align*}
\hat{u}/u_0 & = 1.2 \\
\hat{u}/u_0 & = 1.3 \\
\hat{u}/u_0 & = 1.4
\end{align*} \]

\[ \begin{align*}
R' & = \frac{r - r_1}{r_0 - r_1} \\
Z & = \frac{z}{D_{H/2}}
\end{align*} \]
S-SHAPED ANNULAR DIFFUSER

* k-ε Model and RNG Model Failed to Predict the Correct Location of the Maximum Velocity Downstream
* Computations with CFD-ACE; Publication Under Preparation

Confined Swirling Flow for a Dump Combustor

* K-ε model failed to preserve the vortex core strength near center (see x/h=10 & 18)
* Computational results to be presented at 1994 ASME Winter Annual Meeting (Chicago)
BACKWARD-FACING STEP

Sensitivity to Grid Refinement

- Low-Re Model Requires >30 Nodes in the Inter Layer


BACKWARD FACING STEP

2-Layer Model;

80 x 60 Grid, Central Differencing

Computations with CFD-ACE; To Be Published
EXAMPLE APPLICATIONS

• Gas Turbine Combustors
• Liquid Rocket Engines
• Seals and Bearing Cavities
• Impellers, Inducers, and Fans
• IC Engines
• CFD Reactors
• External Aerodynamic Flows
• Plus Many More

STRENGTHS & WEAKNESSES

Strengths of 2-Equation Models

• Numerically Economical
• Easy to Modify
• Reasonable Applicability Within Engineering Accuracy

Weaknesses

• Use of Wall Functions Requires First Grid Outside the Viscous Sublayer. This is Difficult to achieve, a Priori
• Low-Re Approach Does Not Offer Overall Advantage.
• Two-Layer Approach Needs More Work (e.g. Smoothing)
• Reynolds Analogy Inadequate for Heat-Transfer Applications.
• Effect of Surface Roughness on Turbulence.
CMOTT/CP QUESTIONS

1. What Are Your Customers Telling You?

2. What Are You Doing In-House?

3. How Can NASA-CMOTT Help?

WHAT ARE CUSTOMERS TELLING?

• PLEASE Don't Confuse Us, with Additional Models and False Hopes

• Conclusions (Confusion) Over Last 15-Years
  - Use k-ε Model, with Wall Functions
    - Wall Functions, Oh No!, Never!!
      Use Low-Re k-ε : Which One?, How?? (Good Questions)
    - k-ε Is No Good; Neglects Non-Isotropicity, etc., etc.
    - Jump on RSM Wagon, Now!
      It Can Take You Anywhere, Eventually!!

    - Look How Great is this k-ε++
      When and How to Use it? (Good Questions)

    - Look How Accurate is this Scheme, No Numerical Diffusion.
      Don't Contaminate the Solutions with Turbulence
WHAT IS CFDRC DOING?

- Using What is Available, in Best Possible Ways
- Listening to Both Sides (Model Developers and Users)
- Trying to Resist Peer Pressures
- Struggling to Find Resources for Mundane Goals Such as Developing Guidelines for Correct Use of Turbulence Models

HOW CAN CMOTT HELP?

- CMOTT Has Been Providing Commendable Service in the Very Difficult Subject: Turbulence
- "Turbulence Subprogram" Should Help Further
- Additional Effort is Needed in Many Areas, Such As:
  - Near Wall Treatment
  - Effect of Surface Roughness
  - Economical Heat Transfer Model
  - Documentation of Experiences in:
    a) Model Robustness (In Addition to Accuracy)
    b) Model Sensitivity to Grid Distribution and Boundary Conditions
  - Transition Model (if Possible Suitable for k-ε Framework)
HOW CAN CMOTT HELP? (Continued)

- NASA-CMOTT is one of the few groups sustaining momentum for turbulence modeling.

- It is in unique (privileged) position for embracing the challenge of developing specific recommendations (guidelines) for:
  a) Selection of adequate models for different class of problems
  b) Correct use of each model

- The task is difficult but practical

- Select fewer roads, post milestones, and go further

- Move an inch closer to users

![CMOTT Diagram]

Developers  ---  CMOTT  ---  Users
BACKGROUND AND OBJECTIVES

- Two-equation eddy-viscosity models (TEM's) are the most cost effective for the purposes of applied CFD. Give best accuracy vs. cost balance.

- There is a lot of confusion about true strengths and limitations of TEM's especially that of standard $k$-$\varepsilon$ model.

- We have embarked on extensive study of TEM's over wide range of flows:
  - Identify true strengths and limitations of standard $k$-$\varepsilon$ model.
  - Evaluate other TEM's.
  - Assess emerging models and novel modeling trends.
  - Identify key areas requiring further research.

- This talk provides brief review of TEM's from perspective of applied CFD.
  - It provides objective assessment of both well-known and newer models.
  - It compares model predictions from various TEM's with experiments.
  - It identifies sources of modeling error and gives historical perspective of their effects on model performance and assessment.
  - It recommends directions for future research on TEM's.

REMARK:

- Many reported poor predictions of TEM's are primarily due to combination of improper choice of near-wall model and over-diffuse numerics.

- TEM performance can be much improved form further research in:
  - Length scale determining equation.
  - Advanced (Anisotropic/Nonlinear) Eddy-viscosity models.
INTRODUCTION AND BACKGROUND

○ About FDI
  ▶ Over 10 years in business.
  ▶ Primary product FIDAP (Fluid Dynamics Analysis Package).

○ About FIDAP
  ▶ First commercial general-purpose finite element CFD program.
  ▶ Models wide range of flows.
  ▶ Over 700 FIDAP licenses worldwide.

○ FIDAP Turbulence Modeling Capabilities
  ▶ Based on two-equation eddy-viscosity models:
    ▶ Standard k-ε model (Launder and Spalding).
    ▶ Extended k-ε model (Chen and Kim).
    ▶ RNG k-ε model (Yakhot, Orszag, Thangam, Gatski and Speziale).
  ▶ Low-Re near-wall modeling based on two-layer approach:
    ▶ Viscous sublayers spanned by single layer of specialized elements.
    ▶ van Driest’s model used in viscous sublayers.
    ▶ Interpolation functions based on universal flow profiles.
  ▶ Latest turbulence modeling enhancements (to appear soon):
    ▶ Anisotropic eddy-viscosity models.
    ▶ Wilcox’s k-ω model.
    ▶ Anisotropic version of the standard k-ε model.

○ Typical Industrial User
  ▶ Design engineer.
  ▶ Trained in fluid mechanics and heat/mass transfer.
  ▶ Familiar with range of flows of interest to his/her organization.
  ▶ NOT CFD expert.
  ▶ Little or no background in turbulence modeling.

○ Turbulence Modeling Requirements of Applied CFD Codes
  ▶ Optimal balance of cost and accuracy:
    ▶ Turbulence modeling overhead of critical concern.
    ▶ Overall accuracy of ± 15% adequate for most cases.
  ▶ Consistent performance over wide range of flows:
    ▶ Heat/mass transfer
    ▶ 2-D and 3-D (Cartesian, axisymmetric)
    ▶ Complex geometries
    ▶ Transient flows
  ▶ Adaptable to a wide range of complex flow physics:
    ▶ Low-Re effects
    ▶ Variable density/compressibility effects
    ▶ Combustion
    ▶ Two-phase
  ▶ Minimum level of user input/intervention:
    ▶ No fine tuning model coefficients and/or solution parameters.
    ▶ No physical input other than boundary and/or initial conditions.
  ▶ No geometry dependence:
    ▶ Distance to wall and/or y* dependence.
  ▶ Stable numerical characteristics.
TURBULENCE MODELING CONSIDERATIONS

Key Modeling Issues

1. Accurate modeling of mechanisms governing $\rho u'u'_j, \rho u'u'_j, \rho u'_k$.
   a) Pressure-scrambling
   b) Body forces
   c) Transport effects
   d) Dissipation

2. Accurate modeling of characteristic turbulent length scales.

3. Accurate modeling of low-Re near-wall phenomena.

Optimal Level of Turbulence Model for Applied CFD

- Second-Moment Closures (DSMC's) and (ASMC's)
  (+) DSMC's ideally suited to modeling aspects 1-a,b,c above, however,
  (-) DSMC's costly, especially in 3-D in presence of heat/mass transfer.
  (-) Geometry dependence in current pressure-scrambling models.
  (-) ASMC's perform erratically (1-c above not well modeled).
  (-) ASMC's numerically less stable (stiff equations).

- Two-Equation Eddy-Viscosity/Diffusivity Models (TEM's)
  (+) Least costly.
  (+) No geometry dependence (except some low-Re TEM's).
  (+) Numerically more stable.
  (-) Conventional TEM's not suitable for modeling effects 1-a,1-b,1-c.
  (+) Room for significant improvement in predicting effects of complex
     strain and anisotropy through the combined use of improved length
     scale equations and advanced eddy-viscosity models.
  (-) Transport effects (1-c), however, cannot be directly predicted.

LENGTH SCALE DETERMINING EQUATION

THE STANDARD $k-\varepsilon$ MODEL

$$\rho \frac{D e}{D t} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial e}{\partial x_j} \right] + c_1 \frac{\varepsilon}{k} G - c_2 \rho \frac{e^2}{k}$$

where,

$$G = -\rho u'_mu'_j \frac{\partial u_j}{\partial x_j} = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j}$$

and,

$$c_\mu = 0.09, \quad c_1 = 1.44, \quad c_2 = 192, \quad \sigma_e = 1.0, \quad \sigma_\varepsilon = 1.3$$

Remarks on standard $k-\varepsilon$ model:

- Use is made of Boussinesq's "isotropic" viscosity model.
- Fine scale isotropy is assumed in modeling $\varepsilon$ equation.
- Is high-Re model. Must be used with suitable near-wall sub-model.
- Many reported poor predictions are due to improper choice of near-wall
  model, mesh density, discretization scheme and boundary conditions.
- Model predicts much better than commonly believed, if used properly.
- It does however have its shortcomings in predicting difficult flows
  involving strong anisotropy and/or non-equilibrium effects - it tends
  to be over-diffuse. It predicts flatter flow profiles, shorter recirculating
  zones, and occasionally does not predict subtle separation bubbles.
O THE EXTENDED k-\(\varepsilon\) MODEL OF CHEN AND KIM

- Employs modified \(\varepsilon\) equation containing extra generation term.
- Rationale is that in addition to turbulence time scale \(k/\varepsilon\), there is further time scale \(\rho k/\varepsilon\) characterizing response of \(\varepsilon\) to mean strain.

\[
\frac{\rho}{Dt} \frac{\partial}{\partial x_j} \left( \frac{\mu + \mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{c_p}{k} \frac{\varepsilon}{\rho} + c_3 \frac{\varepsilon^2}{k} - c_2 \rho \frac{\varepsilon^2}{k}
\]

\(c_p = 0.09, c_1 = 1.15, c_2 = 1.9, c_3 = 0.25, \sigma_\varepsilon = 0.75, \alpha = 1.15\)

- Remarks on extended k-\(\varepsilon\) model of Chen and Kim:

  - Is high-Re turbulence model. Needs near-wall model.
  - Gives similar predictions to standard model in equilibrium flows.
  - We find Chen and Kim's (1987) recommended model produce predictions that are too under-diffuse in confined flows.
  - We have tuned constants \(c_1 = 1.35\) and \(c_3 = 0.05\) to improve performance
  - Revised model gives better results for some well-known benchmark flows, but improved predictions over standard model are not realized consistently. More experience and possibly fine tuning is needed.

O THE RNG k-\(\varepsilon\) MODEL

- RNG k-\(\varepsilon\) model has undergone two major revisions.
- Latest version due to Yakhot, Orszag, Thangam, Gatski, and Speziale

\[
\frac{\rho}{Dt} \frac{\partial}{\partial x_j} \left( \frac{\mu + \mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{c_1}{k} \frac{\varepsilon}{\rho} + c_3 \frac{\varepsilon^2}{k} - c_2 \rho \frac{\varepsilon^2}{k}
\]

where

\[
R = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \delta_{ij} = \frac{c_p \eta^2 (1 - \eta/\eta_0)}{1 + \eta^3} \frac{\varepsilon^2}{k}
\]

\[
\eta = \frac{s}{\varepsilon}; \quad s = \sqrt{G/\mu_t}
\]

\[c_\mu = 0.085, c_1 = 142, c_2 = 168, \sigma_k = \sigma_\varepsilon = 0.7179, \eta_0 = 4.38, \beta = 0.015\]

- Above version is high-Re turbulence model. Needs near-wall model.
- Most testing of model has been done with simple near-wall model.
- Our testing of model with more accurate near-wall model indicates that RNG model is often under-diffusive in internal flows and can be very over-diffusive in some external flows.
- We have tuned model constants and obtained better overall predictions.

\[
c_\mu = 0.0865, c_1 = 1.45, c_2 = 1.83, \sigma_k = \sigma_\varepsilon = 0.8, \sigma_\varepsilon = 1.15, \eta_0 = 4.618, \beta = 0.17\]

- Revised model gives better results for some well-known benchmark flows, but improved predictions over standard model are not realized consistently. More experience and possibly fine tuning is needed.
Additional Remarks on RNG $k-\varepsilon$ Model:

- Interesting development though no major breakthrough.
- Most model constants are predicted from RNG theory.
- In applying RNG theory it is assumed that turbulence field has very wide spectrum and that inertial sub-range is isotropic.
- Values of model constants predicted by RNG theory are approximate owing to simplifying assumptions made in applying RNG method.
- Model predictions critically dependent on additional term $R$.
- The $R$ term reflects proposed contributions from fine scale anisotropy.
- The $R$ term is not derived and modeled using RNG theory.
- The $R$ term has essential similarities with extra term in $\varepsilon$ eq'n of extended $k-\varepsilon$ model of Chen and Kim.
- Latest model does not predict von Karman constant.

