NASA Conference Publication 10165
ICOMP–94–30; CMOTT–94–9

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

N95- 27883

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.

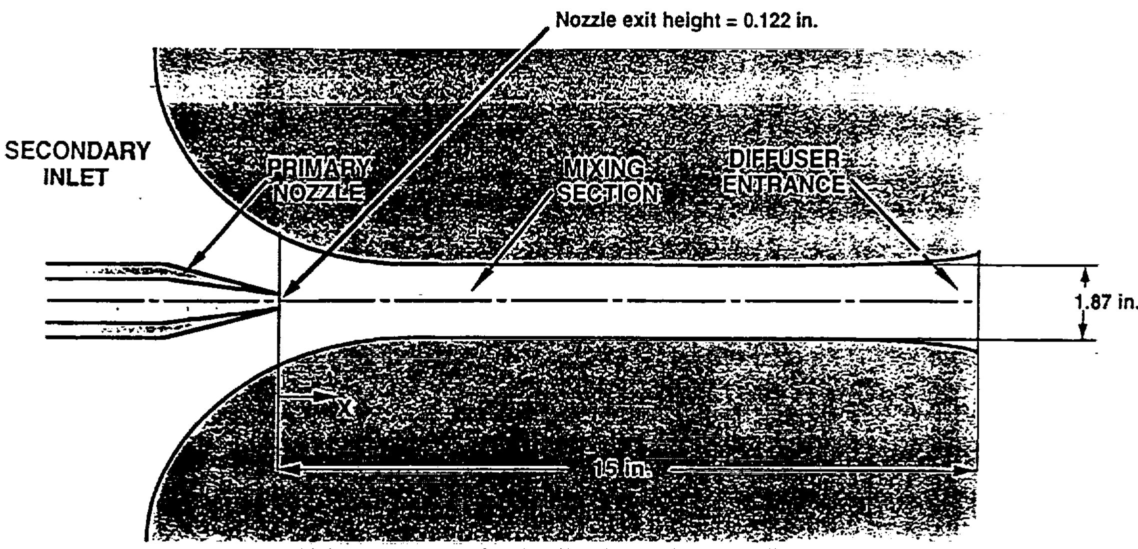
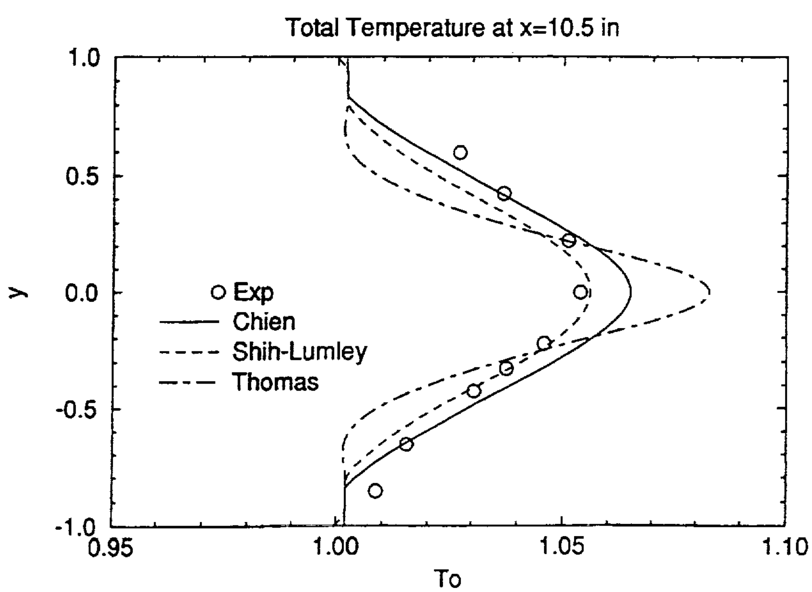
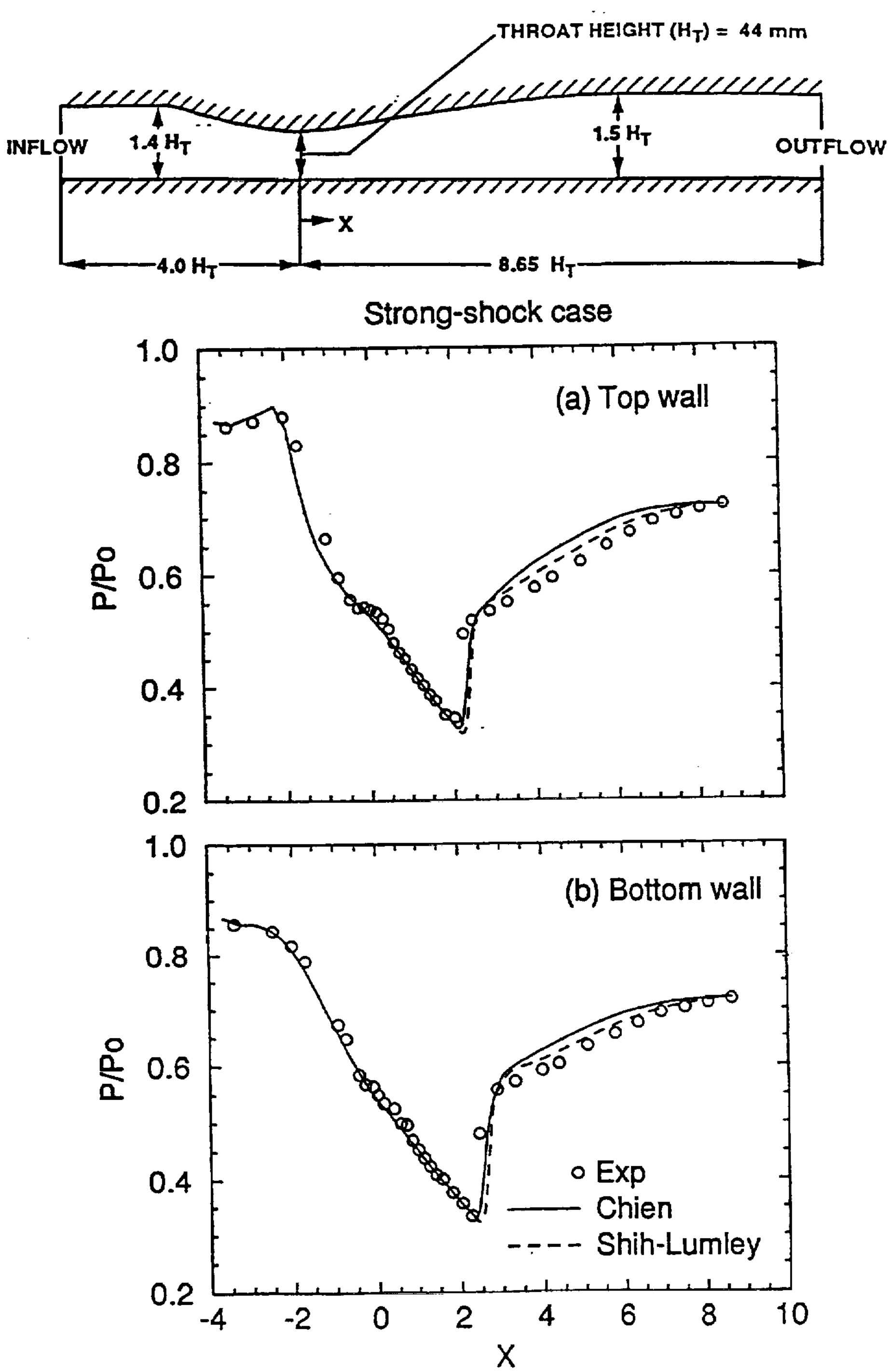


Fig. 3. Schematic of ejector nozzle test case.




THROAT HEIGHT (H_T) = 44 mm
INFLOW
1.4 H_T
1.5 H_T
OUTFLOW
X
4.0 H_T
8.65 H_T
Strong-shock case
(a) Top wall
(b) Bottom wall
P/Po
X
Exp
Chien
Shih-Lumley

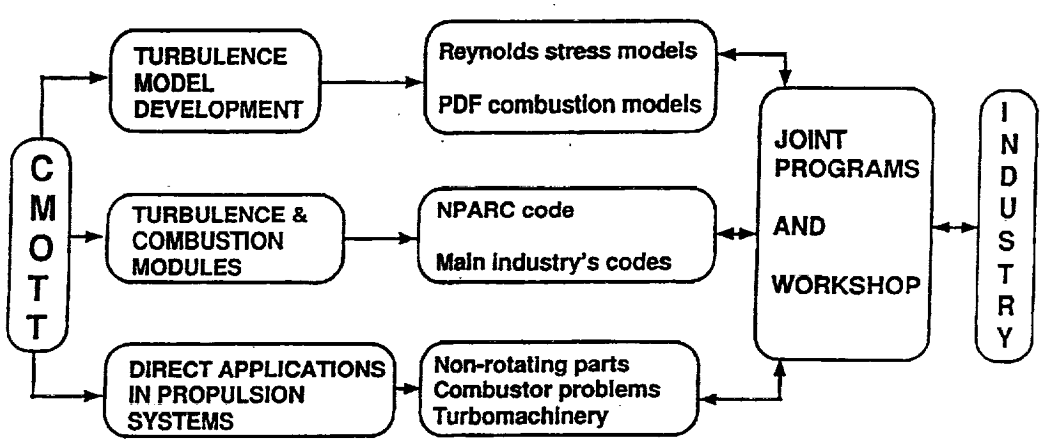
Collaboration with Industry and Technology Transfer

- 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

$$-\overline{u_i u_j} = \nu_T(U_{i,j} + U_{j,i}) - \frac{2}{3}k\delta_{ij}, \qquad \nu_T = C_\mu f_\mu \frac{k^2}{\varepsilon}$$

$$\frac{Dk}{Dt} = T^{(k)} + P^{(k)} - \varepsilon + \text{W.C.}, \quad \frac{D\varepsilon}{Dt} = T^{(\varepsilon)} + P^{(\varepsilon)} - D^{(\varepsilon)} + \text{W.C.}$$

  - ◇ 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 ε 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

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

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

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

◇ f_μ, f_1, f_ϕ are functions of $R = k/S\nu$, which is tensorially invariant

◇ $C_\mu = \frac{1}{A_0 + A_s U^*\ k/\varepsilon}$, which ensures realizability for normal stresses

◇ Φ 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

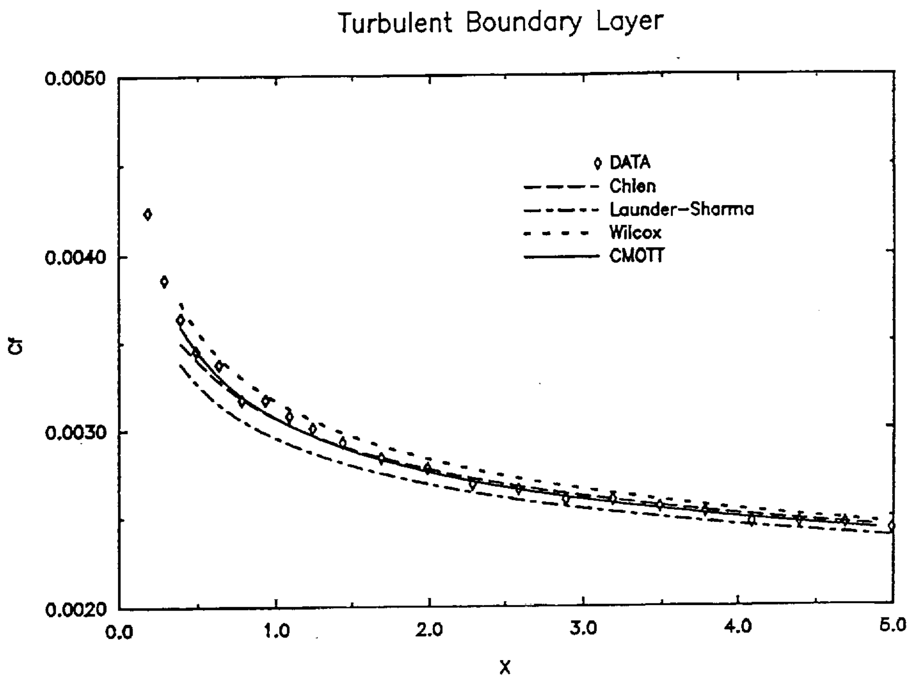
◇ Complex flows related to industrial applications

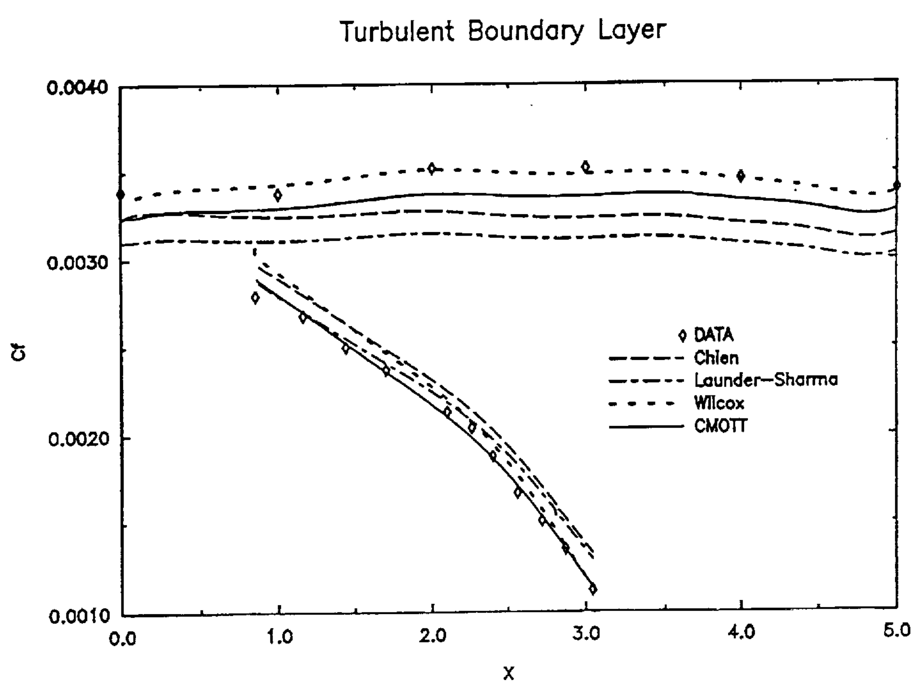
Models:

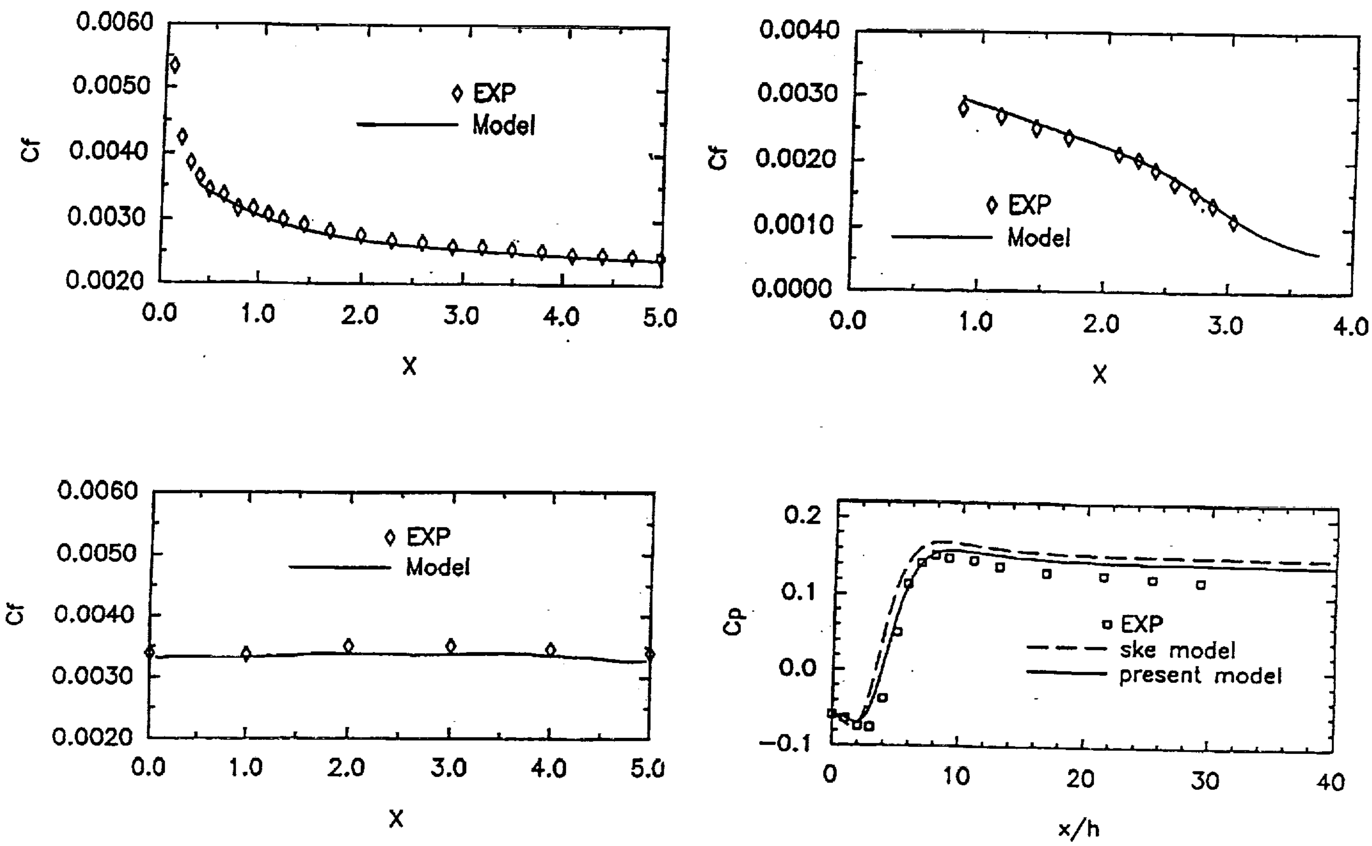
◇ Launder-Sharma, Lam-Bremhorst, Chien,
Nagano-Hishida, ...

◇ $k-\omega$ model (Wilcox)

◇ CMOTT $k-\varepsilon$ model


Turbulent Boundary Layer
0.0050
0.0040
0.0030
0.0020
Cf
0.0
1.0
2.0
3.0
4.0
5.0
X
DATA
Chien
Launder-Sharma
Wilcox
CMOTT

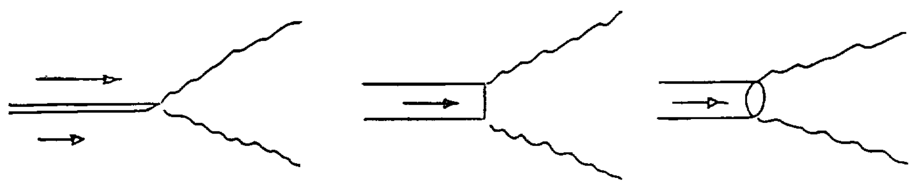

Turbulent Boundary Layer
0.0040
0.0030
0.0020
0.0010
Cf
0.0
1.0
2.0
3.0
4.0
5.0
X
DATA
Chien
Launder-Sharma
Wilcox
CMOTT



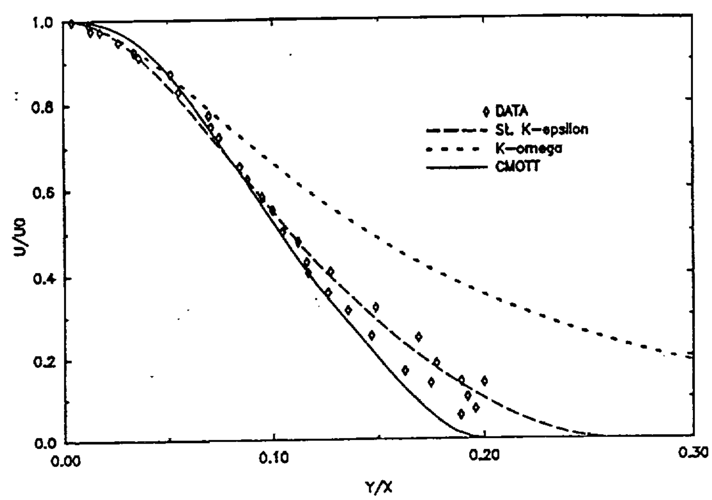
Present model with the modified wall function

Spreading Rate of Free Shear Flows

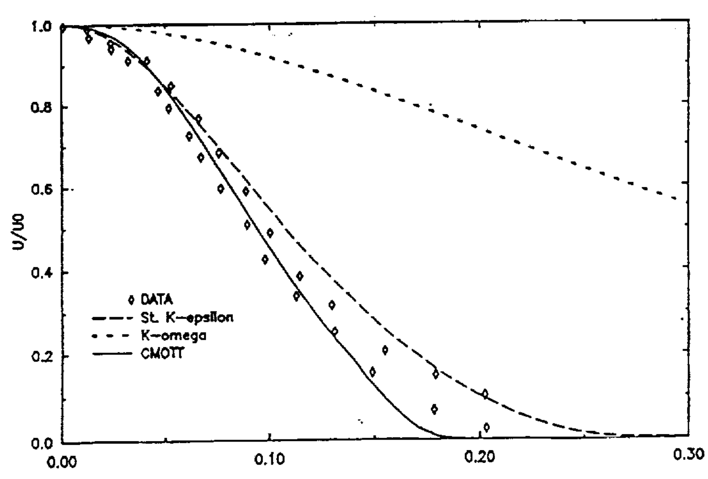
	exp.	st. $k-\epsilon$	Chien	$k-\omega$	CMOTT
Planar Jet	**0.10-0.11**	**0.108**	**0.098**	**0.14***	**0.102**
Round Jet	**0.085-0.095**	**0.116**	**0.104**	**0.32***	**0.095**
Mixing Layer	**0.13-0.17**	**0.152**	**0.152**	**0.16***	**0.154**



Planar Jet



Round Jet



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)

$$\frac{\overline{u_i u_j}}{k}(P - \varepsilon) = -\overline{u_i u_k} U_{j,k} - \overline{u_j u_k} U_{i,k} - \frac{1}{\rho}(\overline{p_{,i} u_j + p_{,j} u_i}) - 2\nu \overline{u_{i,k} u_{j,k}}$$

Comments:

  - * Assumption: $\overline{u_i u_j}/k = \text{Const.}$, $(\overline{u_i u_j u_k})_{,k} = (\overline{k u_i})_{,i} = 0$
  - * 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

$$\overline{u_i u_j} = \frac{2}{3}k\delta_{ij} + 2a_2\frac{K^2}{\varepsilon}(U_{i,j} + U_{j,i} - \frac{2}{3}U_{i,i}\delta_{ij}) + 2a_4\frac{K^3}{\varepsilon^2}(U^2_{i,j} + U^2_{j,i} - \frac{2}{3}\Pi_1\delta_{ij})$$
$$+ 2a_6\frac{K^3}{\varepsilon^2}(U_{i,k}U_{j,k} - \frac{1}{3}\Pi_2\delta_{ij}) + 2a_7\frac{K^3}{\varepsilon^2}(U_{k,i}U_{k,j} - \frac{1}{3}\Pi_2\delta_{ij})$$

$$+ 2a_8\frac{K^4}{\varepsilon^3}(U_{i,k}U^2_{j,k} + U^2_{i,k}U_{j,k} - \frac{2}{3}\Pi_3\delta_{ij}) + 2a_{10}\frac{K^4}{\varepsilon^3}(U_{k,i}U^2_{k,j} + U_{k,j}U^2_{k,i} - \frac{2}{3}\Pi_3\delta_{ij})$$
$$+ 2a_{12}\frac{K^5}{\varepsilon^4}(U^2_{i,k}U^2_{j,k} - \frac{1}{3}\Pi_4\delta_{ij}) + 2a_{13}\frac{K^5}{\varepsilon^4}(U^2_{k,i}U^2_{k,j} - \frac{1}{3}\Pi_4\delta_{ij})$$
$$+ 2a_{14}\frac{K^5}{\varepsilon^4}(U_{i,k}U_{l,k}U^2_{l,j} + U_{j,k}U_{l,k}U^2_{l,i} - \frac{2}{3}\Pi_5\delta_{ij})$$
$$+ 2a_{16}\frac{K^6}{\varepsilon^5}(U_{i,k}U^2_{l,k}U^2_{l,j} + U_{j,k}U^2_{l,k}U^2_{l,i} - \frac{2}{3}\Pi_6\delta_{ij})$$
$$+ 2a_{18}\frac{K^7}{\varepsilon^6}(U_{i,k}U_{l,k}U^2_{l,m}U^2_{j,m} + U_{j,k}U_{l,k}U^2_{l,m}U^2_{i,m} - \frac{2}{3}\Pi_7\delta_{ij})$$

- RDT and realizability constraints (Reynolds, Lumley)
- CMOTT algebraic Reynolds stress model

$$\overline{u_i u_j} = \frac{2}{3}k\delta_{ij} - C_\mu\frac{k^2}{\varepsilon}2S^*_{ij} + 2C_2\frac{k^3}{\varepsilon^2}(-S^*_{ik}\Omega^*_{kj} + \Omega^*_{ik}S^*_{kj})$$

$$k_{,t} + U_j k_{,j} = [(\nu + \frac{\nu_t}{\sigma_k})k_{,j}]_{,j} - \overline{u_i u_j}U_{i,j} - \varepsilon$$
$$\varepsilon_{,t} + U_j\varepsilon_{,j} = [(\nu + \frac{\nu_t}{\sigma_\varepsilon})\varepsilon_{,j}]_{,j} - C_{\varepsilon 1}\frac{\varepsilon}{k}\overline{u_i u_j}U_{i,j} - C_{\varepsilon 2}\frac{\varepsilon^2}{k}$$

where

$$C_\mu = \frac{1}{A_0 + A^*_s\frac{U^* k}{\varepsilon}}, \qquad C_2 = \frac{\sqrt{1 - 9C^2_\mu(\frac{S^* k}{\varepsilon})^2}}{C_0 + 6\frac{S^* k}{\varepsilon}\frac{\Omega^* k}{\varepsilon}}$$
$$\nu_t = C_\mu\frac{k^2}{\varepsilon}, \quad A_0 = 6.5, \quad C_0 = 1.0$$
$$C_{\varepsilon 1} = 1.44, \quad C_{\varepsilon 2} = 1.92, \quad \sigma_k = 1, \quad \sigma_\varepsilon = 1.3$$

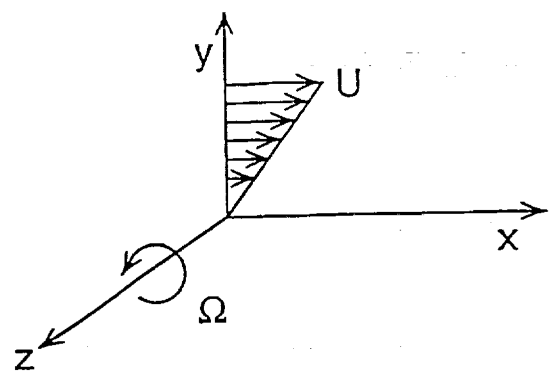
- **Validation**

  ◇ **Rotating homogeneous shear flows**

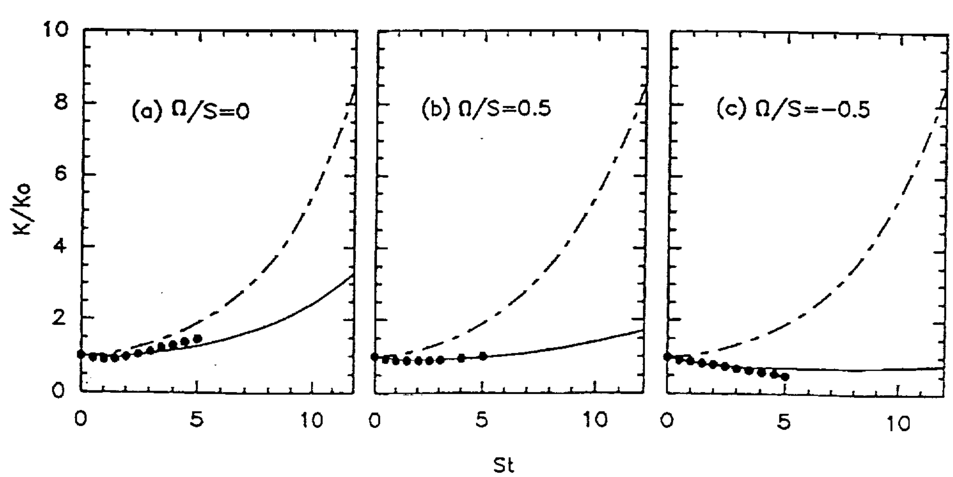
  ◇ **Backward-facing step flows**

  ◇ **Confined jets**

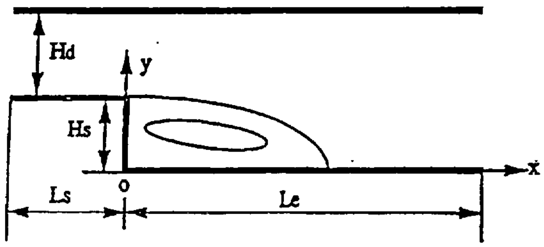
  ◇ **Complex flows related to industrial applications**