- The most notable fact about the RNG $k-\varepsilon$ model of YOTGS is that it challenges the notion of fine scale isotropy of turbulence
  - Thus $\varepsilon$ (and consequently the characteristic turbulent length scale) is assumed to be significantly influenced by the fine scale structure. These effects are heuristically modeled via the time scale ratio $\eta$.
  - It is interesting to note that the assumption of fine scale anisotropy used in modeling $R$ conflicts with notion of a wide and isotropic turbulent spectrum used in applying RNG theory to rest of model.
  - It is more likely that the turbulent length scale is influenced strongly by large scale anisotropy as characterized by the anisotropy tensor $a_s$.
  - Anisotropic eddy-viscosity models can provide estimates of $a_s$ which can be used to design improved length scale determining eqn's.

Advanced Eddy-Viscosity Models (Beyond Boussinesq)

Anisotropic Eddy Viscosity Models (AEVM's)

- There has been renewed emphasis in developing AEVM's.
- Lead to better approximations of the normal and shear stresses and therefore turbulence anisotropy effects.
- In addition to more accurate modeling of $\mu_\kappa \nabla^2$, AEVM's could potentially be used to improve modeling of:
  - Length scale determining eq'n.
  - Generation rate of turbulence energy.
- Examples of AEVM's are:
  - Lumely (1970)
  - Speziale (1987)
  - Yoshizawa (1984), DIA
  - Rubinstein and Baron (1990), RNG
  - Taulbee (1992) and Speziale (1993), derived from DSMC's
  - Launder (1993)

Remarks:

- Potential of models have been demonstrated using simple tests.
- Improvements in accuracy often of second-order in magnitude.
- Not been extensively tested especially for swirling flows.
- Anisotropic models not yet extended to turbulent heat/mass fluxes.
- We are presently investigating AEVM's of Speziale (1987) and Launder (1993).
THE LOW-RE NEAR-WALL MODEL

Wall Function Models

- Produce over-diffuse solutions in off-equilibrium boundary layers.
- Often fail to predict separation or vortex shedding.
- Unfortunately still in extensive use in applied CFD codes.

Specialized Finite Element Model (FIDAP)

- Is essentially two-layer model.
- Avoids fine near-wall mesh via use of one layer of specialized elements.
- Employs van Driest's low-Re mixing-length model in near-wall layer.
- Combines low cost of wall function models with accuracy of two-layer models.
- \( y^+ \) dependence confined to single layer and transparent to user.

Remarks:

- Most of historical testing and verification of TEM's has been done using wall functions. The excess diffusion has lead to much confusion in assessing TEM's.
- Proper assessment of TEM's requires at least two-layer models.
- Wall function approach is simply unacceptable for applied CFD.

IMPACT OF DISCRETIZATION ERROR

Sources of Discretization Error:

- Grid refinement (grid convergence).
- Location of computational boundaries (e.g., outlet, inlet, entrainment).
- Choice of discretization scheme in space and time.

Remarks:

- Effect of discretization error has received less attention in turbulence model development and testing.
- Most serious source of error results from discretization of advection terms (i.e., the upwinding scheme).
- Common but dangerous upwinding strategy is used in many CFD codes:
  - Use accurate unbounded scheme in mean flow equations.
  - Use inaccurate numerically diffuse scheme in turbulence equations.
  - Overall scheme is stable but often highly diffusive.
  - Most of development and testing of turbulence models has been done using above upwinding strategy.
  - In our computations we employ the accurate streamline upwind (SU) scheme in both mean and turbulence equations.
  - Even more accurate schemes are available which are based on Petrov-Galerkin finite element formulations.

- Accurate schemes must be used in both mean flow and turbulence eq's.
NUMERICAL RESULTS

○ Free Jets
  ▶ Round jet
  ▶ Plane jet

○ Internal Flows with Separation
  ▶ Flow past backward facing step (Kim et. al)
  ▶ Flow past step in channel with diffuser wall (Driver and Seegmiller)
  ▶ Flow in pipe expansion (Szszepura)

○ Transient Flow (Vortex Shedding)
  ▶ Flow past square prism (Lyn)

○ 3-D Flow
  ▶ Flow past passenger car models

REMARKS:

▶ Five sets of model predictions are presented:
  ▶ Standard k-ε model
  ▶ Extended k-ε model (original)
  ▶ Extended k-ε model (revised)
  ▶ RNG k-ε model with (original)
  ▶ RNG k-ε model with (revised)

FREE JETS

The Submerged Plane and Round Jets

<table>
<thead>
<tr>
<th></th>
<th>Plane Jet</th>
<th>Round Jet</th>
</tr>
</thead>
<tbody>
<tr>
<td>dδ/dx</td>
<td>% error</td>
<td>dδ/dx</td>
</tr>
<tr>
<td>Experiment</td>
<td>0.105</td>
<td>0.095</td>
</tr>
<tr>
<td>Standard k-ε model</td>
<td>0.104</td>
<td>-1</td>
</tr>
<tr>
<td>Extended k-ε model (original)</td>
<td>0.10</td>
<td>-5</td>
</tr>
<tr>
<td>Extended k-ε model (revised)</td>
<td>0.102</td>
<td>-3</td>
</tr>
<tr>
<td>RNG k-ε model (original)</td>
<td>0.131</td>
<td>25</td>
</tr>
<tr>
<td>RNG k-ε model (revised)</td>
<td>0.101</td>
<td>-4</td>
</tr>
</tbody>
</table>
TURBULENT FLOW OVER BACKWARD FACING STEP

Kim et al Test Case: Re = 45000

<table>
<thead>
<tr>
<th>Model Type</th>
<th>$X_a$ (m)</th>
<th>% error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>7.0 ±0.5</td>
<td></td>
</tr>
<tr>
<td>Standard k-$\varepsilon$ model</td>
<td>6.5</td>
<td>-7.1</td>
</tr>
<tr>
<td>Extended k-$\varepsilon$ model (original)</td>
<td>8.4</td>
<td>20.0</td>
</tr>
<tr>
<td>Extended k-$\varepsilon$ model (revised)</td>
<td>7.1</td>
<td>1.4</td>
</tr>
<tr>
<td>RNG k-$\varepsilon$ model (original)</td>
<td>7.5</td>
<td>7.1</td>
</tr>
<tr>
<td>RNG k-$\varepsilon$ model (revised)</td>
<td>7.46</td>
<td>6.6</td>
</tr>
</tbody>
</table>

TURBULENT FLOW OVER STEP IN CHANNEL WITH DIFFUSER WALL

Driver and Seegmiller Test Case: Re = 36000

<table>
<thead>
<tr>
<th>Angle</th>
<th>$X_a$ (m)</th>
<th>% error</th>
<th>$X_a$ (m)</th>
<th>% error</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 degrees</td>
<td>6.2</td>
<td>8.1</td>
<td>6.6</td>
<td>-18.5</td>
</tr>
<tr>
<td>Standard k-$\varepsilon$ model</td>
<td>5.3</td>
<td>-14.5</td>
<td>6.6</td>
<td>9.55</td>
</tr>
<tr>
<td>Extended k-$\varepsilon$ model (original)</td>
<td>6.6</td>
<td>6.5</td>
<td>9.55</td>
<td>17.9</td>
</tr>
<tr>
<td>Extended k-$\varepsilon$ model (revised)</td>
<td>5.76</td>
<td>-7.1</td>
<td>7.4</td>
<td>-8.6</td>
</tr>
<tr>
<td>RNG k-$\varepsilon$ model (original)</td>
<td>6.17</td>
<td>-0.5</td>
<td>8.33</td>
<td>2.8</td>
</tr>
<tr>
<td>RNG k-$\varepsilon$ model (revised)</td>
<td>6.11</td>
<td>-1.5</td>
<td>8.33</td>
<td>2.8</td>
</tr>
</tbody>
</table>
TURBULENT FLOW IN PIPE EXPANSION

Szczepura Test Case: Re = 890,000

<table>
<thead>
<tr>
<th>Experiment</th>
<th>Xe</th>
<th>% error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard k-ε model</td>
<td>9.51</td>
<td>0.9</td>
</tr>
<tr>
<td>Extended k-ε model (original)</td>
<td>9.59</td>
<td>0.9</td>
</tr>
<tr>
<td>Extended k-ε model (revised)</td>
<td>12.44</td>
<td>30.8</td>
</tr>
<tr>
<td>RNG k-ε model (original)</td>
<td>10.6</td>
<td>11.5</td>
</tr>
<tr>
<td>RNG k-ε model (revised)</td>
<td>11.35</td>
<td>19.5</td>
</tr>
</tbody>
</table>

TURBULENT FLOW PAST SQUARE PRISM

Lyn's Test Case: Re = 21400

<table>
<thead>
<tr>
<th>Strouhal No.</th>
<th>C4</th>
<th>C1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>0.132 ± 0.035</td>
<td>2.0</td>
</tr>
<tr>
<td>Standard k-ε model</td>
<td>0.128</td>
<td>1.68</td>
</tr>
<tr>
<td>Extended k-ε model (original)</td>
<td>0.131</td>
<td>2.56</td>
</tr>
<tr>
<td>Extended k-ε model (revised)</td>
<td>0.135</td>
<td>2.04</td>
</tr>
<tr>
<td>RNG k-ε model (original)</td>
<td>0.133</td>
<td>2.38</td>
</tr>
<tr>
<td>RNG k-ε model (revised)</td>
<td>0.133</td>
<td>1.9</td>
</tr>
</tbody>
</table>
CONCLUSIONS AND DIRECTIONS FOR FURTHER RESEARCH

- For applied CFD, TEM's strike balance between accuracy and efficiency.

- The use of inadequate near-wall models and over-diffuse numerical schemes obscures true performance characteristics of TEM's. And this has lead to much confusion in evaluation of TEM's.

- Consequences of using better near-wall model and accurate numerics are:
  - Standard k-ε model performs much better than commonly believed.
  - Extended k-ε model with original set of model constants produces under-diffuse predictions.
  - RNG k-ε model with original set of model constants gives predictions that can be both under-diffusive or over-diffusive depending on flow.
  - The extended and RNG models with revised set of model constants perform better than with original set of model constants.

- Newer models are quite promising, but do not yet perform consistently better than standard k-ε model.

- Significant advances in TEM capabilities may potentially result from further research in two key areas:
  - Advanced constitutive-type laws for the Reynolds stresses:
    - AEMV's appear to be best candidates.
  - Improved length scale determining equation:
    - Better modeling of off-equilibrium effects.
    - Better modeling of large-scale anisotropy effects.
TURBULENCE MODELING NEEDS OF COMMERCIAL CFD CODES: COMPLEX FLOWS IN THE AEROSPACE AND AUTOMOTIVE INDUSTRIES

Bizhan A. Befrui
adapco
Melville, New York

CONTENT OF PRESENTATION

• STAR-CD: COMPUTATIONAL FEATURES
• STAR-CD: TURBULENCE MODELS
• COMMON FEATURES OF INDUSTRIAL COMPLEX FLOWS
• INDUSTRY-SPECIFIC CFD DEVELOPMENT REQUIREMENTS
• INDUSTRIAL COMPLEX FLOWS: APPLICATIONS & EXPERIENCES
  - FLOW IN ROTATING DISC CAVITIES
  - DIFFUSION HOLE FILM COOLING
  - INTERNAL BLADE COOLING
  - EXTERNAL CAR AERODYNAMICS
• CONCLUSION: TURBULENCE MODELING NEEDS

STAR-CD: COMPUTATIONAL FEATURES

• BODY-FITTED NON-ORTHOGONAL COORDINATE SYSTEM
• UNSTRUCTURED COMPUTATIONAL MESH, DIFFERENT CELL TOPOLOGIES, IMBEDDED MESH REFINEMENT, DISCONTINUOUS MESH INTERFACE, MOVING BOUNDARY AND INTERNAL INTERFACES
• PRIMITIVE VARIABLE, SELF-ADAPTIVE ELLIPTIC-HYPERBOLIC PRESSURE CORRECTION METHOD
• COLLOCATED-VARIABLE ARRANGEMENT
• EULER-IMPLICIT TEMPORAL INTEGRATION
• UD, CD, LUD, SFCD SPATIAL DISCRETIZATION, WITH BLENDING CAPABILITY
STAR-CD: TURBULENCE MODELS

• TWO-EQUATION MODEL
  - STANDARD $k-\varepsilon$ WITH CORRECTIONS FOR BULK DILATATION AND BUOYANCY
  - HIGH REYNOLDS NO. RNG BASED $k-\varepsilon$ MODEL

• TWO-ZONE (TWO-LAYER) MODEL
  - HIGH REYNOLDS NO.: $k-\varepsilon$ VARIANTS
  - LOW REYNOLDS NO.: $k-\varepsilon$ VARIANTS, PRANDTL MIXING LENGTH MODEL

STAR-CD: TURBULENCE MODELS

• REYNOLDS STRESS TRANSPORT MODEL
  - TRANSPORT EQUATIONS FOR CARTESIAN STRESS TENSOR IN NON-ORTHOGONAL COORDINATE SYSTEM, ON NON-STRUCTURED MESH
  - LAUNDER, RODI, REECE (1975) FORMULATION WITH LAUNDER (1989) MODEL CONSTANTS
  - GIBSON & LAUNDER (1978) WALL REFLECTION MODEL
Driver & Seegmiller Backward Facing Step
Flow Domain = 29°H to 32°H
Mesh = 105 (Axial) x 45 (Radial)
Graph Plot Frames

Legend
- Exp. data
- RS model
- KE model

Dover & Seegmiller Backward Facing Step
Data Inter B.C.; No Wall Damping Funct.
Location X/d = 1.5
COMMON FEATURES OF INDUSTRIAL COMPLEX FLOWS

- THREE DIMENSIONAL WITH MULTIPLE FLOW "COMPLEXITIES"
  - BODY-FORCE FIELDS
  - STREAM SURFACE CURVATURE
  - STRONG PRESSURE GRADIENTS
  - COMPRESSIBILITY EFFECTS
  - LAMINAR-TURBULENT TRANSITION
  - COMBUSTION, SHOCK, MULTIPHASE, NON-NEWTONIAN
- LARGE SCALE DOMAIN AND COMPLEX GEOMETRIC CONFIGURATION
- IRREGULAR, UNSTRUCTURED COMPUTATIONAL MESH
- SPATIAL RESOLUTION DIFFICULT TO ACHIEVE ON O(10^6 - 10^8) MESH CELLS
- INSUFFICIENT AND UNCERTAIN EXPERIMENTAL DATA FOR TURBULENCE MODEL VALIDATION/IDENTIFICATION OF DEFICIENCIES
INDUSTRY-SPECIFIC CFD DEVELOPMENT REQUIREMENTS

• AUTOMOTIVE INDUSTRY
  - EFFICIENT COMPLEX-GEOMETRY, MOVING-BOUNDARY CAPABILITIES
  - MEMORY/SOLUTION PERFORMANCE FOR LARGE SCALE DOMAIN CFD SIMULATION
  - DIAGNOSTIC/COMPARATIVE EVALUATION OBJECTIVES
  - GEOMETRIC FIDELITY AND SPATIAL RESOLUTION ARE PRIMARY ACCURACY FACTORS

• AEROSPACE INDUSTRY
  - REGULAR AND SMALL-SCALE FLOW DOMAIN (BENCHMARK EXPERIMENTAL MODELS)
  - DESIGN/PERFORMANCE OPTIMIZATION OBJECTIVES
  - NUMERICAL AND TURBULENCE MODEL ACCURACY IMPORTANT
  - REQUIREMENTS
    • HEAT TRANSFER
    • LOW REYNOLDS NO. FLOW
    • BODY FORCE FIELDS
Figure 1: Exterior boundary conditions for W202 40 kph analysis

- Exterior tube (right half removed)
- Symmetry plane
- Moving wall simulation of ground motion relative to vehicle (40 kph)
- Moving wall simulation of rotating tires (371 RPM)
- Inlet simulation of vehicle traveling at 40 kph (T = 30°C, P = 100 kPa)
W202 UNDERHOOD FLOW ANALYSIS
CASE 3: 40 kph SIMULATION
Velocity near the surface of the vehicle.
## APPLICATIONS & EXPERIENCES

<table>
<thead>
<tr>
<th>APPLICATION (DATA)</th>
<th>FLOW COMPLEXITY</th>
<th>TURBULENCE MODEL</th>
<th>FINDINGS</th>
<th>T.M. NEEDS</th>
</tr>
</thead>
<tbody>
<tr>
<td>ROTATING DISC CAVITY¹</td>
<td>• FORCE FIELD</td>
<td>• k-ε</td>
<td>• EKMAN LAYER RESOLVED</td>
<td>• RSTM + SUITABLE 2 LAYER</td>
</tr>
<tr>
<td></td>
<td>• WALL EFFECT</td>
<td>• 2 LAYER k-ε</td>
<td>• FAIR PRESSURE DROP</td>
<td>• LOW Re RSTM</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• EXCESSIVE E.V.</td>
<td></td>
</tr>
<tr>
<td>DIFFUSION HOLE FILM COOLING²</td>
<td>• JET-CROSS FLOW</td>
<td>• k-ε</td>
<td>• JET SEPARATION SENSITIVE TO MESH TOPOLOGY/ RESOLUTION</td>
<td>• RSTM + SUITABLE 2 LAYER</td>
</tr>
<tr>
<td></td>
<td>• WALL ANISOTROPY</td>
<td>• RNG, k-ε</td>
<td>• POOR SPANWISE SPREAD</td>
<td>• LOW Re RSTM</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 2 LAYER k-ε</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

¹ GRABER et al (1987)

---

**COMPRESSOR DRUM TEST RIG STAR-CD CONJUGATE HEAT TRANSFER MODEL**

![Diagram of Compressor Drum Test Rig]

- Upstream Endwall
- Downstream Endwall
- Bore Tube
- Axis of Rotation
- Disk 1
- Disk 5
Compressor Drum Test Rig Cold Flow Benchmark Analysis
Secondary Flow
Compressor Drum Test Rig Cold Flow Benchmark Analysis
Secondary Flow in Cavity 2
Velocity Vectors at r = 7.40 inches
CAVITY 4: PRESSURE DROP

Dimensionless Pressure Drop

- STAR-CD Model
- Test Data
- Forced Vortex
- Free Vortex

Refined mesh, M = 0.5, pipe grid abutting plate grid.
Mesh = 330000 fluid cells
Two-Layer mesh

1.000
1.000
1.000
ANGLE
0.000
DISTANCE
84.234
CENTER
268.461
78.277
118.614
E:HOIOEN PLOT

183
CFD Discrete Hole Film Cooling Verification Study
Simulation of experiment of Goldstein, et. al. [1988]; Blowing ratio M = 0.5
Velocity vectors on spanwise planes; 2-layer model.

CFD Discrete Hole Film Cooling Verification Study
Simulation of experiment of Goldstein, et. al. [1988]; Blowing ratio M = 0.5
Temperature contours on spanwise planes; 2-layer model.
EXPERIMENTS OF GOLDSTEIN ET AL., 1968
COMPARISON OF FILM COOLING EFFECTIVENESS
M = 0.5 : 2 Layer mesh.
### APPLICATIONS & EXPERIENCES (cont’d)

<table>
<thead>
<tr>
<th>APPLICATION (DATA)</th>
<th>FLOW COMPLEXITY</th>
<th>TURBULENCE MODEL</th>
<th>FINDINGS</th>
<th>T.M. NEEDS</th>
</tr>
</thead>
<tbody>
<tr>
<td>INTERNAL BLADE COOLING³</td>
<td>• FORCE FIELD</td>
<td>• $k$-$\epsilon$</td>
<td>• DEPENDENCE ON MESH RESOLUTION</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• B.L. DISRUPTION</td>
<td></td>
<td>• GOOD $\Delta P$, $h$</td>
<td></td>
</tr>
<tr>
<td>EXTERNAL CAR AERO-DYNAMICS⁴</td>
<td>• B.L. STRUCTURE INTERACTION</td>
<td>• $k$-$\epsilon$</td>
<td>• DEPENDENCE ON MESH RESOLUTION</td>
<td>RSTM</td>
</tr>
<tr>
<td></td>
<td>• COMPLEX WAKE</td>
<td>• RNG $k$-$\epsilon$</td>
<td>• GOOD $C_p$</td>
<td>LOW Re RSTM</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 2 LAYER $k$-$\ell$</td>
<td>• POOR LIFT</td>
<td></td>
</tr>
</tbody>
</table>

³GE AIRCRAFT ENGINES [ABUAF & KERCHER (1991)]
⁴10 FORD 1/4 SCALE MODELS IN WIND TUNNEL TEST [WILLIAMS et al (1994)]
29 Aug 94
PRESSURE
N/M^2
LOCAL MX= 4.427E+06
LOCAL MN= 2719E+05

6 Sep 94
MAGNITUDE VELOCITY
M/SEC
PSYS= 2
LOCAL MX= 314.5
LOCAL MN= 4850
"PRESENTATION GRID"
Figure 4a Leading edge channel heat transfer distribution with distance from the inlet. Comparison of model turbulent convex surface maximum, minimum and average measurements with blade CFD average predictions.
WIND TUNNEL AERODYNAMICS STUDY OF NOTCHBACK TEST SHAPE
KE RESULTS - KE TURBULENCE MODEL WITH LID
VELOCITY MAGNITUDE NEAR THE VEHICLE

VIEW FROM REAR
EXPERIMENT RESULTS
COMPARISON OF EXPERIMENTAL AND COMPUTATIONAL LIFT COEFFICIENTS
K Epsilon TURBULENCE MODEL - """"INITIAL RESULTS """

EXPERIMENTAL RESULTS
COMPARISON OF EXPERIMENTAL AND COMPUTATIONAL DRAG COEFFICIENTS
K Epsilon TURBULENCE MODEL - """"INITIAL RESULTS """
CONCLUSIONS: TURBULENCE MODELING

IMMEDIATE NEEDS

• NEAR-WALL TURBULENCE
  - ECONOMICAL, ROBUST LOW REYNOLDS NUMBER 2 EQ. EVM’s AND RSTM
  - A GENERAL AND VERSATILE NEAR-WALL TREATMENT FOR RSTM

• RSTM MODEL
  - ALTERNATIVE CLOSURE OF THE WALL REFLECTION COMPONENT, WITHOUT NEED OF WALL TOPOGRAPHY PARAMETERS

• EDDY-VISCOSITY MODELS
  - EXTENSION OF THE NON-LINEAR k-ԑ TO INCORPORATE FORCE-FIELD EFFECTS

• BENCHMARKING
  - A RELIABLE DATABASE OF BENCHMARK SET OF REPRESENTATIVE COMPLEX FLOWS
  - BENCHMARK PERFORMANCE CLASSIFICATION OF VARIOUS EVM’s (k-ԑ, k-ω, RNG AND NON-LINEAR k-ԑ, MULTISCALE EVM’s) AND RSTM CLOSURE VARIANTS

CONCLUSIONS: TURBULENCE MODELING

PROGRAM NEEDS

• A LARGER VIEW OF THE RSTM DEVELOPMENT TOWARDS IMPLEMENTATION IN GENERAL COORDINATE, COMPLEX GEOMETRY DOMAIN, UNSTRUCTURED CFD METHOD

• A BROADER APPLICATION OF DNS TO COMPLEX FLOWS TO ASSIST TURBULENCE MODEL DEVELOPMENT/OPTIMIZATION

• WELL-POSED EXPERIMENTAL DATA, OBTAINED IN THE ORIGINAL OR REDUCED SCALE MODEL OF THE INDUSTRIAL COMPONENT FOR CFD VALIDATION

• COLLABORATIVE INDUSTRY-CFD RESEARCH/DEVELOPMENT PROGRAMS FOR EXPERIMENTATION - CFD VALIDATION (CALIBRATION) FOR SPECIFIC INDUSTRIAL APPLICATIONS
Outline

- Profiles
  - ASC
  - Application
  - Client
- Needs
  - Clients'
  - ASC's
- ASC Directions
  - Research
  - Development
  - Products
- How Can CMOTT Help?
Profile of ASC

- Established in 1985
- Components of business
  - development
  - applications
  - licensing and service
- Geographic markets
  - North America
  - Europe
  - Pacific rim countries

Application Profile

- Rotating machinery components
  - hydraulic turbines
  - pump
  - compressors
  - turbines
  - stators
  - wicket gates
  - scrolls
  - volutes
  - inlets and diffusers
  - seals
  - stage
  - rotor stator
• Combustion
  - gas turbine combustor
  - coal fired boilers
  - gasification
  - fire suppression
  - emissions reduction
  - safety
• High speed external - ballistics
  - explosively formed projectiles
  - finned projectiles
  - sabot discard
• Heat transfer
  - turbine cooling
  - nuclear reactors
  - heat exchangers
  - electronics system cooling
• Typical uncertainties
  - geometry
  - initial and boundary conditions
  - transient effects
  - transition
  - limitations of physical models
  - numerical error
Client Profile

- Companies or divisions
  - industrial/manufacturing/research
  - 10 - 200 employees
  - limited or no access to high performance computing

- Users
  - design and/or analysis
  - < 3 people
  - network of engineering workstations
  - turnaround time in less than a day for analysis, hours for design

Clients’ Needs

Needs are most readily identified through typical questions from clients.