Configuration of rotating homogeneous shear flow



Evolution of turbulent kinetic energy with time.
—— : present model; — - — : SKE; • : LES



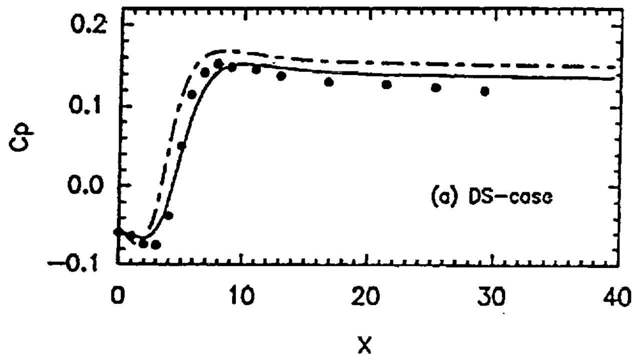
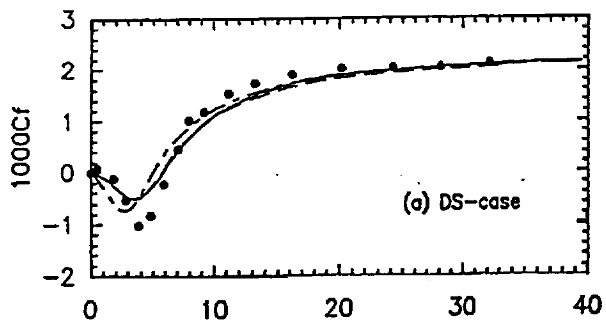
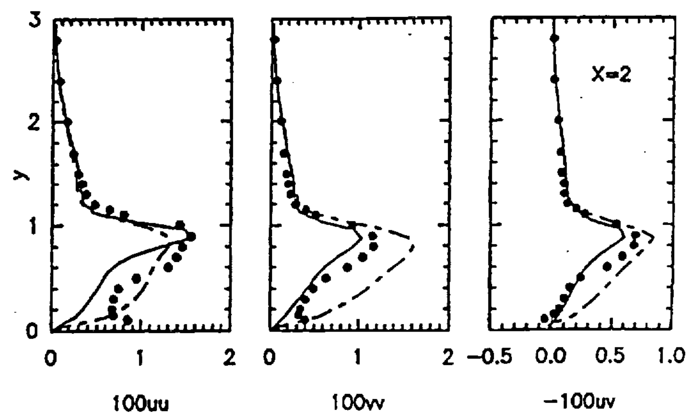


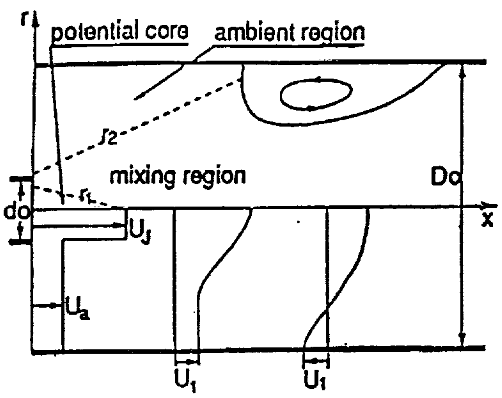
Table Comparison of the reattachment points

Case	measurement	SKE	PRESENT
DS	6.1	4.99	5.82
KKJ	7± 0.5	6.35	7.35

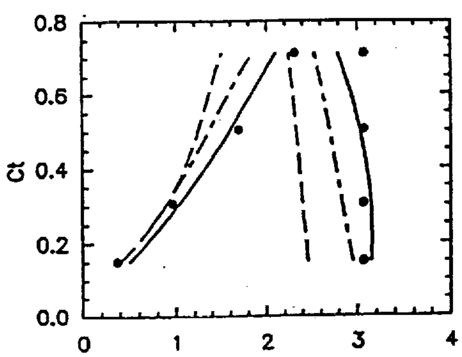




—— : present model; — - —: SKE; • : EXP

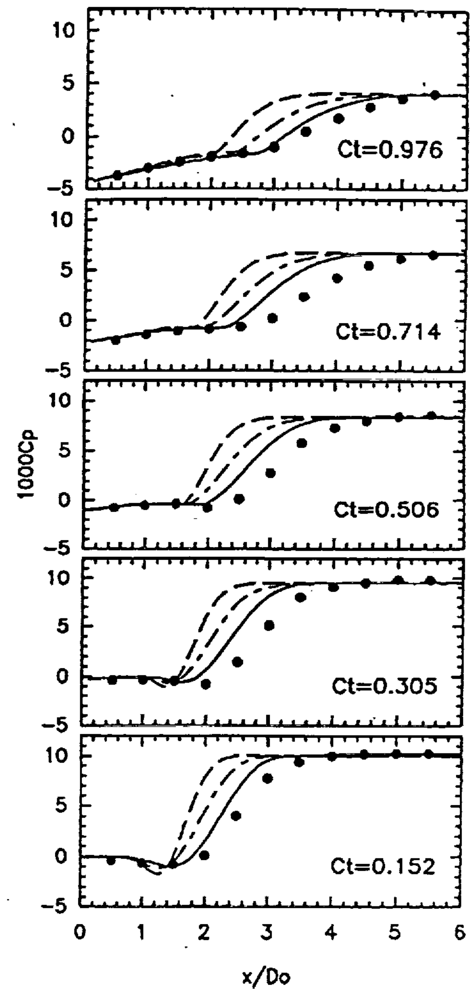


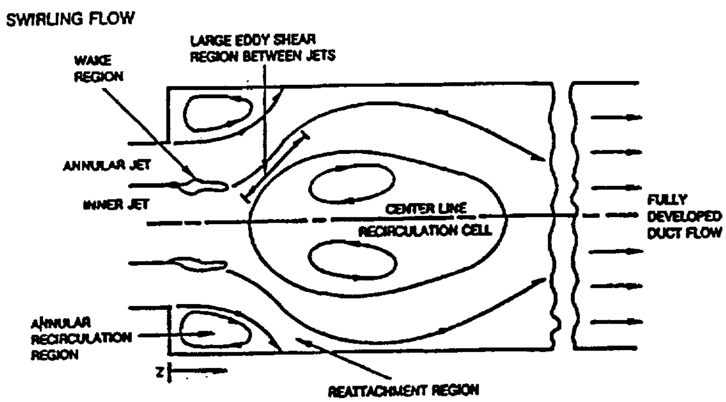
Flow configuration and notations

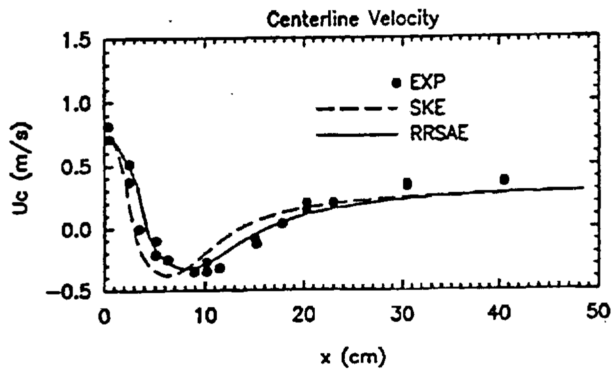


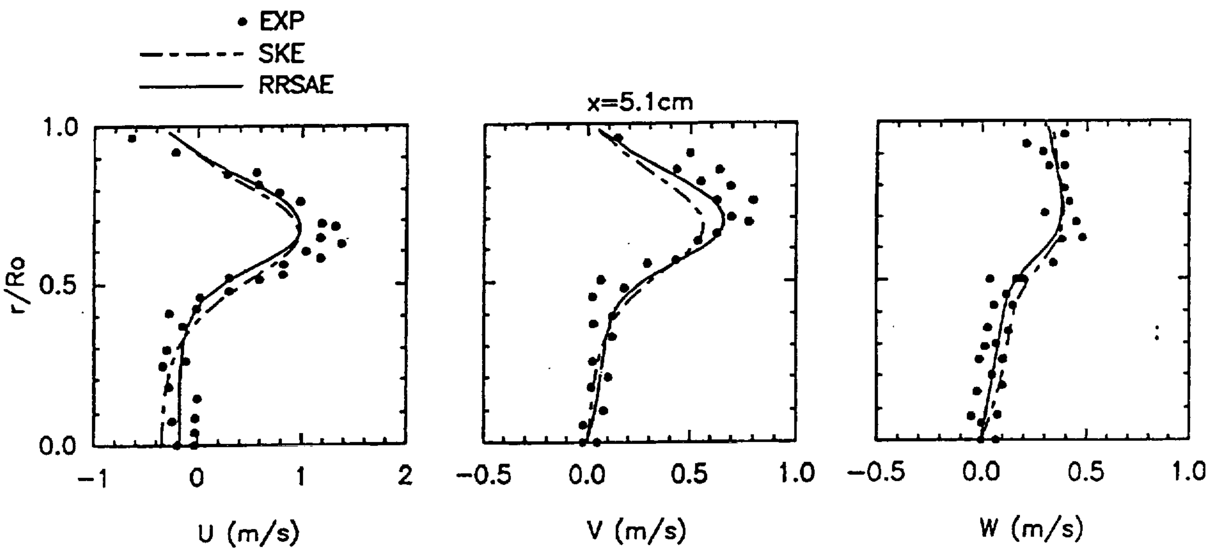
Separation and reattachment points

• EXP
SKE
RNG
RRSAE




SWIRLING FLOW
LARGE EDDY SHEAR REGION BETWEEN JETS
WAKE REGION
ANNULAR JET
INNER JET
CENTER LINE RECIRCULATION CELL
FULLY DEVELOPED DUCT FLOW
ANNULAR RECIRCULATION REGION
z
REATTACHMENT REGION


Centerline Velocity
EXP
SKE
RRSAE
Uc (m/s)
x (cm)
1.5
1.0
0.5
0.0
−0.5
0
10
20
30
40
50


EXP
SKE
RRSAE
x=5.1cm
r/Ro
1.0
0.5
0.0
−1
0
1
2
U (m/s)
−0.5
0.0
0.5
1.0
V (m/s)
W (m/s)

Scalar turbulence model

- Objective

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

  ◇ A new scalar flux constitutive relation

  ◇ A new scalar dissipation rate model equation

$$\overline{u_i\theta} = - C_\lambda \frac{k^2}{\epsilon}(\frac{2}{r})^{1/2}\Theta_{,i} + \frac{k^3}{\epsilon^2}(\frac{2}{r})^{1/2}(a_2 U_{i,j} + a_3 U_{j,i})\Theta_{,j}$$

$$U_j \frac{\partial \overline{\theta^2}}{\partial x_j} = (\frac{\alpha_T}{\sigma_t}\overline{\theta^2}_{,j})_{,j} - 2\overline{u_i\theta}\frac{\partial \Theta}{\partial x_i} - 2\epsilon_\theta$$

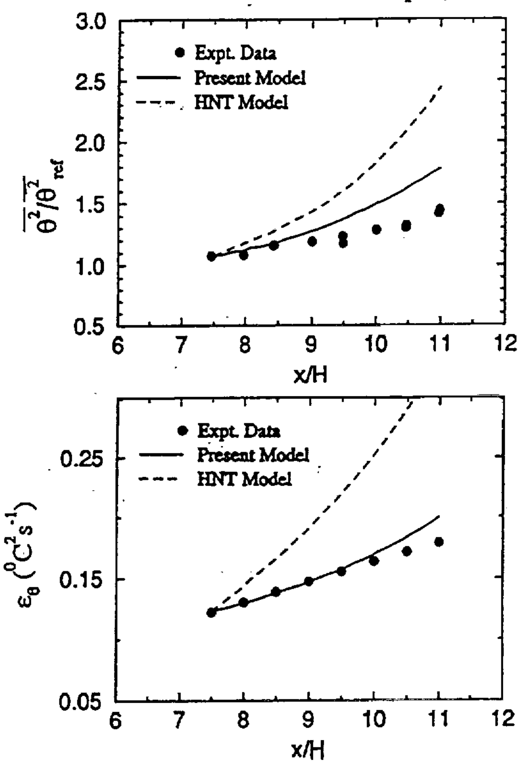
$$U_j \frac{\partial \epsilon_\theta}{\partial x_j} = (\frac{\alpha_T}{\sigma_\phi}\epsilon_{\theta,j})_{,j} + C_{\theta 1}\epsilon_\theta S + C_{\theta 2}\sqrt{\frac{\epsilon_\theta \epsilon}{Pr}} S_T - C_{\theta 3}\frac{\epsilon_\theta \epsilon}{k}$$

$$C_\lambda = \frac{(2 + 2r + 0.5r^2)}{26 + 3.2\eta^2 + 2\xi^2}$$

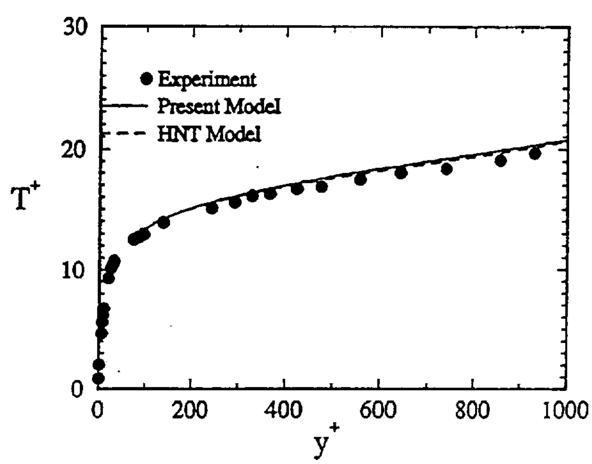
$$S_T = \sqrt{\Theta_{,i}\Theta_{,i}}, \quad S = \sqrt{2S_{ij}S_{ij}}, \quad \eta = Sk/\epsilon, \quad \xi = \tfrac{k}{\epsilon}(\tfrac{k}{\overline{\theta^2}})^{1/2} S_T, \quad r = \tfrac{2k}{\epsilon}\tfrac{\epsilon_\theta}{\overline{\theta^2}}$$

$$C_{\theta 1} = C_1 - 0.13, \quad C_{\theta 2} = 0.63, \quad C_{\theta 3} = C_2 - 1, \quad \sigma_t = 1.0, \quad \sigma_\phi = 1.8$$