- General
  - I am using k-ε or two-layer or k-ω, or RNG ..., what does it mean to my calculation? Tell me in words what the deficiencies of the model means for my application?
  - What is the relative price/performance of the various turbulence models?
  - Has the model I am using been validated for type of flows I am trying to model? If so, when, where, how ... ?
  - How well does the model handle the interaction between turbulence and rotation, curvature, adverse pressure gradients, separation, swirl, buoyancy, extinction, droplets and particles, anisotropies ...?
  - How can I use Navier-Stokes solvers for design? Can I tune the turbulence model to suit my needs? If so, what are the appropriate settings for my application?
Clients’ Needs cont’d

• Grid
  - I don't have access to high performance computing, I don't have any more time, I have a coarse non-orthogonal mesh, is my CFD result useful?
  - I have just made my grid finer, why should I have to worry about whether y+ is in a given range?

• High speed flows
  - I am solving a flow with many speed regimes including low speed separations and shocks, why do turbulence levels become unphysical as the grid is refined through shocks?
  - How should experimental data be compared to results from time or Favre averaged calculations?

Clients’ Needs cont’d

• Combustion
  - Which of the many different combustion models in combination with which turbulence model works best for my application?
  - How appropriate is the single scale implicit in the turbulence model for the combustion model?
  - How can the Bousinesq assumption be valid in the presence of counter-gradient diffusion?
  - How important are turbulent fluctuations to my problem?
  - If I had all the mean flow and fluctuating components of the turbulent flow, how can the effects of stretch and curvature on the instantaneous flame front be modelled.
  - Can extinction due to vortex stretching be modelled?
  - What is the influence of the flame front on the turbulence?
Clients' Needs cont’d

- **Calculated pdf models**
  - If I use a more detailed chemistry model - like a pdf transport model - how much improvement can I expect in the results for my application? How can I measure that?
  - Is it the case that the results for my application will not be sensitive to the shape of the pdf? If not, then why should I incur the costs associated with a pdf transport equation.
  - I am solving a pdf transport equation, how much are the results dominated by the limitations of modelling of the diffusion transport term?

Clients' Needs cont’d

- **Flamelet models**
  - I am using a flamelet model in modelling my gas turbine combustor, but in some regions of the combustor the model is not strictly appropriate - can any of the results be used? If so, how much?
  - In some models like the flamelet model, it is assumed that the turbulent time scale is inversely proportional to the velocity gradient of a “laminar” model flame. What is the validity of this assumption?
  - How sensitive are my results to the assumption of statistical independence of the quantities in a joint pdf?
ASC's Directions

- Develop in-house model expertise
  - two-layer model
  - alternative two-equation models
  - second moment closure models
  - expanded EBU models
  - flamelet model
- Develop in-house expertise applying models
  - turbomachinery
  - combustion
  - heat transfer
- Promote high performance computing
  - parallel computing

How Can CMOTT Help?

- Model improvements to address between turbulence and
  - rotation
  - curvature
  - adverse pressure gradients
  - separation
  - swirl
  - bouyancy
  - droplets and particles
  - anisotropies ...
  as well issues related to
  - extinction
  - trace species
  - vortex stretching
  - flame fronts
  - time and length scales
  - ...
- Great, but is this what users really want?
How Can CMOTT Help? cont’d

- Curator of information on existing models
  - define
  - validate
  - process
  - educate

as an independent agency

Define models
- unified conceptual framework
- establish baseline for various models
- set context for model improvements
- for each model
  > document derivation
  > identify assumptions
  > clearly state implications of assumptions
  > separate physics from numerics
How Can CMOTT Help? cont’d

Validate models

- fundamental flows
  > validate assumptions
- benchmark problems
  > select real engineering problems relevant to identified applications (in propulsion)
  > review selection of benchmark on regular basis
- experimental data
  > collect and review existing data
  > define new experiments
  > review quality of resulting data for validation of models

Process data

- collect
- distil
- review
- interpret
- describe
- compile
Educate
- document
- publish
- workshops
- seminars
- short courses
- market

Summary

Provide information so users, for their applications can:
- make an educated choice of model
- understand how to appropriately use existing models
- move forward with existing models and technology
- understand implications of improvements to existing models
SECOND-ORDER CLOSURES FOR COMPRESSIBLE TURBULENCE

J.L. Lumley, S. Savarese, and C.C. Volte
Mechanical and Aerospace Engineering Department
Cornell University
Ithaca, New York

OUTLINE

• I. Project Description

• II. Turbulence Modeling

• III. Computational Engine / Results

FUTURE WORK

I. PROJECT DESCRIPTION

1. Flows of Interest

2. Motivation

3. Method
Schlieren photograph of a shock-wave turbulent boundary-layer interaction
M=0.90 Re=1,750,000 [Liepmann]

1.2 MOTIVATION

- Industry
- Project
- Modeling
- CFD

204
1.3. METHOD

• 1-Point Closures: from EVM to Second-Order Closures

• Dynamical Compressibility Effects

• 3D / Finite Volume Approach
II. TURBULENCE MODELING

1. Closure Levels

2. Compressibility Effects

3. Shock Wave Interactions
II.1. Closure Levels

1. EVM Mixing-Length
   (Baldwin-Lomax)

2. EVM Multi-Equation
   (k-ε-S)

3. Second-Order Closure
   (Shih and Lumley)

II.2. Compressibility Effects

1. New Physics & Averaging

2. Models
II.2.1. New Physics
(Turbulent Kinetic Energy Sink)

\[ - \langle \tau_{ij} u_{i,j} \rangle = \Pi_d - \varepsilon_d - \varepsilon_s \]

- \( \varepsilon_d = (\mu_B + \frac{4}{3} \mu) < d^2 > \)
- \( \Pi_d = < pd > \)

II.2.3. Turbulence Modeling
(Zeman, Sarkar et al., Yoshizawa)

- dilatation dissipation:
  \[ \varepsilon_d = (\mu_B + \frac{4}{3} \mu) < d^2 > \]
- Sarkar et al. (asymptotic analysis)
- Zeman
  (Shocklet model)
• pressure-dilatation correlation:

\[ \Pi_d = \langle pd \rangle \]

- Zeman (acoustic model):

- Sarkar et al.
  (DNS & asymptotic analysis)

Response of turbulence kinetic energy to the passage through shock
II.3. Shock Wave Interactions

1. Experimental Observations

2. Physics

3. Modeling

II.3.1. Experimental Results

• Oscillation increases with Shock Strength (Dolling)

• Oscillation increases with Separation Region

• Normal Stresses Preferentially Amplified (Déleroy et al.)
II.3.2. Physics

Oscillation Caused by (?):

- "Breathing" of Separation Region

- Vortex Bursting
  in Incoming Boundary Layer
  (Dolling)

II.3.3. Shock Oscillation Modeling

- Parametrized Source Terms
  in Normal RS Evolution Equation
  (gradient activated)

- Separation region Extend
III. COMPUTATIONAL ENGINE

1. Numerical Method

2. Turbulence Models

3. Validation Procedure / Results

III.1. Numerical Method

Initial Code: flo103
(A.Jameson L.Martinelli, Princeton)

1. Geometry
   C-mesh
   2D

2. PDE Solver
   spatial discretization: FV
   time integration: RK

3. Convergence
   Acceleration:
   variable time step
   residual smoothing
   artificial dissipation
   multigrid preconditioning

4. I/O
   PLOT3D format

5. Turbulence Models
   Baldwin-Lomax

Current Code: cyste
(D.Caughey)

1. Geometry
   O-R-meshes
   (EAGLEView MSU)

2. PDE Solver
   variable number of PDE's
   consistent gradient comp.

3. Convergence Acceleration
   Enhanced multigrid sequencing

4. I/O
   Restart option
   Post-processing (DX,Tecplot,...)
   convergence histories

5. Turbulence Models
   k-epsilon (-S)

6. Software Engineering
   Dynamical mem. allocation (C)
   Vectorized data structure
   Unix Integration

Future
1. Geometry
   3D
2. Turbulence Models
   SOC

212
III.2. Turbulence Models:
Incompressible / Compressible: an additive approach

- Baldwin-Lomax

- k-Epsilon / k-Epsilon-S: B.C's

- Second-Order Closures

Boundary Conditions: Wall-Functions

III.3. Validation Procedure / Results

- Calibration against simple well-documented flows (flat plate, jet)

- Results and Comparison of models
**FUTURE WORK**

- **Numerics**
  - 2D $\Rightarrow$ 3D
  - More Complex Wall Functions
  - Realizability Conditions (SOC)

- **Modeling**
  - Refinement of Existing Models ($e_d$, $<pd>$)
  - Shock Oscillation Model
Homogeneous Turbulence (R) / k-eps / Mach=0.045 Re=24357

Downstream Evolution

\[ y = 2 \times 10^{-5} x^{-2.33} \]

\[ y = 2 \times 10^{-5} x^{1.29} \]
Homogeneous Turbulence (R) / k-eps-S / Mach=0.045 Re=24357

Downstream Evolution

\[ y = (2E-4) x^{-1.24} \]

\[ y = (2E-5) x^{-2.41} \]

Homogeneous Turbulence (R) / RSC / Mach=0.5

Downstream Evolution

\[ y = (0.3E-3) x^{-1.28} \]

\[ y = (0.2E-4) x^{-2.33} \]
MODELING OF TURBULENT CHEMICAL REACTION

J.-Y. Chen
Department of Mechanical Engineering
University of California, Berkeley
Berkeley, California

Modeling Turbulent Reacting Flows

Model for Turbulent Flows

Model for Effects of Turbulence on Chemical Reactions

Model for Chemical Kinetics

\[ \tau, I, \ldots \rightarrow \rho \]
Regimes of Turbulent Combustion


Regimes of Premixed Turbulent Combustion

K_q = 1 / D_a_q


220
Regimes of Non-Premixed Turbulent Combustion

Chemical Closure Models

(1) Laminar Chemistry
\[ \langle \rho w_i \rangle = \rho w_i (\overline{Y_i}, \overline{T}) \]

(2) Fast Chemistry
\[ \langle \rho w_i \rangle = \frac{1}{2} \rho \overline{\chi_f} \frac{\partial^2 Y^e(f)}{\partial f^2} \]

(3) Flamelet model
\[ \langle \rho w_i \rangle = \int \rho w_i (\eta, \overline{\chi_f}) P_{r, \eta} (\eta, \epsilon_f) d\eta d\epsilon_f \]

(4) Assumed PDF:
\[ \langle \rho w_i \rangle = \int \ldots \int \rho w_i (\phi_1) \cdot P_\phi \cdot d\phi_1 \cdot d\phi_2 \ldots d\phi_n \]
Assumed the shape of \( P_\phi \).

(5) Scalar PDF method:
Solve for \( P_\phi \) directly.

(6) Conditional Moment Closure (CMC)
\[ \langle \rho w_i \rangle = \int \langle \rho w_i | \eta \rangle \cdot P_r (\eta) d\eta \]
Flamelet library with one side being burned premixed $\phi=1.4$

Flamelet Model: 69%H2 + 31%CH4
Turbulent Jet Flame, Rev. = 10,000

Flamelet Model: 69%H2+31%CH4
Turbulent Jet Flame, Rev.=10,000


Flamelet Model: 69%H2+31%CH4
Turbulent Jet Flame, Rev.=10,000

**Conditional Moment Closure (CMC)**

**Definition:**
\[ <Y,|\eta| >= <Y,(x,t)|f(x,t) = \eta > \]

**Equation:**
\[
\begin{align*}
\frac{\partial <Y,|\eta| >}{\partial t} + \nabla <Y,|\eta| > + \frac{\nabla \cdot (p u' y'|\eta| > p,|\eta|)}{p,|\eta|} \\
= <p w,|\eta| > + <p D, \nabla f \cdot V f|\eta| > \frac{\partial^2 <Y,|\eta| >}{\partial \eta^2}
\end{align*}
\]

**Modeling:**
\[
\begin{align*}
<w,|\eta| = w,(<T|\eta|, <Y,|\eta|, >,\ldots) \\
<p D, \nabla f \cdot V f|\eta| > = <p D, \nabla f \cdot V f >= \frac{1}{2} \delta \chi_f \\
<p u' y'|\eta| > = \bar{p} \bar{u} \\
<p u' y'|\eta| >= 0 \\
<p l|\eta| = p(<Y,|\eta|, <T|\eta|>)
\end{align*}
\]
**Conditional Moment Closure (CMC)**

![Graph showing mole fraction vs mixture fraction with various mole fractions and mixture fractions labeled.]

**NOx Emissions from Turbulent H2 Jet Flames**

![Graph showing log(EOX/wat) vs log(U1/D1) with markers for measurement, scalar PDF, CMC, and assumed PDF.]
**Conditional Moment Closure (CMC)**

Applications:

- Incorporated into existing moment closure CFD codes for complex geometry flows
- Realistic Chemistry - Detailed or reduced

Research issues:

- Modeling of conditional statistics
- Preferential diffusion
- Parallel computing algorithms

**Probability Density Function (PDF)**

Applications:

- NO\textsubscript{x} from methane jet flames with reduced chemistry
- Sooting flames
- 2-D flows

Research Topics:

- Mixing model
- Extension to droplet spray & particle laden flows
- Preferential diffusion
- Efficient stochastic algorithm
- Construction of chemical tables
- Parallel computing - 3D Flows or 2D flows with complex chemistry
Departures From Chemical Equilibrium

(Ref) Hydrogen (Exp)

Methane

Methanol

227
Mixing Models for PDF Methods

- Modified Curl's Model (stochastic)

\[ \frac{\partial^2}{\partial t^2} \left[ \epsilon \alpha \beta \phi = \psi \right] \frac{\partial \phi}{\partial \psi} \left\{ \frac{\phi}{\phi_0} \right\} = \]

\[ \frac{1}{\omega_{\text{mix}}} \left[ \int \phi \left( \psi', \phi \right) \phi_0 \left( \psi'' \right) \delta \left( \psi - \psi'' \right) \right] \]

- IEM (Interaction-by-Exchange-with-the-Mean) Model (deterministic)

\[ \frac{\partial^2}{\partial t^2} \left[ \epsilon \alpha \beta \phi = \psi \right] \frac{\partial \phi}{\partial \psi} \left\{ \frac{\phi}{\phi_0} \right\} = \frac{C_\phi}{\tau_{\text{mix}}} \frac{\partial}{\partial \psi} \left[ \psi - \phi_0 \right] \]

Mixing Frequency: \( \omega_{\text{mix}} = \frac{1}{\tau_{\text{mix}}} \)

PaSR: H2/NOx Detailed Chemistry \( \phi = 1 \) \( \tau = 1 \)ms

![Graphs showing mixing models with data points](image-url)

228
Comparison of Predicted and Measured H2O Mass Fractions
Turbulent Nonpremixed Jet Flames

Fuel: (CO/H2/N2: 0.30/0.10/0.6)

Experimental Evidence of Preferential Diffusion in Turbulent Jet Flames
(Fuel: 36%H2/64% CO2)

Computation of Turbulent Reacting Flows

Development

Direct Numerical Simulation

Large Eddy Simulation

Turbulence Model

Reduced Mechanisms

Stochastic Simulation

Laminar Flames with Detailed Mechanisms

Practical Interests

Combustion in Compressible Turbulence

Soot in Turbulent Flames

Flame Extinction & Reignition

NOx in Turbulent Flames
INTRODUCTION TO TURBULENCE SUBPROGRAM

T.-H. Shih and J. Zhu
Institute for Computational Mechanics in Propulsion
and Center for Modeling of Turbulence and Transition
NASA Lewis Research Center
Cleveland, Ohio

OBJECTIVES

• A means for CMOTT to interact with industry

• A vehicle for technology transfer to industry

CONCEPT OF TURBULENCE MODULE

• Exact CFD equations:

\[
\frac{D\rho U_i}{Dt} = \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \delta_{ij} \right) - \rho \overline{u_i u_j} \right] - \frac{\partial P}{\partial x_i},
\]

• Reynolds stresses will be recast as:

\[
-\rho \overline{u_i u_j} = \mu T \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \delta_{ij} \right) + \left[ -\rho \overline{u_i u_j} - \mu T \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \delta_{ij} \right) \right];
\]

\[
\mu T \equiv C_{\mu} \frac{k^2}{\varepsilon};
\]

• CFD equations become:

\[
\frac{D\rho U_i}{Dt} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu T \right) \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \delta_{ij} \right) \right] + \frac{\partial T_{ij}}{\partial x_j} - \frac{\partial P}{\partial x_i};
\]

• The task of turbulence module: Provide \( \mu_T \) and \( T_{ij} \)
• Turbulence Module:

  ◦ Input: $U_i$, $\rho$ and $\mu$ ... from the mean flow solver

  ◦ Output:

    $\mu_T = C_{\mu} \frac{k^2}{\varepsilon} \left[ \frac{Dk}{Dt} = ..., \quad \frac{D\varepsilon}{Dt} = ... \right]$

    $T_{ij} = -\rho \overline{u_i u_j} - \mu_T \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \delta_{ij} \right)$

  ◦ Models for $\rho \overline{u_i u_j}$
    - One- and two-equation eddy viscosity models
    - Reynolds stress algebraic equation models
    - Reynolds stress transport equation models

![Diagram of the Turbulence Module and Mean Flow Solver](Image)
Module with CMOTT research code (incompressible)

- CFD equations in CMOTT research code:

\[
\frac{D\rho U_i}{Dt} = \frac{\partial}{\partial x_j}[(\mu + \mu_T)(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i})] + \frac{\partial}{\partial x_j}T_{ij} - \frac{\partial P}{\partial x_i}
\]

- Turbulence module: provide \(\mu_T\) and \(T_{ij}\)
  
  ◦ Built-in models without wall function:
    - Launder-Sharma and Chien \(k-\epsilon\) models
    - CMOTT \(k-\epsilon\) model
  
  ◦ Built-in models with wall function:
    - \(k-\omega\) model, standard \(k-\epsilon\) model
    - CMOTT \(k-\epsilon\) model
    - CMOTT Reynolds stress algebraic equation model

Module with NPARC code

- CFD equations in NPARC code:

\[
\frac{D\rho U_i}{Dt} = \frac{\partial}{\partial x_j}[(\mu + \mu_T)(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \delta_{ij})] - \frac{\partial P}{\partial x_i}
\]

- Turbulence module (present time): provide isotropic \(\mu_T\)
  
  ◦ Build-in models without wall function:
    - Baldwin-Lomax model and Chien \(k-\epsilon\) model
    - CMOTT \(k-\epsilon\) model
  
  ◦ Further development:
    - Models with wall function
    - Reynolds stress algebraic equation models
    - Reynolds stress transport equation models
Joint Program with Industry on Turbulence Module

- For those who want to use the available modules:
  - Need interface program for particular industry codes
    - Grid informations, Boundary treatment, ...

- For those who want a module for their own codes:
  - Need modules exclusively for particular industry codes

- Maintain and update the turbulence modules along with model development.
General Transport Equations

\[ \frac{\partial}{\partial t}(rJ^{-1}\rho \phi) + \frac{\partial}{\partial \xi_i}(C_i \phi - D_i \phi) = rJ^{-1} S_\phi \]

- Non-dimensional form \((\mu, \mu_t \leftrightarrow \mu/Re, \mu_t/Re)\)
- Conservative form
- Cartesian velocity components
  1. Easy to transform (chain rule)
  2. No curvature terms
Discretization

- Finite-volume method
- Source term
  \[ S_\phi = S_1 + S_2 \phi, \quad S_1 \geq 0 \text{ and } S_2 \leq 0 \]
- Transient term
  1. 1st-order fully implicit scheme
  2. 2nd-order three-level fully implicit scheme
- Diffusion term
  Standard central differencing scheme

- Convection term: HLPA scheme
  (Hybrid Linear/Parabolic Approximation)

\[
\phi_w = \phi W + \gamma(\phi C - \phi W)\tilde{\phi}_W, \quad \tilde{\phi}_W = \frac{\phi W - \phi WW}{\phi C - \phi WW}
\]

\[
\gamma = \begin{cases} 
  1 & \text{if } |\tilde{\phi}_W - 0.5| < 0.5 \\
  0 & \text{otherwise}
\end{cases}
\]

- Second-order accurate
- Bounded (non-oscillatory)
- Diagonally dominant matrix
Example 1

Initial profile at t = 0

Predicted profiles at t = 100 (201 x 2 grid, \( \Delta t = 0.4 \))

Predicted profiles at t = 100 (1001 x 2 grid, \( \Delta t = 0.1 \))

Example 2

S-profiles at outlet (C, exact solution)

Orthographic projection of S-field.
Solution Procedure

- Non-delta form
  Positiveness ($\phi \geq 0$ but $\Delta \phi$ may $< 0$)
  Simple linearization

- Algebraic equations
  $$A_C \phi_C = A_W \phi_W + A_E \phi_E + A_S \phi_S + A_N \phi_N + S$$
  $A'^s, S \geq 0$

- Decoupled solution

- Alternating direction TDMA solver

Boundary Conditions

- Inflow: $\phi$ specified

- Outflow: Fully-developed condition

- Symmetry: $\partial \phi / \partial n = 0$

- Wall:
  1. Low-Reynolds number turbulence models
  2. Standard wall-function approach
Sub-Programs

- NPARC2D version
  Plane or axisymmetric, without swirling
  Compressible
  Non-vectorized

- FAST2D version
  Plane or axisymmetric, with or without swirling
  Incompressible
  Vectorized

NPARC2D Version

- Grid arrangement
  Control volume centers
  Boundary nodes
  Embedded bodies

J-Patches

K-Patches
• Input from the main code
  1. Geometric quantities: \( x, y, \xi_x, \xi_y, \eta_x, \eta_y, J \)
  2. Flow variables: \( \mu, J^{-1}\rho, J^{-1}\rho U, J^{-1}\rho V, J^{-1}E \)
  3. Patch control: \( 5 \times 2 \) parameters
  4. Boundary conditions: \( 7 \times 2 \) parameters

• Output
  1. To the main code: \( \mu_t \)
  2. For post-processing: \( K, \epsilon, y^+, y_n, f_\mu \)

### FAST2D Version

• Grid arrangement
  - CV centers
  - Boundary nodes
  - Embedded bodies
• Vectorization

Single-index:
\[ ii = i + (j-1)ni \]
\[ \phi(i,j) = \phi(ii) \]
\[ \phi(i+1,j) = \phi(ii+1) \]
\[ \phi(i,j-1) = \phi(ii-ni) \]

Control parameter:
\[ KBLK = \begin{cases} 
1 & \text{for computational nodes} \\
0 & \text{otherwise} 
\end{cases} \]
\[ \phi = KBLK \cdot \phi_c + (1-KBLK) \phi_b \]

• Input from the main code

1. Geometric quantities: \( x, y, x_\xi, x_\eta, y_\xi, y_\eta, J \)
2. Flow variables: \( \mu, \rho, U, V, W, C_w, C_s \)
3. Vectorization parameters
4. Boundary parameters

• Output

1. To the main code: \( \mu_t, T_{ij} \)
2. For post-processing: \( K, \epsilon, y^+, y_n, f_\mu \)
OVERVIEW OF PROBABILITY DENSITY FUNCTION (PDF) MODELING AT LeRC

D.R. Reddy
Internal Fluid Mechanics Division
NASA Lewis Research Center
Cleveland, Ohio

OBJECTIVE

Accurately model the effect of turbulence on chemical reactions in a fluid flow

APPROACH

Use Probability Density Function (PDF) model -
Express dependent variables as functions representing statistically realizable events

POSSIBLE MODELING STRATEGIES

1. Evolution PDF - solve for function
   a. Joint PDF for velocities and chemical species
   b. Joint PDF for only chemical species & energy

2. Assumed PDF - function prescribed
   Limited range of applicability -
   reaction time << or >> turbulence time scale
CURRENT APPROACH

- Develop evolution PDF model for compressible reacting flows & extend to spray combustion
- Solve for joint PDF for species and energy using Monte-Carlo technique
- Couple with conventional CFD codes

AREAS OF IMPACT

- NOx Prediction - HSCT and AST application
- Spray combustion - swirling turb. reacting flows
- Scramjet flow path analysis
- Ignition kinetics - prediction of blow-off, etc.
- Combustion instability studies
CODE FEATURES

- Modular - can be coupled with any CFD code

- Applicable for compressible flows with discontinuities

- Monte-Carlo solver for generalized curvilinear coordinate system

- Easily adaptable for parallel computation (currently under progress)

CURRENT STATUS

- 2-D and axisymmetric version released (default H2-air chemistry - 5 species)
  - parallel version to be released

- 3-D version demonstrated for supersonic combustion (jet in cross flow)
  - validation planned for HSCT-type configurations

- General chemistry (CHEMKIN)
  - Hydrocarbon spray combustion case currently under study

- CFD codes used - RPLUS, ALLSPD, & SIMPLE-type
FUTURE PLANS

- Further application/validation of 3-D model

- Improved closure models - mixing and turbulence
  (use available DNS data)

- Parallel processing - workstation clusters

- Unsteady applications - long-term

- Extend scope of impact
OUTLINE

- Motivation
- PDF modeling of reactive flows
- The Lewis PDF module
- Validations and applications
- Current research
- Technology transfer

COMPUTATION OF TURBULENT COMBUSTION

[Diagram showing CFD, Chemistry Model, Turbulence Model, Engine Design, and connections with Conventional Model and PDF Model]

247
GOVERNING EQUATIONS

\[ p_t + (\rho u_i)_i = 0 \]
\[ (\rho u_i)_i + (\rho u_j u_i)_j = -p_i + \tau_{ij} \]
\[ (\rho E)_t + (\rho u_j E)_j = -q_i + \Phi \]
\[ (\rho Y_k)_i + (\rho u_j Y_k)_j = \nu_{ij} + \omega_k \]

\[ A_t = \frac{\partial A}{\partial t} \]
\[ A_j = \frac{\partial A}{\partial x_j} \]

CLOSURE PROBLEM:

\[ u_i = u_i + u_i' \]
\[ Y_i = Y_i + Y_i' \]

\[ \bar{w}_i u_j' \] — Turbulence Modeling

\[ \bar{Y}_i u_j' \] — Analogy of shear stress: Diffusion model.

\[ \rho w_i \] — ???