Tavoularis & Corrsin Expt (1981)



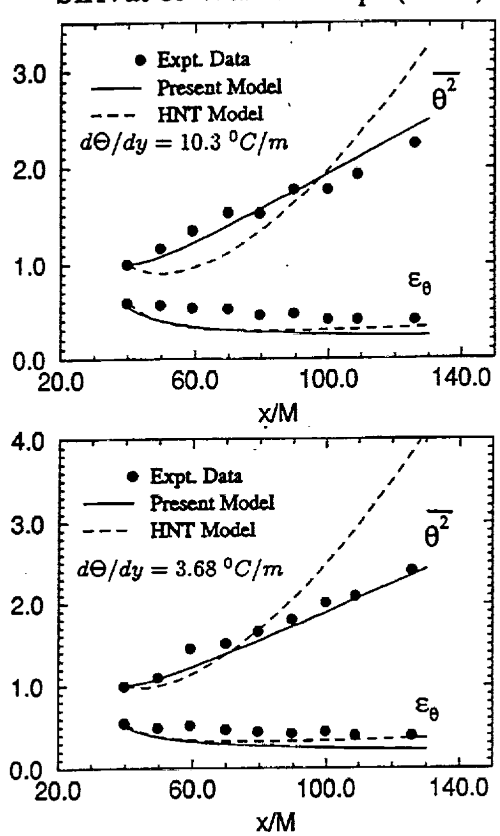
Flat plate boundary layer with constant surface temperature



- 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

Sirivat & Warhaft Expt (1983)



Second Order Closure Models

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

- Objective
  - ◇ To assess existing models
  - ◇ To find the direction of improving closure models
- Basic model forms

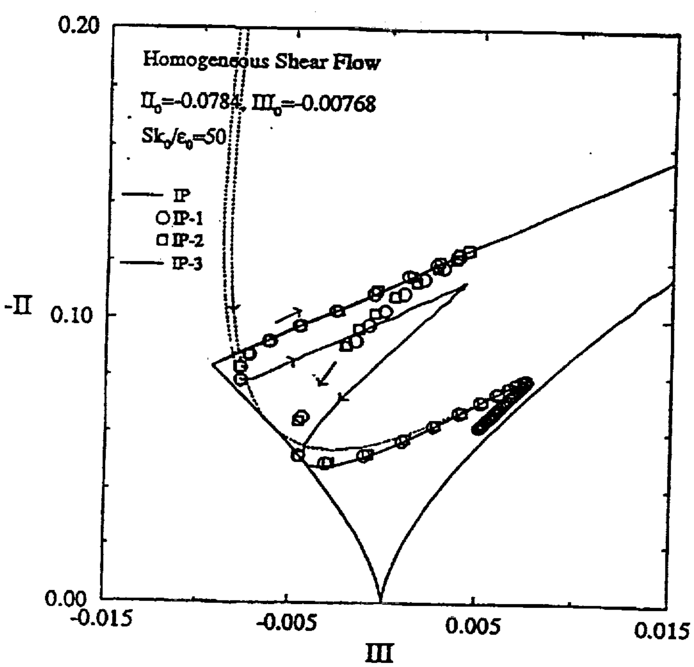
$$\Pi_{ij}^{\mathrm{Rapid}} = F_{ij}(S_{ij}, \overline{u_i u_j}),$$

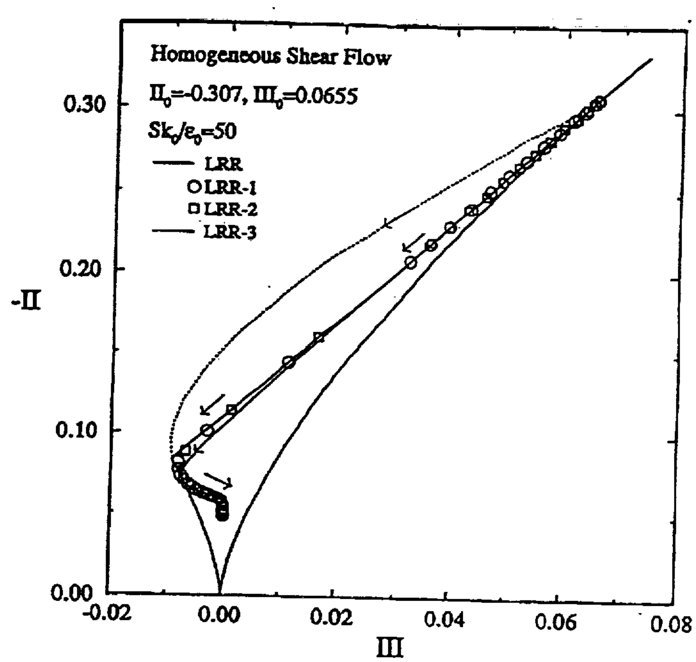
$$\Pi_{ij}^{\mathrm{Return}} = F_{ij}(\overline{u_i u_j}, \nu, k, \varepsilon),$$

$$T_{ij} = F_{ij}\left((\overline{u_i u_j})_{,k}, k, \varepsilon\right)$$

- General comments on second order closures:
  - ◇ The model, $\Pi_{ij}^{\mathrm{Rapid}}$, is relatively well developed compared with other terms
  - ◇ The model, $\Pi_{ij}^{\mathrm{Return}}$, 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-ε 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

$$\bar{\rho}\frac{\overline{\mathrm{D}}\widetilde{k_p}}{\overline{\mathrm{D}}t} = \frac{\partial}{\partial y}[(\overline{\mu} + \frac{\mu_T}{\sigma_{\widetilde{k_p}}})\frac{\partial \widetilde{k_p}}{\partial y}] \; + \; \mu_T(\frac{\partial \widetilde{u}}{\partial y})^2 \; - \; \bar{\rho}\widetilde{\epsilon_p} \; + \; \mathrm{fc}_1$$

$$\bar{\rho}\frac{\overline{\mathrm{D}}\widetilde{\epsilon_p}}{\overline{\mathrm{D}}t} = \frac{\partial}{\partial y}[(\overline{\mu} + \frac{\mu_T}{\sigma_{\widetilde{\epsilon_p}}})\frac{\partial \widetilde{\epsilon_p}}{\partial y}] + \; Cp_1\frac{\widetilde{\epsilon_p}}{\widetilde{k_p}}\mu_T(\frac{\partial \widetilde{u}}{\partial y})^2 \; - \; Cp_2\bar{\rho}\frac{\widetilde{\epsilon_p}^2}{\widetilde{k_p}} \; + \; \mathrm{fc}_2$$

- fc_1 – exchanges between the turbulent kinetic energy and internal energy
- fc_2 – increased spectral energy transfer due to compressibility effects

☐ Small Scale

$$\bar{\rho}\frac{\overline{\mathrm{D}}\widetilde{k_t}}{\overline{\mathrm{D}}t} = \frac{\partial}{\partial y}[(\overline{\mu} + \frac{\mu_T}{\sigma_{\widetilde{k_t}}})\frac{\partial \widetilde{k_t}}{\partial y}] + \bar{\rho}\widetilde{\epsilon_p} - \bar{\rho}\widetilde{\epsilon_t}$$

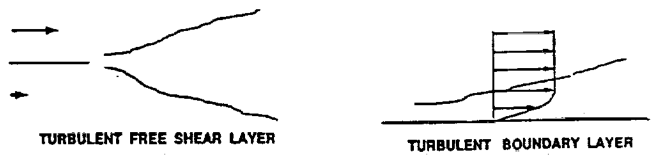
$$\bar{\rho}\frac{\overline{\mathrm{D}}\widetilde{\epsilon_t}}{\overline{\mathrm{D}}t} = \frac{\partial}{\partial y}[(\overline{\mu} + \frac{\mu_T}{\sigma_{\widetilde{\epsilon_t}}})\frac{\partial \widetilde{\epsilon_t}}{\partial y}] + Ct_1\bar{\rho}\frac{\widetilde{\epsilon_t}\widetilde{\epsilon_p}}{\widetilde{k_t}} - Ct_2\bar{\rho}\frac{\widetilde{\epsilon_t}^2}{\widetilde{k_t}}$$

☐ Eddy Viscosity

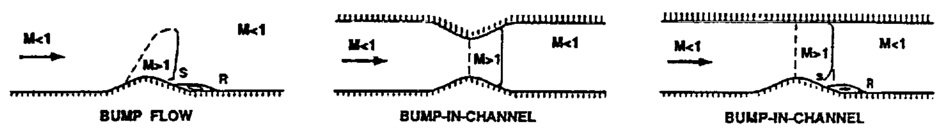
$$\mu_T \; \approx \; \bar{\rho}\; u\; l \; \approx \; \bar{\rho}(\widetilde{k_p} + \widetilde{k_t})^{\frac{1}{2}}\frac{(\widetilde{k_p} + \widetilde{k_t})^{\frac{3}{2}}}{\widetilde{\epsilon_p}}$$

Model Evaluation

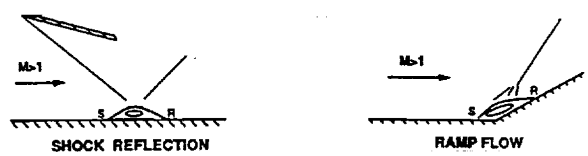
- Turbulent Shear Flow



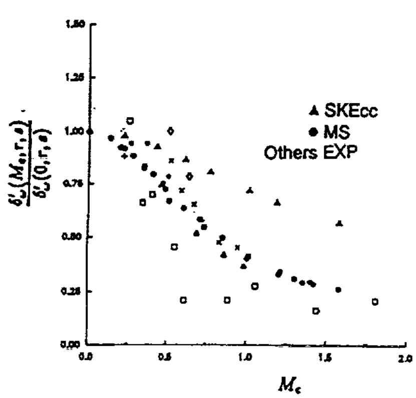
- Shock/Turbulent-Boundary-Layer Interactions
  - ◇ transonic flow

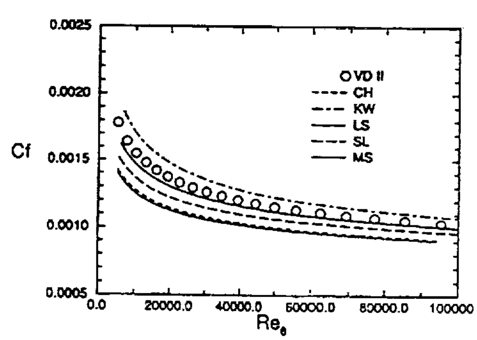


  - ◇ supersonic flow

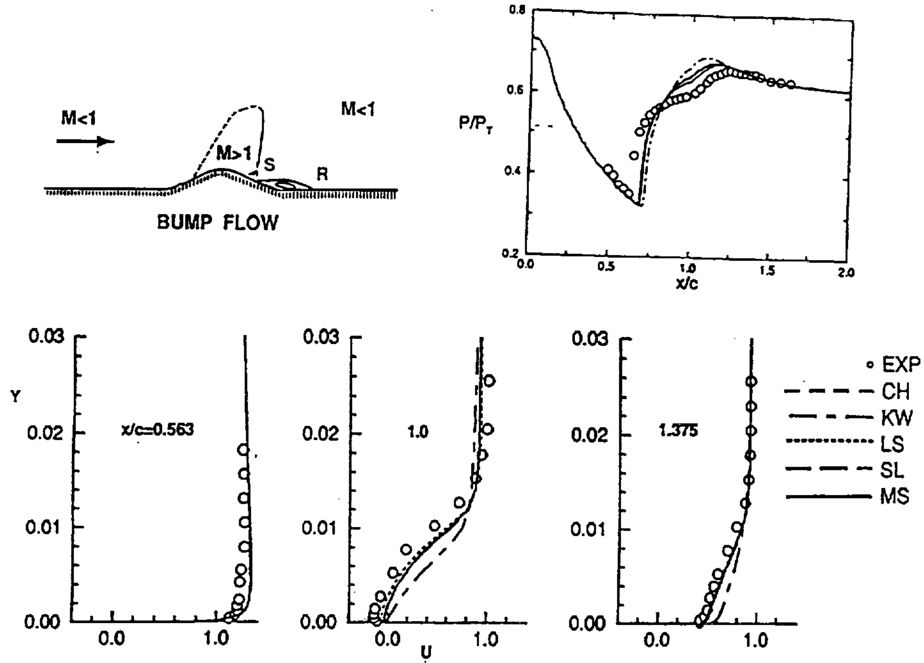


Compressible Turbulent Shear Flow

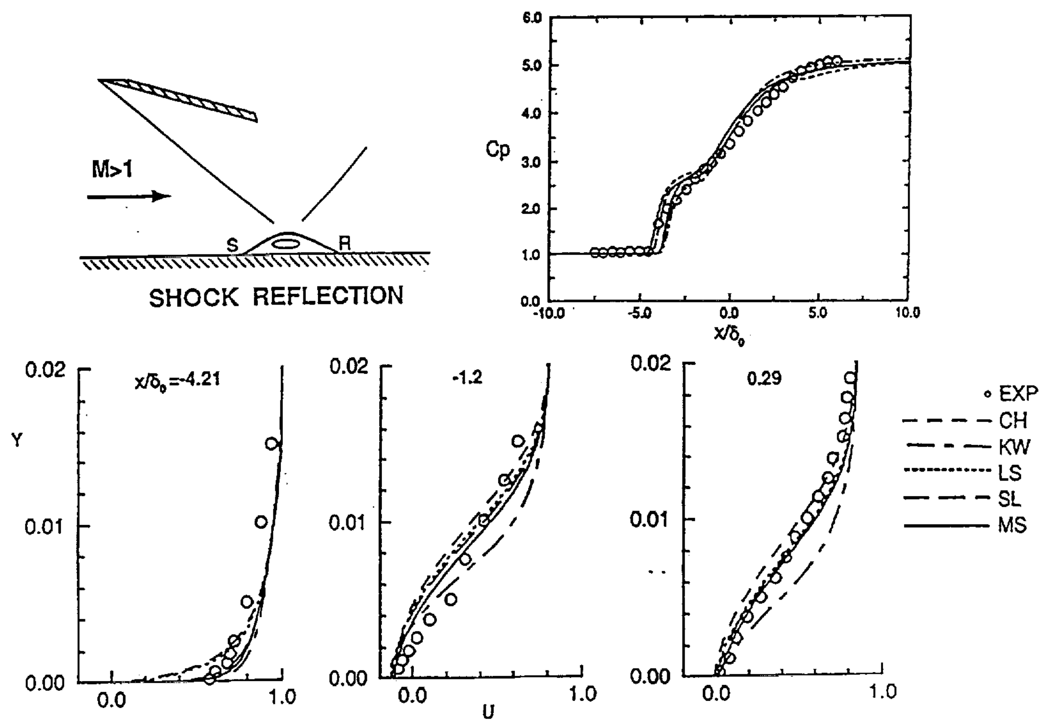




Flow over a Bump—Bachalo and Johnson (1979)