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

But in general:

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

248
PDF Modeling of Turbulent Reactive Flows

Current status

- Assumed PDF (Spalding, 1971; Gosman & Lockwook, 1973; ...)
  - Advantage: simple, fast.
  - Disadvantages: Need unique mixture fraction; assumed shape may not be real.

- Composition PDF (Pope, 1976; Dopazo & O’Brian, 1974)
  - Advantage: Reaction rate treated exactly; existing moment closure codes easily adapted.
  - Disadvantages: Turbulent diffusion needs model.

- Velocity-Composition joint PDF (Pope & Chen 1980, Pope 1981)
  - Advantage: Reaction rate treated exactly; no diffusion model needed.
  - Disadvantages: Models for velocity field relatively untried; Require more computer resource.

PDF Modeling of Turbulent Reactive Flows

- Objective:
  - Develop models that can accurately simulate finite rate chemical reactions in turbulent flows.
  - Develop and validate independent PDF modules.
  - Technology transfer.

- Criteria
  - Accuracy and robustness.
  - Practical in terms of today’s computing power.
  - Easy integration with existing industry computational platform.
PDF Modeling of Turbulent Reactive Flows

- Approach:
  - Joint pdf method for scalar compositions.
  - Moment closure schemes for velocity field.
  - Develop hybrid solver consisting of Monte Carlo method and finite-difference/finite-volume method.

PDF Modeling of Turbulent Reactive Flows

- Current status (Lewis)

\[
\begin{align*}
(\rho P)_t &+ (\rho \nabla Y_i \cdot \mathbf{n} > P)_j + (\rho \omega_j P)_j = (D_j P)_j + \mathcal{M}(P) - (S_{\rho P})_j.
\end{align*}
\]

- Continuous mixing model developed.
- Model for compressibility effect proposed.
- 2D and 3D Monte Carlo PDF module developed.
- Validation studies.
- Code released to industry during a workshop.
Validation Cases

- Scalar field in homogeneous turbulence.
- Oblique shock.
- 2D supersonic hydrogen combustor.
- Axisymmetric supersonic combustor.
- Piloted flame near extinction.
Scalar field in homogenous turbulence
pdf compared with Gaussian distribution

Current model

Modified curl's model

Figure 1: Asymptotic pdf distribution for a scalar in homogeneous turbulence. — present model; —— Gaussian.

Figure 2: Asymptotic pdf distribution for a scalar in homogeneous turbulence. — modified Curl model; —— Gaussian.

Scalar field in homogenous turbulence
3rd and 4th moments compared with Gaussian

Current model

Modified curl's model

Evolution of moments from the present model — standard deviation, —— 0.1 × fourth central moment, —— 2.01 × sixth central moment, — 0.1 × fourth moment for Gaussian distribution. O 3.01 × sixth moment for Gaussian distribution.

Evolution of moments from the modified Curl model. — standard deviation, —— 3.01 × fourth central moment, —— 0.0001 × sixth central moment, — 0.0001 × fourth moment for Gaussian distribution. O 0.0001 × sixth moment for Gaussian distribution.
Temperature across an oblique shock: pdf solution compared with analytical prediction.

![Graph showing temperature across an oblique shock with pdf solution compared to analytical prediction.]

Supersonic hydrogen combustor (Exp. Burrows & Kurkov, 1973)

![Diagram of a supersonic hydrogen combustor with dimensions and static pressure ports labeled.]

Temperature Contour (pdf solution)
Supersonic hydrogen combustor:
Mole fraction:
pdf solution compared with exp. data

Coaxial burner: geometry and test condition
(Exp. Cheng, et al. 1991)
Mean H2O mole fraction
Coaxial burner
pdf solution compared with exp. data

Piloted flame (Masri et al., 1994)
Fuel: 45% CO, 15% H2, and 40% N2
Flame close to extinction

255
Piloted Flame
Mean Temperature

Piloted Flame
Flame Location

Expt. (Tmax = 1400)

PDF (Tmax ≈ 1350)
Non-PDF (Tmax ≈ 2000)

Axis (mm)

Radius (mm)
Current Projects

- Application of PDF module to emission predictions
- Incorporate general chemistry procedure.
- Incorporate spray models.
- Use parallel computing for the PDF module.

Collaboration with industry and technology transfer

- Features of independent pdf module:
  - Easily coupled with any existing industry flow codes.
  - Novel averaging scheme to reduce memory requirement.
  - General chemistry package.
  - Parallelized workstation version.
- Technology transfer: workshops
  - July, 1993; code released to 15 US institutions.
INTRODUCTION

• The composition joint PDF method has been used to model a wide class of gaseous turbulent reactive flows. (S.B. Pope)

• Nonlinear chemical reaction rates could be evaluated without any approximation.

• An extension of the PDF method to the modeling of spray flames.

• Evaluate the limitations and capabilities of this method in the modeling of gas--turbine combustor flows.

Composition Joint Pdf Transport Equation

\[
\bar{\rho}\bar{\varphi}_a + \bar{\rho}\bar{u}_a + \left[\bar{\rho}c_a(\psi)\bar{\varphi}_a\right] = 0
\]

\(\{\text{Mean convection}\}\) \(\{\text{Chemical reactions}\}\)

\[-\bar{\rho} < u''_a | \psi > \bar{\varphi}_a - \bar{\rho} < \bar{w}'_a | \psi > \bar{\varphi}_a\]

\(\{\text{Turbulent convection}\}\) \(\{\text{Molecular mixing}\}\)

\[-\bar{\rho} < \frac{1}{2}f'_a | \psi > \bar{\varphi}_a\]

\(\{\text{Liquid--phase exchange}\}\)

- \(\bar{\rho}\) = Density-weighted joint pdf.
- \(w_a\) = chemical source term for the \(a\)-th composition variable.
- \(< u''_a | \psi >\) = conditional average of Favre velocity fluctuations.
- \(< \frac{1}{2}f'_a | \psi >\) = conditional average of scalar dissipation.
- \(< \frac{1}{2}s'_a | \psi >\) = conditional average of liquid-phase source term for the \(a\)-th composition variable.
Modeling Aspects of the Pdf Transport Equation

- $< u^r_l | \psi >$ is modeled using a gradient-diffusion model.
- $< \frac{1}{D_l} p_l | \psi >$ is modeled using a variant of Curl's model.
- The new term $< \frac{1}{p} s_o | \psi >$ involving the conditional average of liquid-phase source term is modeled based on the average values of species and enthalpy:

$$< \frac{1}{p} s_o | \psi >= \frac{1}{p \Delta u} \sum n_k m_k (c_o - \phi_o)$$

for $\phi_o = Y_o, \alpha = 1, 2, ..., s = \sigma - 1$

$$< \frac{1}{p} s_o | \psi >= \frac{1}{p \Delta u} \sum n_k m_k (-k_{eff} + h_k - \phi_o)$$

for $\phi_o = h$.

MODELING ASPECTS


- THE MEAN FLOW AND TURBULENCE EQUATIONS ARE SOLVED BY A CONVENTIONAL CFD SOLVER WITH THE MEAN SPECIES AND TEMPERATURE FIELDS PROVIDED AS INPUTS FROM THE PDF SOLVER AND THE SPRAY SOURCE TERMS FROM THE LIQUID-PHASE SOLVER.

- THE LIQUID-PHASE EQUATIONS ARE FORMULATED IN LAGRANGIAN COORDINATES WITH APPROPRIATE CONSIDERATION TAKEN INTO ACCOUNT OF THE EXCHANGES OF MASS, MOMENTUM, AND ENERGY BETWEEN THE TWO PHASES.
NUMERICAL METHOD

• Mean-Flow and Turbulence Equations
  - Axisymmetric, Unsteady.
  - Incompressible Navier-Stokes (Variable-Density).
  - A Standard Two-Equation $k-\varepsilon$ Turbulence Model.
  - A Pressure-Based CFD Solver Based on the SIMPLE Algorithm of Patankar and Spalding.

• Liquid-phase Equations
  - The Spray Model (Raju and Sirignano).
  - Dilute Spray Assumption.
  - The ODE's for the Particle Size, Velocity, and Location are Solved Using a R–K Method.
  - The PDE's for the Internal Droplet Distribution (Vortex Model) are Solved by an Implicit Method.
  - Droplet Regression Rate is Based on Either a Gas–Phase Boundary Layer–Analysis or Low–Reynolds Correlation.

NUMERICAL METHOD

• The PDF Transport Equation
  - A Fractional Step Monte–Carlo Method (Pope).
  - Spatial Transport, Molecular Mixing, Liquid–Phase Source Terms, and Chemical Kinetics are advanced in a Series of Sequential Steps.
  - Vectorization

• Interaction Between the Two Phases
  - Interpolation of the Gas–Phase Properties at the Particle Location Using an Area–Weighted Averaging.
  - The Source Terms Evaluated at the Particle Location are redistributed among the surrounding Computational Nodes Using an Area–Weighted Averaging.
CHEMICAL KINETICS MODEL

- It is based on a single step global mechanism of Westbrook and Dryer for n-decane/oxygen combustion.

- This global combustion mechanism was shown to provide adequate representation of temperature histories in flows not dominated by long ignition delay times.

Geometry of the combustion chamber. (El Banhawy and Whitelaw)
EXPERIMENTAL DETAILS

• The experimental data corresponds to the following inflow conditions:

  inflow temperature = 310 K,
  air mass flow rate = 355 kg/h,
  air/fuel ratio = 20.17,
  swirl vane angle = 45 deg,
  swirl number = 0.721.

• The reported error in the measurements is about 10 to 15% for the temperature and about 15% for the velocity.

Details of Fuel Injection

• A fuel nozzle of swirl-atomization type was used.
• The liquid fuel injection is simulated by injecting a discretized parcel of liquid mass at the end of each $\Delta t_{injection}$.
• The droplet-size distribution is given by:

\[
\frac{dn}{n} = 4.21 \times 10^6 \left( \frac{D}{D_{32}} \right)^{3.5} \exp \left( \frac{-16.96}{D_{32}} \left( \frac{D}{D_{32}} \right)^{0.4} \right) \frac{dD}{D_{32}}
\]

- The initial droplet injection velocity corresponds to: $u_k = 11.0$ m/s, $w_k = 6.1$, and $v_k = 0.5 - 2.5$. 

263
PARAMETER SELECTIONS

- The computations were performed on a grid with a mesh size of 60x60.

- The PDF solution is obtained by making use of 250 particles per cell.

- $D_t g = D_{t\text{Injection}} = 1.5$ ms, $D_{tK} = 0.0375$ ms, and $D_{t\text{Monte-Carlo}} = 0.015$ ms.

- Two CPU seconds on a CRAY Y-MP per one $D_t g$ and about 2 to 3 CPU hours - 4000 time steps - for the solution to reach steady state.

Velocity vector plot.

Temperature contours and droplet locations.
Spray evaporation rate.

Near wake radial profiles of temperature.
Near wake radial profiles of velocity.

Schematic of an open spray burner.
(Dan Bulzan of IFMD at LeRC)
Photograph of swirl-stabilized, spray flame.
CONCLUDING REMARKS

- The comparisons show that the general features of the flowfield are correctly predicted by the present solution procedure.

- The present solution appears to provide a better representation of the temperature field, particularly, in the reverse-velocity zone.

- The overpredictions in the centerline velocity could be attributed to the following reasons:
  - The use of $k-\varepsilon$ turbulence model is known to be less precise in highly swirling flows.
  - The swirl number used here is reported to be estimated rather than measured.
Overview

- Modeling: What models are used in this package and what are their advantages and disadvantages.

- Numerics: Describe how the PDF model is implemented and what are the features of the program.

- Future Developments: What can be expected in the future from the NASA Lewis PDF code.
PDF Modeled Equations.

- Exact scalar PDF transport equation is:

\[
\frac{\partial}{\partial t}(\bar{\rho}P) + \frac{\partial}{\partial x_i}(\bar{\rho}u_i P) + \frac{\partial}{\partial \phi_\alpha}(\bar{\rho}S_\alpha(\psi, p, \eta)P) = \frac{\partial}{\partial x_i}(\langle \bar{\rho}u_i \psi, \eta \rangle P) + \frac{\partial}{\partial \phi_\alpha}(\langle \frac{\partial \bar{r}_\phi}{\partial x_i} \psi, \eta \rangle P) + \frac{\partial}{\partial \eta}(\langle \frac{\partial q_i}{\partial x_i} \psi, \eta \rangle P) + \frac{\partial}{\partial \eta}(\langle \frac{D_p}{Dt} \psi, \eta \rangle P)
\]

(1)

- Terms on the LHS exact - need to model the four terms on RHS, corresponding to turbulent convection, molecular mixing and the pressure term.