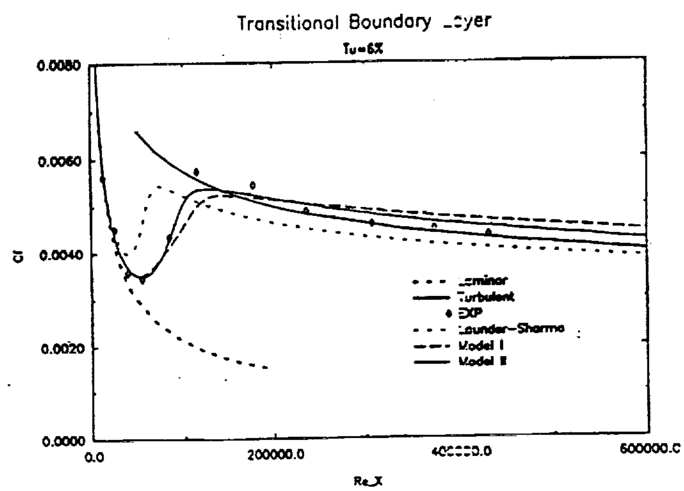


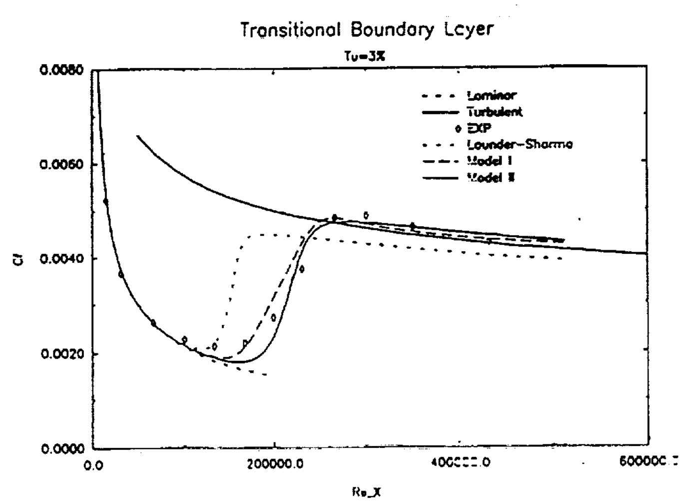
Shock Reflection–Reda *et. al* (1973-1977)



Bypass transition models

- Objective:
  - ◇ Develop transition models for flows with free stream turbulence
- Approach:
  - ◇ Using K-ε model as the base model
  - ◇ Introduce effective intermittency to either the eddy viscosity or the k-ε model equations

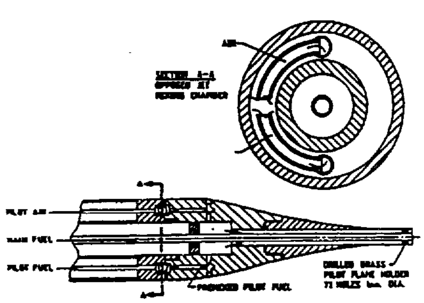




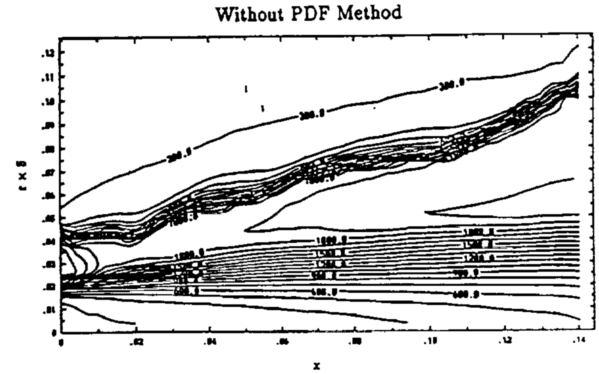
PDF modeling of turbulent reacting flows

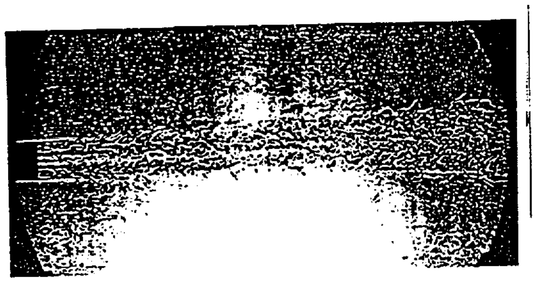
- Objective:
  - ◇ Develop models that can accurately simulate finite chemical reactions in turbulent flows.
  - ◇ Develop and validate independent PDF models.
  - ◇ Technology transfer.
- Approach:
  - ◇ Joint pdf for scalar compositions.
  - ◇ Moment closure schemes for velocity field.
  - ◇ Develop hybrid solver consisting of Monte Carlo method and finite-difference/finite-volume method.

BURNER JET

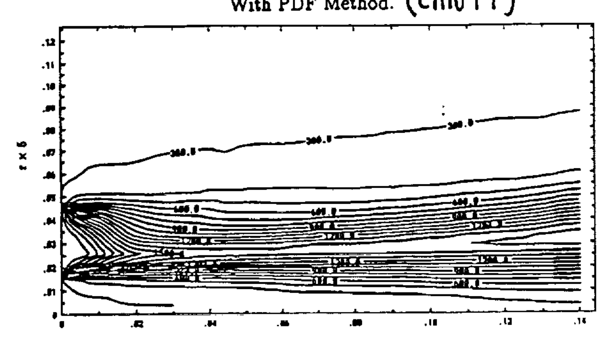


Without PDF Method





With PDF Method. (CMOTT)



TURBULENCE MODEL DEVELOPMENT AND APPLICATION AT LOCKHEED FORT WORTH COMPANY

N95- 27884

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 - l Two Equation Turbulence Models

Advantages of using kl or l instead of ε or ω

kl and l equations are easier to resolve numerically than ε equation

Dissipation Length Scale is an integral length scale

- **Can derive equation for volume integral of two point correlation function.**
- **Theoretical ε equation is dominated by small scales**

k - kl and k - l agree better with compressible boundary layer data than does k - ε

Disadvantage - current formulation requires calculation of distance to walls

<u>k - kl model</u>

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

<u>k - l model</u>

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

The k – kl Model Wall Function

Wall layer model derived from and consistent with the k – kl 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 – kl model

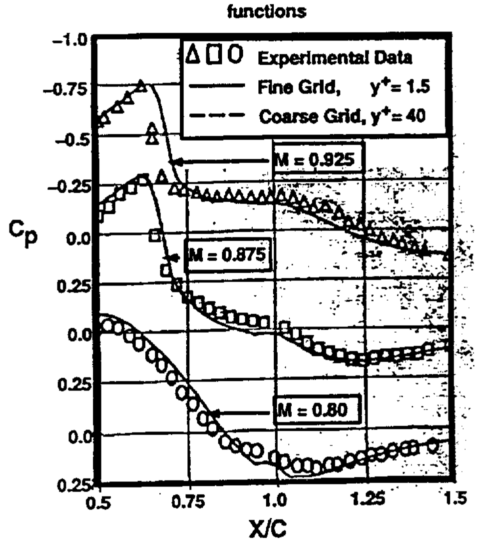
Advantages of wall functions

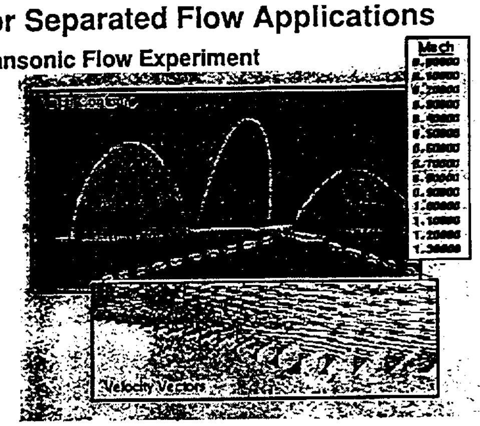
- Reduces number of necessary grid points
- Reduces number of iterations to converge steady state solution 60 – 90%

Wall Functions are Accurate for Separated Flow Applications

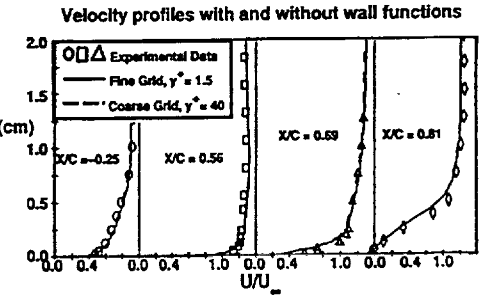
Axisymmetric Bump, Transonic Flow Experiment

Accurate predictions with and without wall functions





Velocity profiles with and without wall functions



The k – l Model with Near Wall Model

kl equation is transformed exactly to an l equation

Advantages of k – l formulation

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

k – l 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

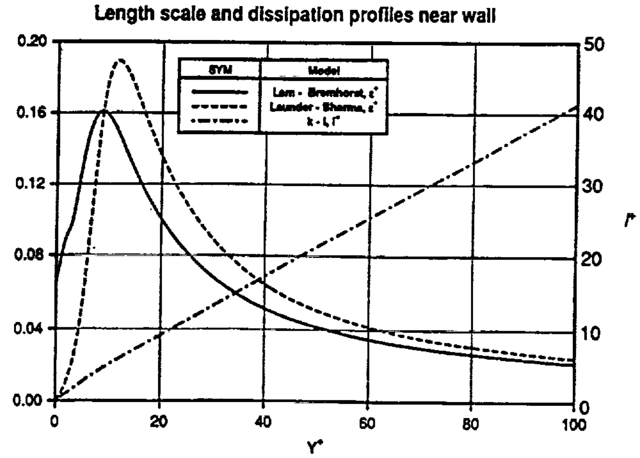
l Equation Much Easier to Resolve than ε Equation

ε equation requires fine grid from wall to y^+ of 20 to resolve peak

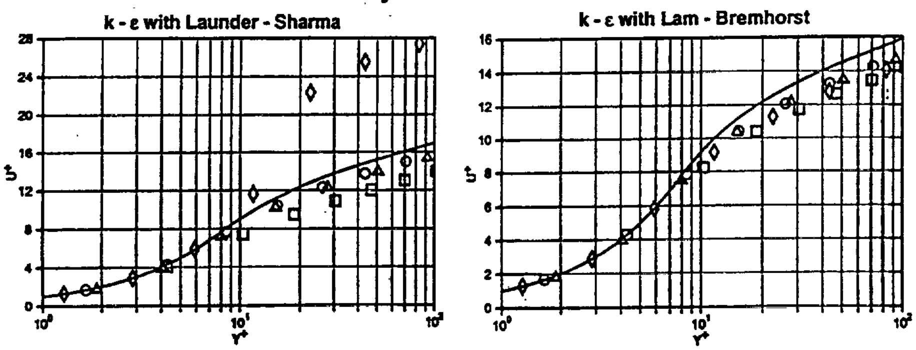
- Exclusion of near wall viscous dissipation term aggravates problem
- Logarithmic region, $\varepsilon \propto 1/y$

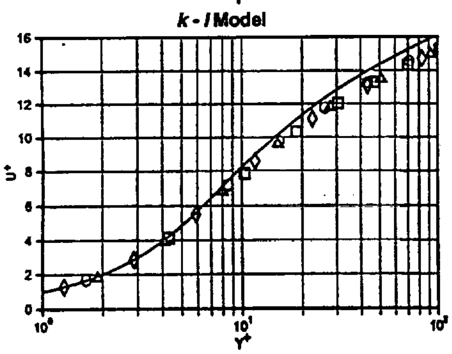
l equation is nearly linear near wall - much less sensitive to grid resolution

Length scale and dissipation profiles near wall



Resolution Study with k - ε and *k* - *l* Models





Symbol	Number of Grid Points	Stretching Rate from wall	Y+, first grid point
———	151	1.04	0.033
□	11	1.4	4.3
○	11	1.6	1.65
△	11	1.8	0.67
◊	11	1.9	0.44

Sample Applications:

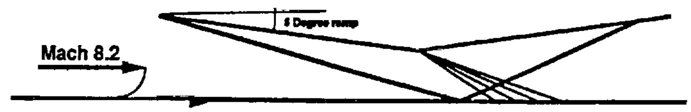
Mach 8 Shock Wave Turbulent Boundary Layer Interactions

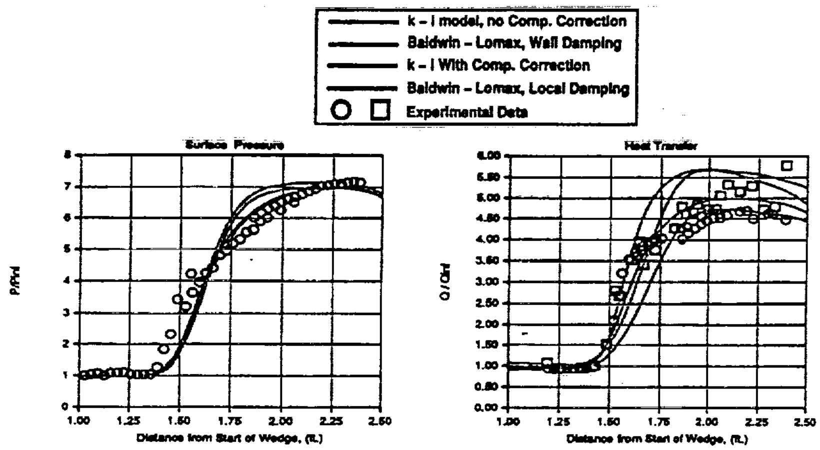
F–16 Inlet Derivitive, Isolated Duct Study

Multi–slot Ejector

F110 Nozzle Drag Reduction Study

k – l Model With Compressibility Correction gives Best Prediction For Mach 8 Shock Boundary Layer Interaction

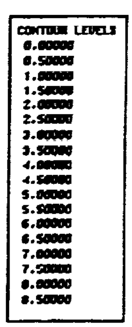


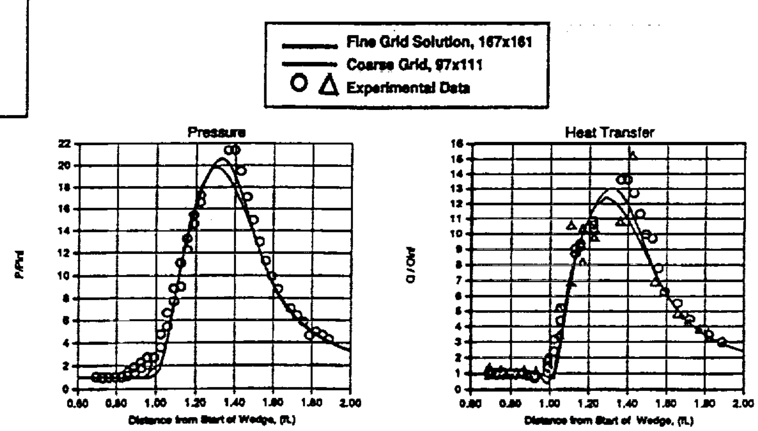


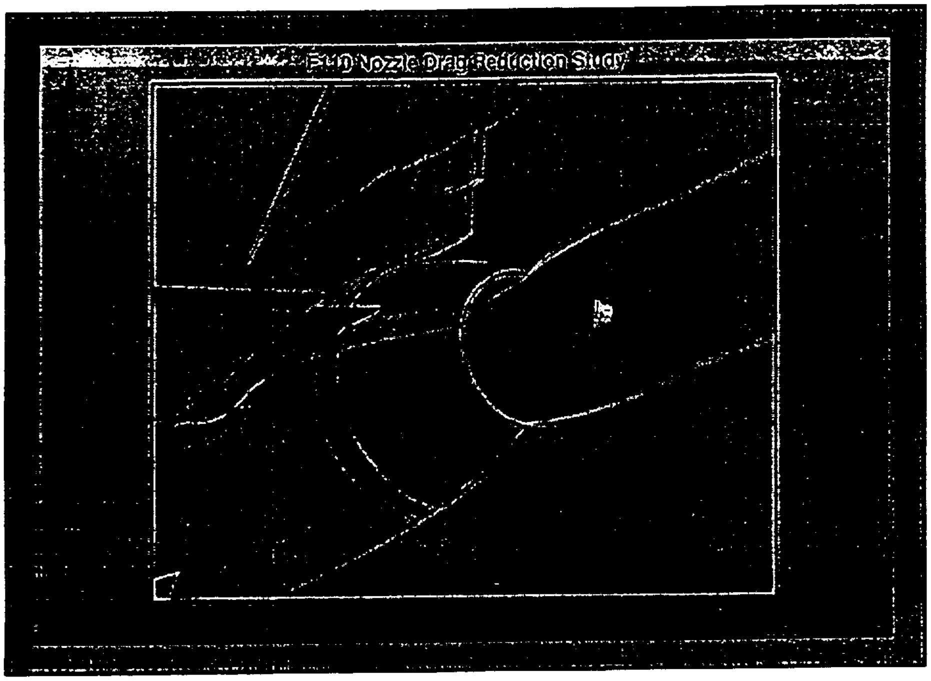
The k – l Model Predicts Turbulent Shock – Wave Boundary Layer Interaction Well

Mach 8, 10 Degree Wedge Generator

2D case, Separated Flow

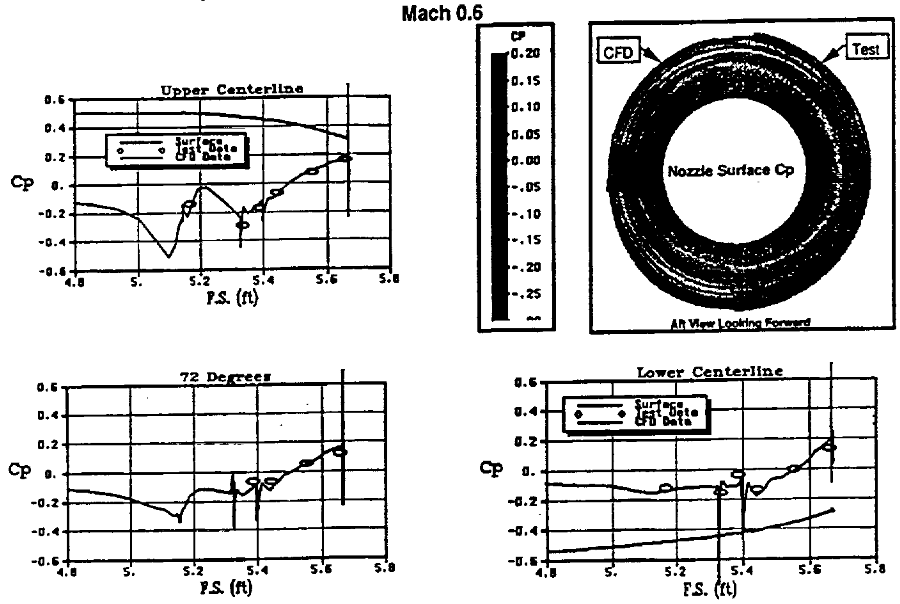




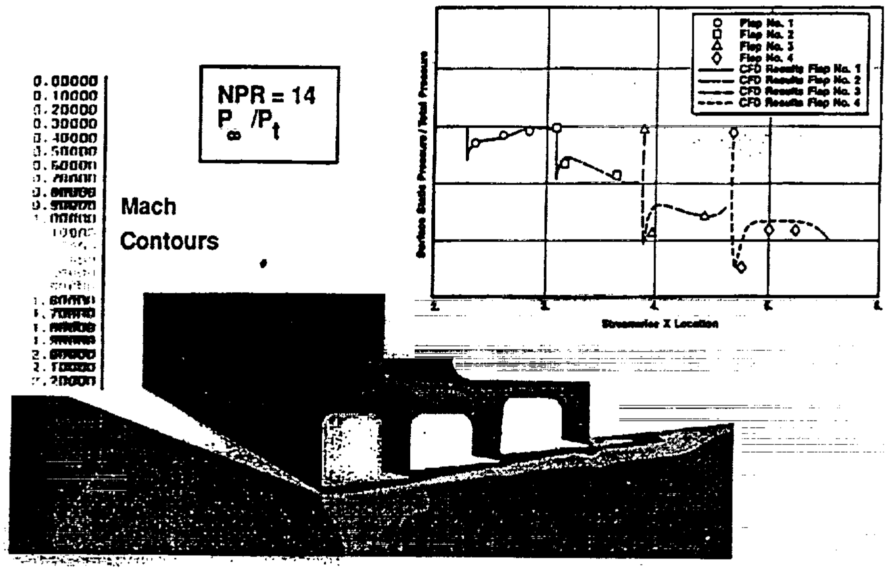


Afterbody/Nozzle Pressure Distributions Match Test Data

Mach 0.6



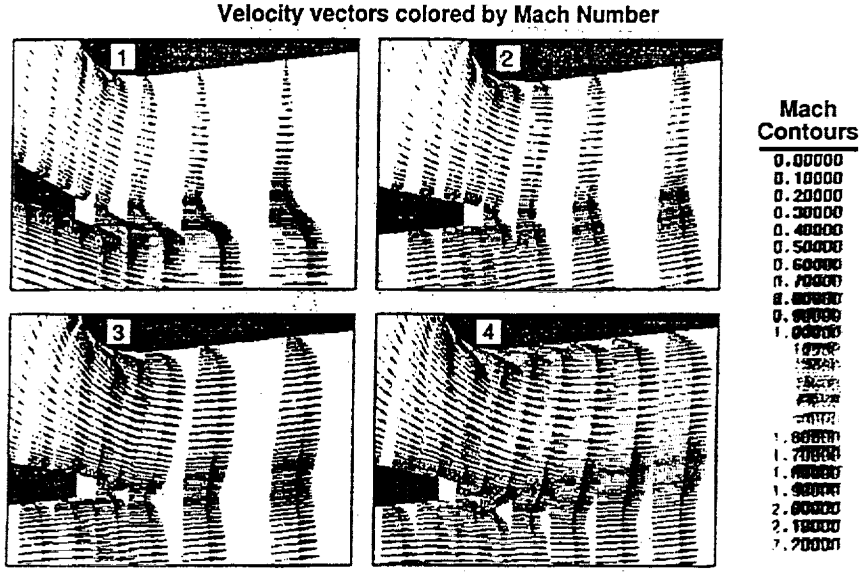
Good Predictions of Multi - Slot Ejector Obtained with k - kl Model



Mach Contours

k - kl Model Predicts Entrainment Effects Near Slots

Velocity vectors colored by Mach Number



Summary

Computationally efficient $k - l$ 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

N95- 27885

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

CONTENTS:

INTRODUCTION:

- Indications are that the standard $k-\varepsilon$ 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 Runga–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–ε turbulence model with standard wall functions.

FILM COOLING SIMULATION (CONT'D):

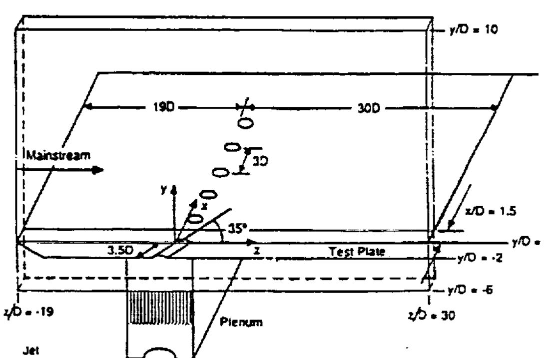


Figure 1. Essential features of experimental film cooling configuration showing overall extent of computation domain and coordinate system.

FILM COOLING SIMULATION (CONT'D):

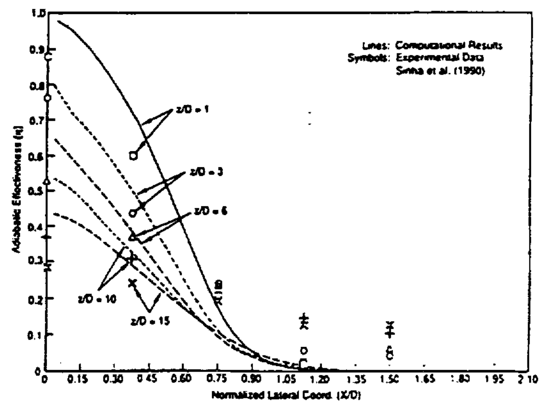


Figure 14. Lateral variation of adiabatic effectiveness from computations and experiments for M=0.5 at five streamwise stations.

FILM COOLING SIMULATION (CONT'D):

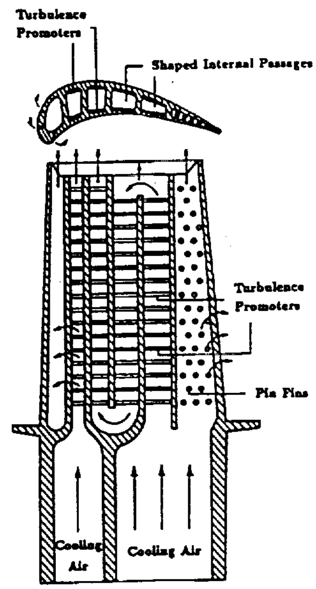
- Summary of Results:
  - The flowfield is dominated by a strong three–way coupling between the plenum, film–hole, and cross–stream regions.
  - Flow within the film hole is extremely complex, with counter–rotating vortices and local jetting effects.
  - A comparison of computed and experimental film 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.

TURBULATED PASSAGE SIMULATION:

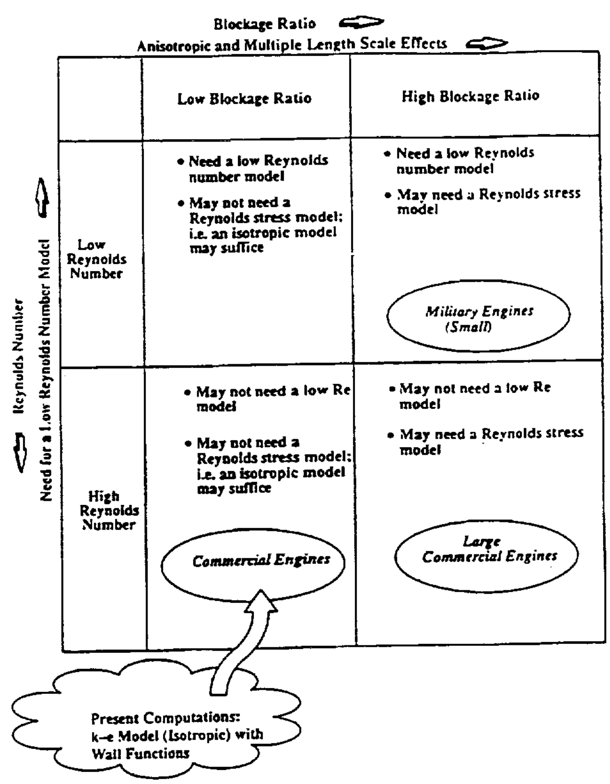
- Modern high–performance turbine blades are cooled by internal radially–rotating serpentine passages.
- The air flowing through these passages is exposed to very large Coriolis and centripetal body forces which induce strong secondary flows and buoyant effects.
- These effects tend to increase heat 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 uniformily 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.