Turbulent Convection

- This term is modeled as a simple gradient diffusion process.

\[
\langle \bar{\rho}u_i \psi, \eta \rangle P \approx D_t \frac{\partial P}{\partial \psi_\alpha}
\]

(2)

- \(D_t\) is the turbulent diffusion coefficient, equal to the eddy viscosity. (Assume unity Schmidt)

- Disadvantage: Counter-gradient diffusion known to occur in some pre-mixed flames.
Molecular Mixing.

- Molecular mixing can be viewed as a process which changes the shape of the scalar PDF without affecting the mean.

- Molecular mixing is modeled by two models: A coalescence/dispersion model (Hsu and Chen) and a relax-to-mean model (Dopazo).

- Advantages of both models is that they are simple and readily adaptable to any number of scalars.

- Disadvantages are the relative lack of physics in the models.

Pressure Term.

- Pressure term model is based on second order closure models for compressible flows (e.g., Sarkar).

\[
\frac{Dp}{Dt} \approx \frac{\partial \langle p \rangle}{\partial t} + \langle U_i \rangle \frac{\partial p}{\partial x_i} + 0.8 \rho(k) \frac{\partial \langle U_i \rangle}{\partial x_i} + 0.15 \rho P_f M_t - 0.2 \rho e M_t^2
\]

- Advantages are that model is tried and tested in finite volume codes. Disadvantage is that only the mean pressure can be used for model. Ideally we would like a stochastic process for two state variables.
Numerics

- Solution of scalar PDF transport equation achieved by a particle based Monte Carlo scheme.

- PDF represented by an ensemble of particles, each with a composition and enthalpy.

- PDF evolves by the motion of these particles in physical, scalar and enthalpy space, by exact and modeled processes. eg. Convection, reaction, mixing.

- Statistics (eg. means) obtained by averaging over ensemble of particles.

Numerical Details - Monte Carlo Scheme

- Module based on cell-centered quantities.

- PDF method is a nodal one. ie. All particles reside at the center of the cell, and can move only to neighboring cells.

- Number of particles at each node is the same and the number remains fixed.

- Statistics obtained by averaging over particles at each node, and also by time-averaging.
Numerics - Convection

- Upwind scheme used for mean convection, and central difference for turbulent convection.

- Evolution achieved by moving particles in from adjacent nodes. Particles are selected at random.

- Fractions of particles are treated by random convection.

\[
\text{if 6.3 particles then } \begin{cases} 
6.0 & 70\% \text{ of the time} \\
7.0 & 30\% \text{ of the time}
\end{cases}
\]

Numerics - Reaction

- Although reaction source term treated exactly, several different numerical schemes are needed. Timing figures are for % of time spent in the PDF part of the code on SPARC II workstation.

- No reaction: For scalar mixing calculations. (Timing: 17.2 %)

- Equilibrium reaction: Assume reaction proceeds at infinite speed. Table of equilibrium composition as a function of mixture fraction obtained from separate CHEMKIN routine. (Timing: 24.4 %)
Numerics - Reaction Cont.

• One-step global reaction schemes. Westbrook and Dryer global reactions integrated for each time-step.
  (Timing: 51.1 %)

• Tabulated reaction increments. Users create their own table of composition increments as a function of scalars using the adaptive tabulation scheme provided, plus the users favourite reduced mechanism.
  (Timing: 58.9 %)

• Chemkin full mechanism integration. Very slow and not recommended except for parallel applications.
  (Timing: 97.8 %)

Numerics - Averaging

• To reduce statistical error in the evaluation of the mean scalar quantities (without increasing the number of particles per node), time averaging is employed.

• A weighted time average is used to give more weight to recent values and less to those in the far past.

\[
\langle \phi \rangle_t = \frac{1}{w_t + 1} (\langle \phi \rangle_t + w_t \langle \phi \rangle_{t-1})
\]

(5)

\[
w_t = c_t (w_{t-1} + 1)
\]

(6)
Numerics: Misc.

- A portable random number generator is now included in the module, set up for 32 bit machines.

- A time step check is now performed to ensure boundedness of the PDF solution. i.e. no negative numbers of particles.

- Rplus/PDF release ported to workstation environment. K-epsilon now standard turbulence model.

Future Work.

- Release of 3D version with new improvements.

- Implementation of parallel processing for distributed cluster environment. (PVM based)