Cooling concepts of a modern multipass turbine blade



OVERALL CONCLUSIONS:

- Application of the standard k−ε turbulence model with wall function is not adequate for accurate CFD simulation of aerodynamic perfor mance and heat transfer in the primary gas path of a gas turbine engine
- New models are required in the near−wall region which include mor physics than wall functions. The two−layer modeling approach ap pears attractive because of its computational economy.
- In addition, improved CFD simulation of film cooling and turbine blad internal cooling passages will require anisotropic turbulence models
- New turbulence models must be practical in order to have a significar impact on the engine design process.
- A coordinated turbulence modeling effort between NASA center would be beneficial to the gas turbine industry.

LIST OF REFERENCES:

1. Zerkle, R. D., and Lounsbury, R. J., 1989, "Freestream Turbulence Effect on Turbin Airfoil Heat Transfer," AIAA *Journal of Propulsion*, Vol. 5, pp. 82−88.

2. Turner, M. G., and Jennions, I. K., 1993, "An Investigation of Turbulence Modelin in Transonic Fans Including a Novel Implementation of an Implicit k−ε Turbulence Mod el," ASME *Journal of Turbomachinery*, Vol. 115, pp. 249−260.

3. Jennions, I. K., and Turner, M. G., 1993, "3−D Navier−Stokes Computations c Transonic Fan Flow Using an Explicit Flow Solver and an Implicit k−ε Solver," ASM *Journal of Turbomachinery*, Vol. 115, pp. 261−272.

4. Dailey, L. D., Jennions, I. K., and Orkwis, P. D., 1994, "Simulating Laminar−Turbu lent Transition With a Low Reynolds Number k−ε Turbulence Model in a Navier−Stoke Flow Solver," AIAA Paper 94−0189.

5. Leylek, J. H., and Zerkle, R. D., 1994, "Discrete−Jet Film Cooling: A Compariso of Computational Results With Experiments," ASME *Journal of Turbomachinery*, Vo 116, pp. 358−368.

6. Prakash, C., and Zerkle, R. D., 1993, "Prediction of Turbulent Flow and Heat Tran fer in a Ribbed Rectangular Duct With and Without Rotation," ASME Paper 93−GT−20

THE APPLICABILITY OF TURBULENCE MODELS TO AERODYNAMIC AND PROPULSION FLOWFIELDS AT McDONNELL DOUGLAS AEROSPACE

N95-27886

Linda D. Kral, John A. Ladd, and Mori Mani
McDonnell Douglas Aerospace
St. Louis, Missouri

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–Launder, 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:

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

Vorticity used in production:

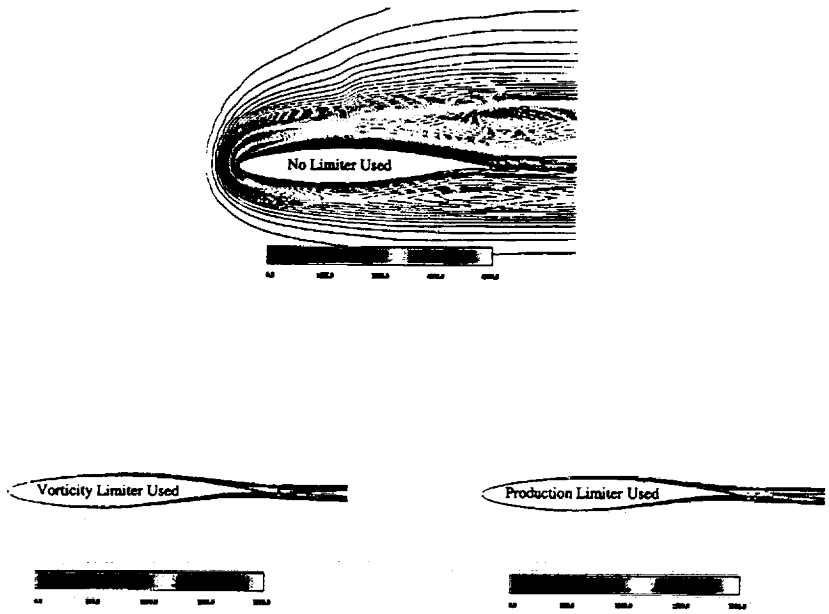
$$P_k^* = \frac{\bar{\mu}_t}{Re}|\omega|^2$$

Production limiter used:

$$P_k^* = \min(P_k, 20D_k) = \min(P_k, 20\, c_2\, \rho\, k\, Re)$$

Effect of Production Limiter for the Chien k–ε Model

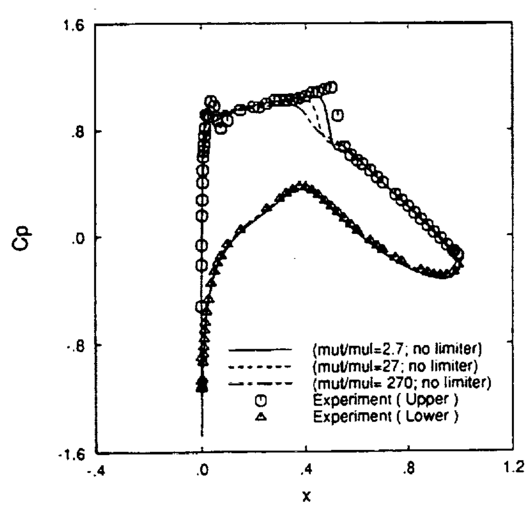
RAE Airfoil Analysis, Turbulent Viscosity Contours
Mach = 0.725, α = 2.55 deg., Re = 6.5 Million



RAE Airfoil Analysis

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

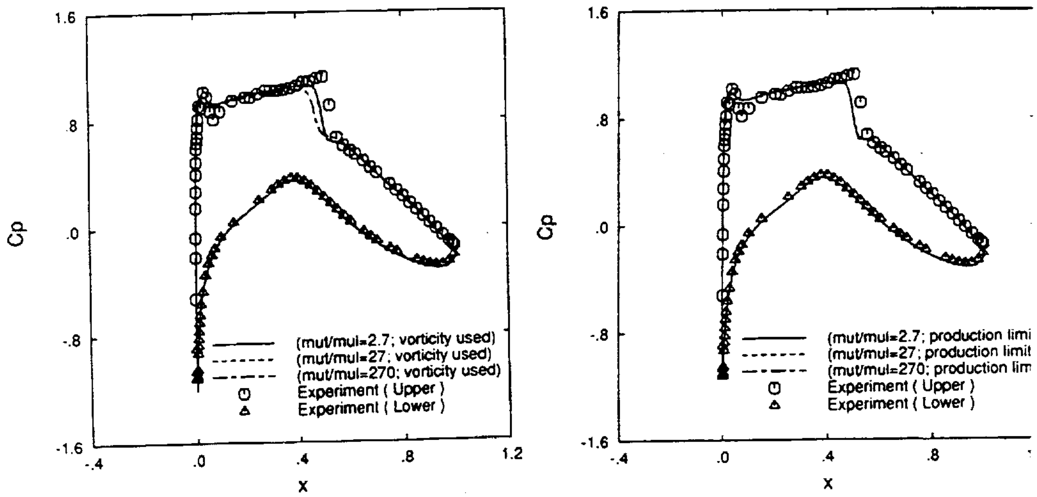
Effect of Freestream Turbulence Level on Surface Pressure
Chien $k - \epsilon$ Turbulence Model



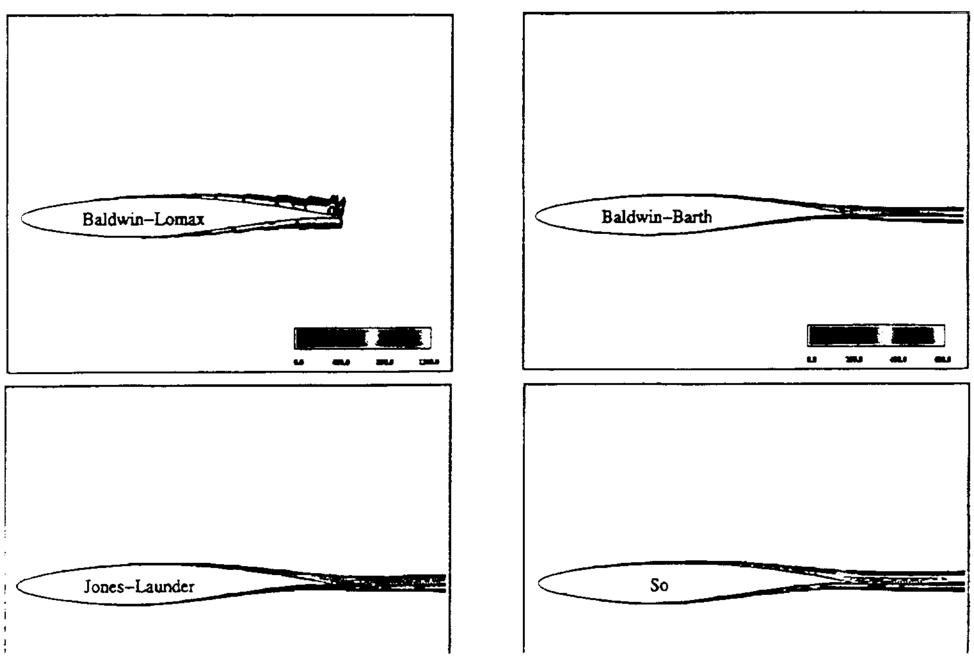
RAE Airfoil Analysis
$M_\infty = 0.725,\ \alpha = 2.55°,\ Re = 6.5$ Million

Production Limiter Used

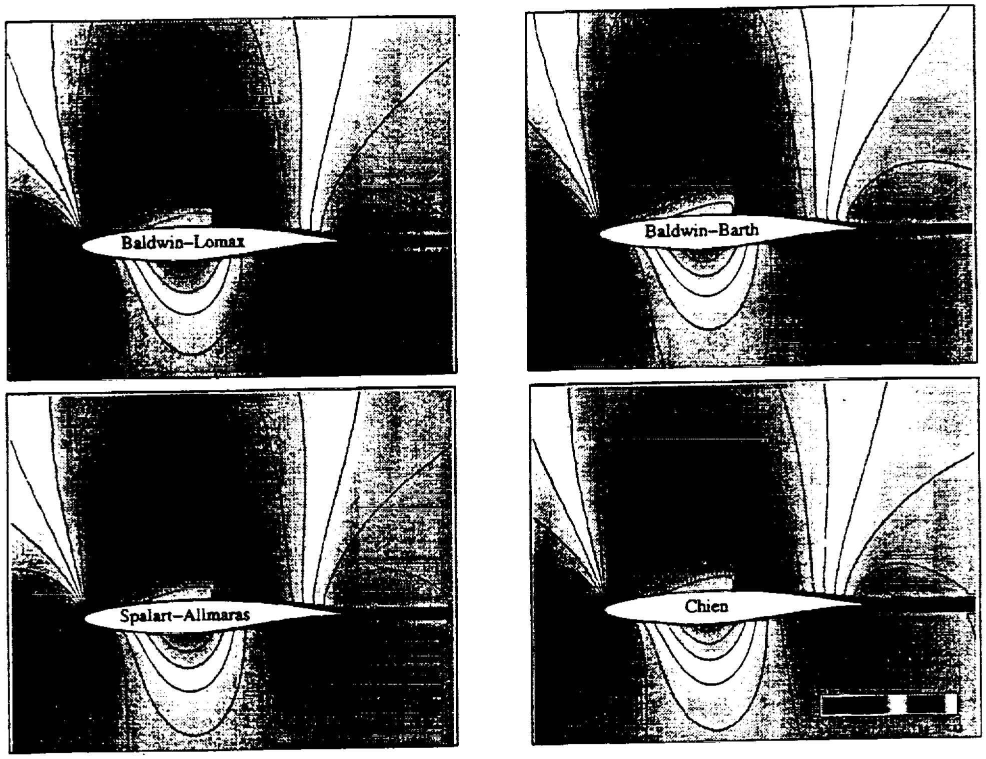
Chien $k-\epsilon$ Turbulence Model



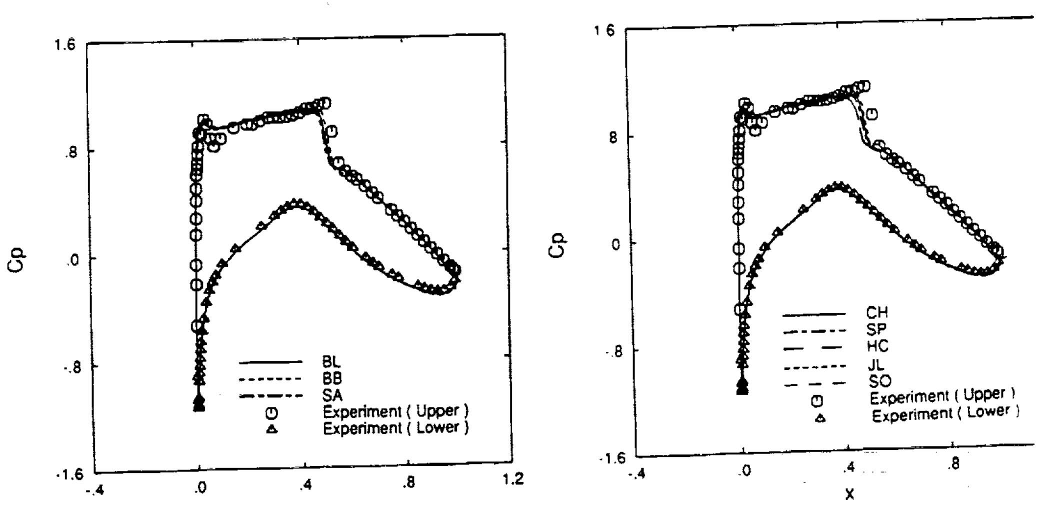
RAE Airfoil Analysis, Turbulent Viscosity Contours
Mach = 0.725, α =2.55 deg., Re = 6.5 Million