- Include model for another state variable to close PDF modeling.
Attendees List

<table>
<thead>
<tr>
<th>Name</th>
<th>Address</th>
<th>Phone/Fax</th>
<th>E-mail Address</th>
</tr>
</thead>
<tbody>
<tr>
<td>Abraham, Mounir</td>
<td>Cleveland State University Dept. of Mechanical Engineering Cleveland, Ohio</td>
<td>(216) 687-2580</td>
<td></td>
</tr>
<tr>
<td>Afmani, Kumud</td>
<td>ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142</td>
<td>(216) 433-3166 (216) 433-3200 (F)</td>
<td><a href="mailto:fskumud@icompo1.lerc.nasa.gov">fskumud@icompo1.lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Anand, M.S.</td>
<td>Allison Engine Company P.O. Box 420, Speed Code T-14 Indianapolis, IN 46206</td>
<td>(317) 230-2828 (317) 230-3691 (F)</td>
<td><a href="mailto:iemsa@agtomed.com">iemsa@agtomed.com</a></td>
</tr>
<tr>
<td>Ashpis, David</td>
<td>NASA Lewis Research Center 21000 Brookpark Road Mail Stop 5-11 Cleveland, OH 44135</td>
<td>(216) 433-8317 (216) 433-5802 (F)</td>
<td><a href="mailto:ashpis@lerc.nasa.gov">ashpis@lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Befrui, Bizhan</td>
<td>ADAPO 60 Broadhollow Road Melville, NY 11747</td>
<td>(516) 549-2300 (516) 549-2654 (F)</td>
<td><a href="mailto:bizhan@adapo.com">bizhan@adapo.com</a></td>
</tr>
<tr>
<td>Brankovic, Andreja</td>
<td>Pratt &amp; Whitney P.O. Box 108600 Mail Stop 715-89 West Palm Beach, FL 33410</td>
<td>(407) 796-8811 (407) 796-5825 (F)</td>
<td><a href="mailto:brankov@pwfl.com">brankov@pwfl.com</a></td>
</tr>
<tr>
<td>Brewster, B. Scott</td>
<td>Brigham Young University 75 D STB P.O. Box 24212 Provo, Utah 84602-4212</td>
<td>(801) 378-6240 (801) 378-3831 (F)</td>
<td><a href="mailto:brewster@homer.et.byu.edu">brewster@homer.et.byu.edu</a></td>
</tr>
<tr>
<td>Burrus, David L.</td>
<td>GE Aircraft Engines 1 Neumann Way, MDA-404 Cincinnati, OH 45215</td>
<td>(513) 243-2611 (513) 243-2541 (F)</td>
<td><a href="mailto:burrus@co348.ae.ge.com">burrus@co348.ae.ge.com</a></td>
</tr>
<tr>
<td>Chen, J.Y.</td>
<td>University of California at Berkeley 6163 Etcherry Hall Berkeley, CA 94720</td>
<td>(510) 642-3286 (510) 642-6163 (F)</td>
<td><a href="mailto:jjchen@euler.berkeley.edu">jjchen@euler.berkeley.edu</a></td>
</tr>
<tr>
<td>Cheng, Chi-Yang</td>
<td>Dept. of Mechanical Engineering Grove City College Grove City, PA 16127</td>
<td>(412) 458-3367</td>
<td></td>
</tr>
<tr>
<td>Chitsomboon, Tawit</td>
<td>ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142</td>
<td>(216) 962-3106 (216) 433-3200 (F)</td>
<td></td>
</tr>
<tr>
<td>Choudhury, D.</td>
<td>Fluent Inc. 10 Cavendish Court Lebanon, NH 03755</td>
<td>(603) 643-2600</td>
<td></td>
</tr>
<tr>
<td>Choi, Dochul</td>
<td>UTRC East Hartford, CT</td>
<td>(203) 727-7791 (203) 727-7656 (F)</td>
<td></td>
</tr>
<tr>
<td>Coirier, William</td>
<td>NASA Lewis Research Center 21000 Brookpark Road Mail Stop 5-11 Cleveland, OH 44135</td>
<td>(216) 433-5764 (216) 433-5802 (F)</td>
<td><a href="mailto:coirier@lerc.nasa.gov">coirier@lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Dalton, Charles</td>
<td>University of Houston Deans Office, College of Engineering Houston, TX 77204-4814</td>
<td>(713) 743-4205 (713) 743-4214 (F)</td>
<td><a href="mailto:dalton@uh.edu">dalton@uh.edu</a></td>
</tr>
<tr>
<td>Ganjoo, Deepak</td>
<td>Swanson Analysis Systems, Inc. P.O. Box 65, Johnson Road Houston, PA 15342-0065</td>
<td>(412) 873-3055 (412) 764-9494 (F)</td>
<td><a href="mailto:dganjoo@swanson.com">dganjoo@swanson.com</a></td>
</tr>
<tr>
<td>Name</td>
<td>Affiliation</td>
<td>Address</td>
<td>Phone Numbers</td>
</tr>
<tr>
<td>--------------</td>
<td>---------------------------------------</td>
<td>----------------------------------------------</td>
<td>---------------------------------------------</td>
</tr>
<tr>
<td>Georgiadis, Nick</td>
<td>NASA Lewis Research Center</td>
<td>21000 Brookpark Road, M. S. 86-7</td>
<td>(216) 433-3958, (216) 433-2182 (F)</td>
</tr>
<tr>
<td>Giel, Paul W.</td>
<td>Nyma, Inc.</td>
<td>2001 Aerospace Parkway</td>
<td>(216) 977-1340, (216) 977-1269 (F)</td>
</tr>
<tr>
<td>Goldman, Louis J.</td>
<td>NASA Lewis Research Center</td>
<td>21000 Brookpark Road, M. S. 5-11</td>
<td>(216) 433-5845</td>
</tr>
<tr>
<td>Hadid, A.</td>
<td>Rocketdyne, MS IB 59</td>
<td>6633 Canoga Avenue</td>
<td>(818) 718-3405</td>
</tr>
<tr>
<td>Harloff, Gary</td>
<td>NYMA, Inc.</td>
<td>2001 Aerospace Parkway</td>
<td>(708) 491-0200, (708) 869-6495 (F)</td>
</tr>
<tr>
<td>Haroutunian, Vahé</td>
<td>Fluid Dynamics International</td>
<td>500 Davis Street, #600</td>
<td>(216) 962-3146, (216) 962-3200 (F)</td>
</tr>
<tr>
<td>Hayder, Ehtesham</td>
<td>ICOMP/OAI</td>
<td>22800 Cedar Point Road</td>
<td>(513) 243-0589, (513) 243-1343 (F)</td>
</tr>
<tr>
<td>Hunter, Scott</td>
<td>GEAE</td>
<td>1 Neumann Way, M/D A405</td>
<td>(216) 977-1302</td>
</tr>
<tr>
<td>Hsu, Andrew</td>
<td>NYMA, Inc.</td>
<td>2001 Aerospace Parkway</td>
<td>(216) 433-5386, (216) 433-3000 (F)</td>
</tr>
<tr>
<td>Jorgenson, Philip</td>
<td>NASA Lewis Research Center</td>
<td>21000 Brookpark Road, Mail Stop 5-11</td>
<td>(314) 233-591, (314) 777-1328 (F)</td>
</tr>
<tr>
<td>Kral, Linda D.</td>
<td>McDonnell Douglas Aerospace</td>
<td>P.O. Box 516, MC 106-4126</td>
<td>(313) 577-3893</td>
</tr>
<tr>
<td>Lai, Ming-Chia</td>
<td>Wayne State University</td>
<td>Mechanical Engineering Department</td>
<td>(205) 536-6576, (205) 536-6590 (F)</td>
</tr>
<tr>
<td>Leonard, Andy</td>
<td>CFD Research Corporation</td>
<td>3325 Triana Blvd.</td>
<td>(216) 962-3152, (216) 962-3200 (F)</td>
</tr>
<tr>
<td>Liou, William</td>
<td>ICOMP/OAI</td>
<td>22800 Cedar Point Road</td>
<td>(513) 556-1923, (513) 556-3231 (F)</td>
</tr>
<tr>
<td>Loh, C.Y.</td>
<td>University of Cincinnati</td>
<td>M.L. #70</td>
<td>(607) 255-4050</td>
</tr>
<tr>
<td>Lumley, Prof J.</td>
<td>Sibley School of Mechanical and</td>
<td>Aerospace Engineering,</td>
<td>(216) 368-4127</td>
</tr>
<tr>
<td>Mishra, Vinod</td>
<td>Department of Physics</td>
<td>Case Western Reserve University</td>
<td>(216) 433-5850, (216) 433-3000 (F)</td>
</tr>
<tr>
<td>Mularz, Edward</td>
<td>NASA Lewis Research Center</td>
<td>21000 Brookpark Road, M. S. 5-11</td>
<td>(519) 886-8435</td>
</tr>
<tr>
<td>Mueller, C. M.</td>
<td>Advanced Scientific Computing Ltd.</td>
<td>554 Parkside ve, Unit 4</td>
<td>(216) 368-4127</td>
</tr>
<tr>
<td>Name</td>
<td>Address</td>
<td>Phone</td>
<td>Email</td>
</tr>
<tr>
<td>------------------</td>
<td>--------------------------------------------------------------------------</td>
<td>------------------------------</td>
<td>--------------------------------------------</td>
</tr>
<tr>
<td>Norris, Andrew</td>
<td>ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142</td>
<td>(216) 962-3071 (216) 962-3200</td>
<td><a href="mailto:fsandy@icomp01.lerc.nasa.gov">fsandy@icomp01.lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Pal, Shankha</td>
<td>University of Cincinnati 2930 Scotto St., #802 Cincinnati, OH 45219</td>
<td>(513) 556-1923 (513) 556-3231</td>
<td><a href="mailto:spal@uceng.uc.edu">spal@uceng.uc.edu</a></td>
</tr>
<tr>
<td>Papp, John L.</td>
<td>University of Cincinnati 727 Martin Luther King Drive, 1012W Cincinnati, OH 45220</td>
<td>(513) 861-9520 (513) 556-3231</td>
<td><a href="mailto:jpapp@uceng.uc.edu">jpapp@uceng.uc.edu</a></td>
</tr>
<tr>
<td>Prakash, Chander</td>
<td>GE Aircraft Engines One Neumann Way Cincinnati, OH 45215</td>
<td>(513) 243-0788 (513) 243-1343</td>
<td><a href="mailto:prakash@hu004.ae.ge.com">prakash@hu004.ae.ge.com</a></td>
</tr>
<tr>
<td>Raju, Manthena</td>
<td>NYMA, Inc. 2001 Aerospace Parkway Brook Park, OH 44142</td>
<td>(216) 977-1366</td>
<td></td>
</tr>
<tr>
<td>Ramachandra, S.</td>
<td>Ram &amp; Ram Research Associates 27743 Edgepark Boulevard North Olmsted, OH 44070</td>
<td>(216) 779-5886 (216) 979-9160</td>
<td><a href="mailto:cco395@cleveland.freenef.edu">cco395@cleveland.freenef.edu</a></td>
</tr>
<tr>
<td>Razdan, Mohan</td>
<td>Allison Engine Company P.O. Box 420, Speed Code T-14 Indianapolis, IN 46206</td>
<td>(317) 230-6404 (317) 230-3691</td>
<td><a href="mailto:iemkx@agtmeds.com">iemkx@agtmeds.com</a></td>
</tr>
<tr>
<td>Reddy, D.R.</td>
<td>NASA Lewis Research Center 21000 Brookpark Road Mail Stop 5-9 Cleveland, OH 44135</td>
<td>(216) 433-8133</td>
<td></td>
</tr>
<tr>
<td>Richardson, Pamela</td>
<td>NASA Headquarters Washington DC 20546</td>
<td>(202) 358-4631</td>
<td><a href="mailto:pf_richardson@aeromail.hq.nasa.gov">pf_richardson@aeromail.hq.nasa.gov</a></td>
</tr>
<tr>
<td>Schwab, John</td>
<td>NASA Lewis Research Center 21000 Brookpark Road, M. S. 5-11 Cleveland, OH 44135</td>
<td>(216) 433-8446 (216) 433-5802</td>
<td><a href="mailto:jrschwab@lerc.nasa.gov">jrschwab@lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Shabbir, Aamir</td>
<td>ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142</td>
<td>(216) 962-3149 (216) 962-3200</td>
<td><a href="mailto:fsaamir@icomp01.lerc.nasa.gov">fsaamir@icomp01.lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Shieh, Geoffrey</td>
<td>Department of Mech &amp; Aero Engineering SUNY at Buffalo Buffalo, NY 14260</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shih, T.-H.</td>
<td>ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142</td>
<td>(216) 962-3161 (216) 962-3200</td>
<td><a href="mailto:fsshih@icomp01.lerc.nasa.gov">fsshih@icomp01.lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Sindir, Munir</td>
<td>Rocketdyne, MS IB 39 6633 Canoga Avenue Canoga Park, CA 91303</td>
<td>(818) 586-1627 (818) 586-0588</td>
<td><a href="mailto:aks@cfdr.com">aks@cfdr.com</a></td>
</tr>
<tr>
<td>Singhal, Ashok</td>
<td>CFD Research Corporation 3325 Triana Blvd. Huntsville, AL 35805</td>
<td>(205) 536-6576 (205) 536-6590</td>
<td></td>
</tr>
<tr>
<td>Smith, Brian</td>
<td>Lockheed - Fort Worth Company P.O. Box 748 Fort Worth, TX 76101</td>
<td>(817) 763-2836</td>
<td></td>
</tr>
<tr>
<td>Sonnemeier, James</td>
<td>University at Buffalo 309 Jarvis Hall Amherst, NY 14260</td>
<td>(716) 645-2593 (716) 645-3875</td>
<td><a href="mailto:jrs@santa.eng.buffalo.edu">jrs@santa.eng.buffalo.edu</a></td>
</tr>
<tr>
<td>Steffen, Chris</td>
<td>NASA Lewis Research Center 21000 Brookpark Road Mail Stop 5-11 Cleveland, OH 44135</td>
<td>(216) 433-8508 (216) 433-5802</td>
<td><a href="mailto:fsstef@faust.lerc.nasa.gov">fsstef@faust.lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Name</td>
<td>Address</td>
<td>Phone</td>
<td>Email</td>
</tr>
<tr>
<td>--------------------</td>
<td>--------------------------------------------------------------------------</td>
<td>----------------</td>
<td>-------------------------------------</td>
</tr>
<tr>
<td>Steinthorsson, Erlendur</td>
<td>ICOMP/OAI&lt;br&gt;22800 Cedar Point Road&lt;br&gt;Brook Park, OH 44142</td>
<td>(216) 962-3162&lt;br&gt;(216) 962-3200 (F)</td>
<td><a href="mailto:fssstein@icomp01.lerc.nasa.gov">fssstein@icomp01.lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Sung, Chao-Ho</td>
<td>David Taylor Model Basin&lt;br&gt;Bethesda, MD 20084-5000</td>
<td>(301) 227-1865&lt;br&gt;(301) 227-4599 (F)</td>
<td><a href="mailto:trldale@euler.eng.buffalo.edu">trldale@euler.eng.buffalo.edu</a></td>
</tr>
<tr>
<td>Syed, Saadat</td>
<td>Pratt &amp; Whitney&lt;br&gt;M.S. 715-89&lt;br&gt;P.O. Box 109600&lt;br&gt;West Palm Beach FL 33410-9600</td>
<td>(407) 796-3560&lt;br&gt;(407) 796-5825 (F)</td>
<td></td>
</tr>
<tr>
<td>Taulbee, Dale B.</td>
<td>State University N.Y. at Buffalo&lt;br&gt;Dept. Mech &amp; Aerospace Engineering&lt;br&gt;Buffalo, NY 14260</td>
<td>(716) 645-2593&lt;br&gt;(716) 645-3875 (F)</td>
<td></td>
</tr>
<tr>
<td>Tekriwal, P.</td>
<td>GE-CRD&lt;br&gt;P.O. Box 8, Bldg. K-1, ES108&lt;br&gt;Schenectady, NY 12301</td>
<td>(518) 387-6732</td>
<td></td>
</tr>
<tr>
<td>Tew, Roy</td>
<td>NASA Lewis Research Center&lt;br&gt;21000 Brookpark Road&lt;br&gt;Mail Stop 301-2&lt;br&gt;Cleveland, OH 44135</td>
<td>(216) 433-8471&lt;br&gt;(216) 433-6133 (F)</td>
<td><a href="mailto:tew@lerc.nasa.gov">tew@lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Tselepidakis, Dimitri</td>
<td>Fluent Inc.&lt;br&gt;10 Cavendish Court&lt;br&gt;Lebanon, NH 03766</td>
<td>(603) 643-2600&lt;br&gt;(603) 643-3967 (F)</td>
<td><a href="mailto:dpt@fluent.com">dpt@fluent.com</a></td>
</tr>
<tr>
<td>To, Wai-Ming</td>
<td>NYMA&lt;br&gt;2001 Aerospace Parkway&lt;br&gt;Mail Stop 5-9&lt;br&gt;Brook Park, OH 44142</td>
<td>(216) 433-5937&lt;br&gt;(216) 433-8864 (F)</td>
<td><a href="mailto:fswmto@perch.lerc.nasa.gov">fswmto@perch.lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Tolpadi, Anil K.</td>
<td>General Electric R&amp;D Center&lt;br&gt;P.O. Box 8, M.S. K1-ES206&lt;br&gt;Schenectady, NY 12301</td>
<td>(518) 387-5787&lt;br&gt;(518) 387-7104 (F)</td>
<td><a href="mailto:tolpadi@crd.ge.com">tolpadi@crd.ge.com</a></td>
</tr>
<tr>
<td>Tran, Le</td>
<td>NYMA, Inc.&lt;br&gt;2001 Aerospace Parkway&lt;br&gt;Brookpark, OH 44142</td>
<td>(216) 977-1357&lt;br&gt;(216) 977-1269 (F)</td>
<td><a href="mailto:totran@lerc.nasa.gov">totran@lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Van Doormaal, J. P.</td>
<td>Advanced Scientific Computing Ltd.&lt;br&gt;554 Parkside ve, Unit 4&lt;br&gt;Waterloo, Ontario&lt;br&gt;Canada N2L 5Z4</td>
<td>(519) 886-8435</td>
<td></td>
</tr>
<tr>
<td>Wu, Jie</td>
<td>ICOMP/OAI&lt;br&gt;22800 Cedar Point Road&lt;br&gt;Brook Park, OH 44142</td>
<td>(216) 962-3096&lt;br&gt;(216) 962-3200 (F)</td>
<td><a href="mailto:fsjiewu@icomp01.lerc.nasa.gov">fsjiewu@icomp01.lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Yang, Zhigang</td>
<td>ICOMP/OAI&lt;br&gt;22800 Cedar Point Road&lt;br&gt;Brook Park, OH 44142</td>
<td>(216) 962-3093&lt;br&gt;(216) 962-3200 (F)</td>
<td><a href="mailto:fsyang@icomp01.lerc.nasa.gov">fsyang@icomp01.lerc.nasa.gov</a></td>
</tr>
<tr>
<td>Yungster, Shaye</td>
<td>ICOMP/OAI&lt;br&gt;22800 Cedar Point Road&lt;br&gt;Brook Park, OH 44142</td>
<td>(216) 962-3162&lt;br&gt;(216) 962-3200 (F)</td>
<td></td>
</tr>
<tr>
<td>Zerke, Ronald</td>
<td>GE Aircraft Engines&lt;br&gt;One Neuman Way&lt;br&gt;Mail Drop A405&lt;br&gt;Cincinnati, OH 45215</td>
<td>(513) 243-2470&lt;br&gt;(513) 243-1343 (F)</td>
<td></td>
</tr>
<tr>
<td>Zhu, Jiang</td>
<td>ICOMP/OAI&lt;br&gt;22800 Cedar Point Road&lt;br&gt;Brook Park, OH 44142</td>
<td>(216) 962-3095&lt;br&gt;(216) 962-3200 (F)</td>
<td><a href="mailto:fszhu@icomp01.lerc.nasa.gov">fszhu@icomp01.lerc.nasa.gov</a></td>
</tr>
</tbody>
</table>
1. AGENCY USE ONLY (Leave blank)

4. TITLE AND SUBTITLE
   Industry-Wide Workshop on Computational Turbulence Modeling

6. AUTHOR(S)
   A. Shabbir, compiler

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
   National Aeronautics and Space Administration
   Lewis Research Center
   Cleveland, Ohio 44135−3191

8. PERFORMING ORGANIZATION REPORT NUMBER
   E−9295

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
   National Aeronautics and Space Administration
   Washington, D.C. 20546−0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
    NASA CP−10165
    ICOMP−94−30
    CMOTT−94−9

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT
    Unclassified - Unlimited
    Subject Category 34
    This publication is available from the NASA Center for Aerospace Information, (301) 621−0390.

12b. DISTRIBUTION CODE

13. ABSTRACT (Maximum 200 words)
    This publication contains the presentations made at the Industry-Wide Workshop on Computational Turbulence Modeling, which was hosted by ICOMP/LeRC, and took place on October 6−7, 1994 at the Ohio Aerospace Institute. The purpose of the workshop was to initiate the transfer of technology developed at Lewis Research Center to industry and to discuss the current status and the future needs of turbulence models in industrial CFD.

14. SUBJECT TERMS
    Turbulence models

15. NUMBER OF PAGES
    286

16. PRICE CODE
    A13