RAE Airfoil Analysis, Mach Contours
Mach = 0.725, α =2.55 deg., Re = 6.5 Million

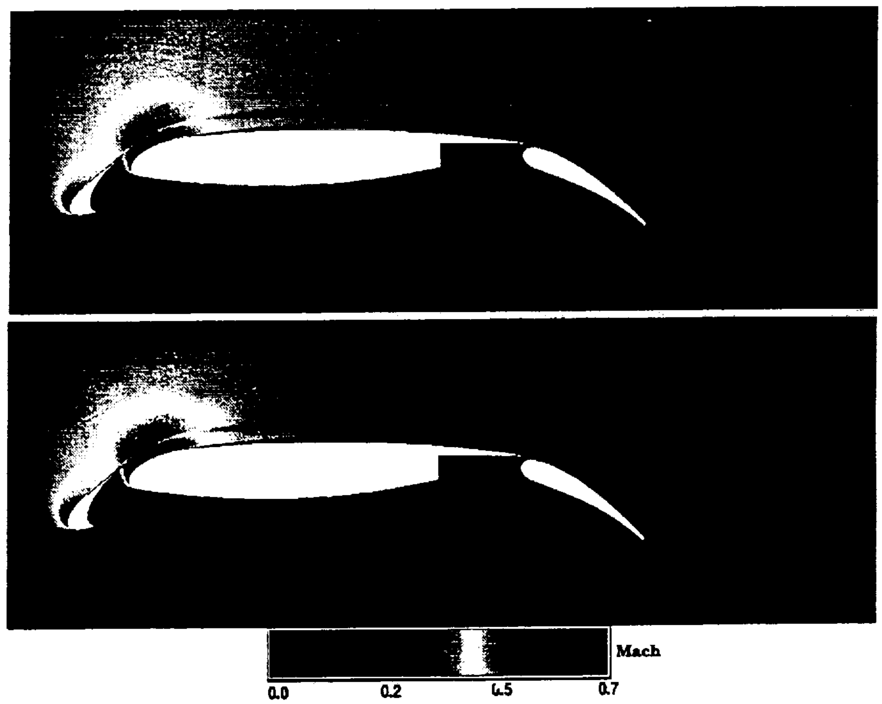


RAE Airfoil Analysis
$M_\infty = 0.725$, $\alpha = 2.55°$, $Re = 6.5$ Million
Effect of Turbulence Model on Surface Pressure



NASTD Solutionof MDA Three-Element High-Lift System

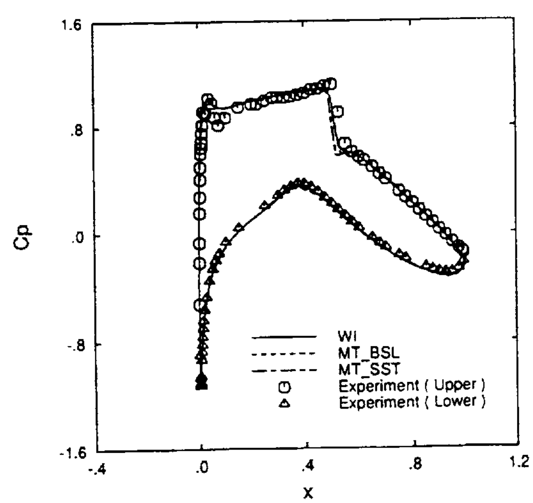
M = 0.2, AOA = 16.21



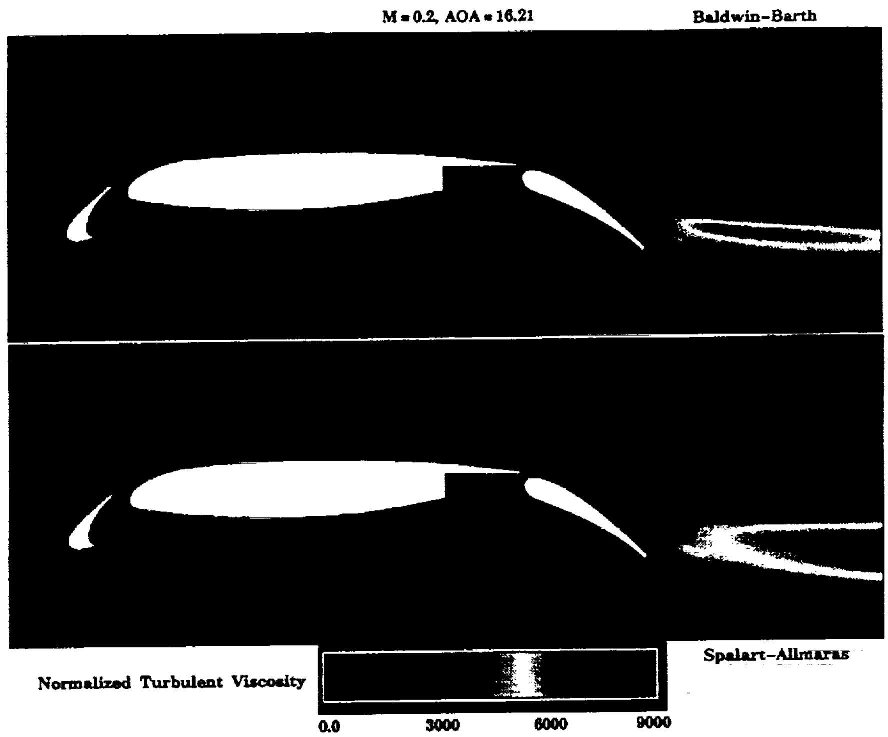
RAE Airfoil Analysis

$M_{\infty} = 0.725$, $\alpha = 2.55°$, $Re = 6.5$ Million

Effect of Turbulence Model on Surface Pressure

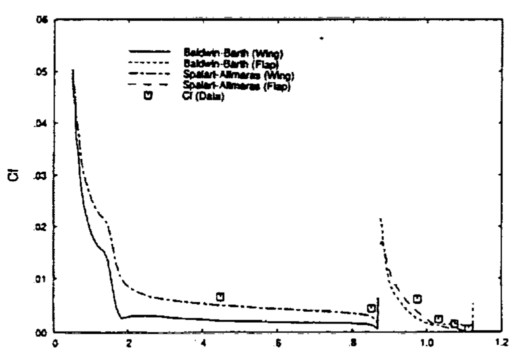


NASTD Solution of MDA Three-Element High-Lift System

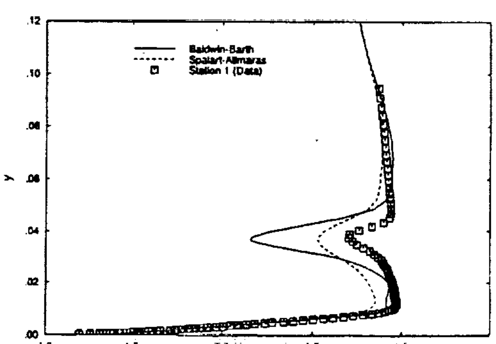


More Accurate Solutions Have Been Obtained With One-Equation Spalart-Allmaras Turbulence Model

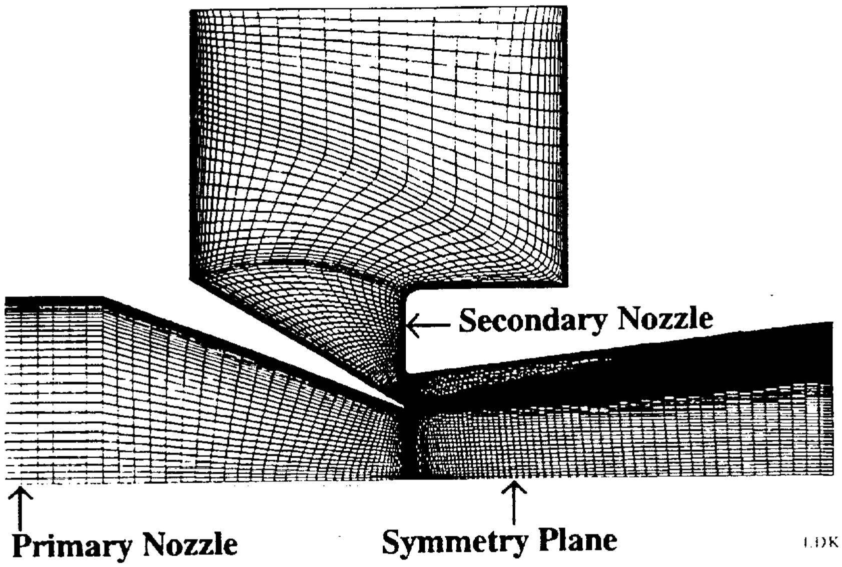
Skin Friction Coefficients on the upper Surfaces



Velocity Profile at Station 1 on the Wing (M=0.2, α=16.21)

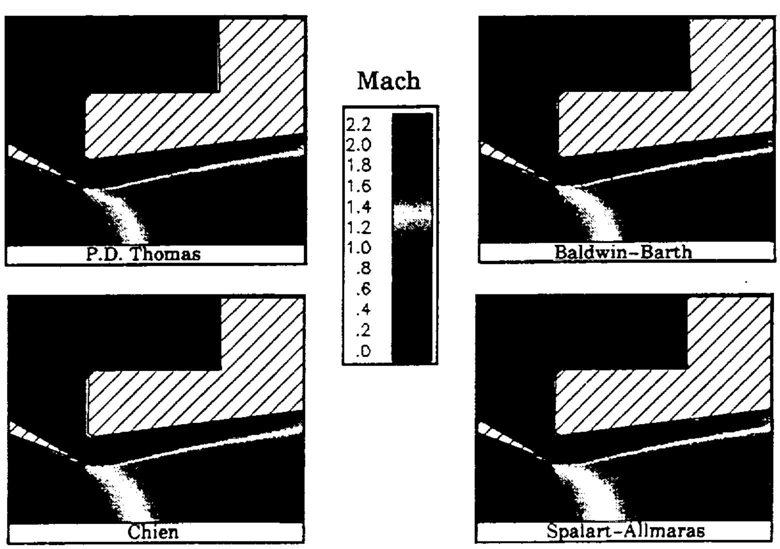


Four–Zone Grid for an Ejector Nozzle



Single Slot Ejector Analysis
NPR=14., Pts/Ptp=.34

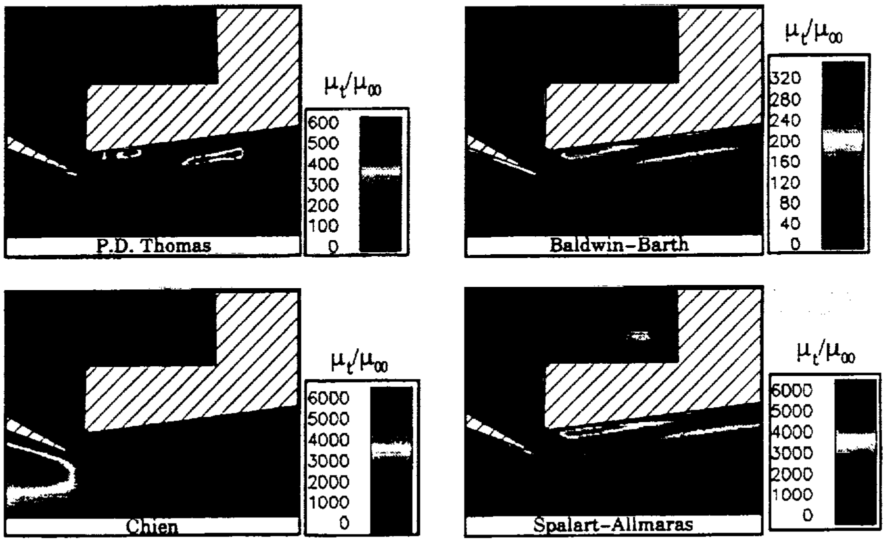
Mach Number Contours from Several Turbulence Models



Single Slot Ejector Analysis

NPR=14., Pts/Ptp=.34

Eddy Viscosity from Several Turbulence Models



Single-Slot Ejector Nozzle Analysis

$NPR = 14,\ P_{t_s}/P_{t_p} = 0.34$

$\mu_t/\mu_l \simeq 100$

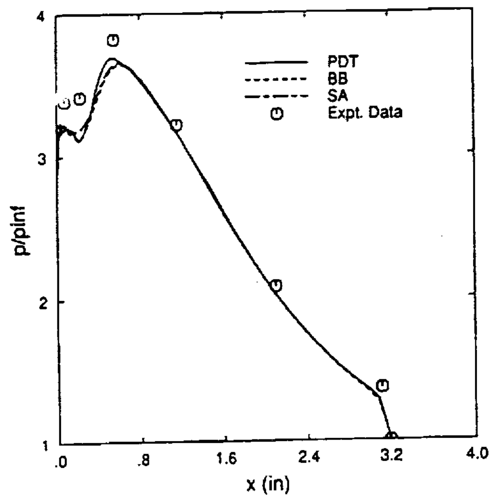
Comparison of Predicted Ejector Flow Rates

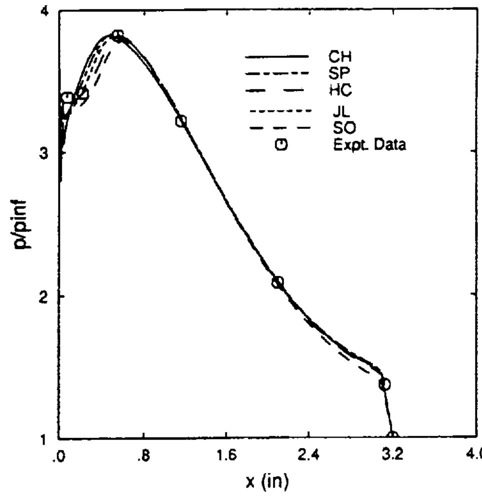
Model	W_s/W_p	% Error
Experiment	0.1010	–
Thomas/Baldwin-Lomax	0.1108	+9.7
Baldwin-Barth	0.1129	+11.8
Spalart-Allmaras	0.1146	+13.5
Chien $k-\epsilon$	0.1168	+15.6
Jones-Launder $k-\epsilon$	0.1126	+11.5
Speziale $k-\epsilon$	0.1127	+11.6
So $k-\epsilon$	0.1148	+13.7
Huang-Coakley $k-\epsilon$	0.1112	+10.1

Single Slot Ejector Nozzle

Surface Static Pressure Comparison with Experimental Data

$NPR = 14.0,\ P_{t_s}/P_{t_p} = 0.34$

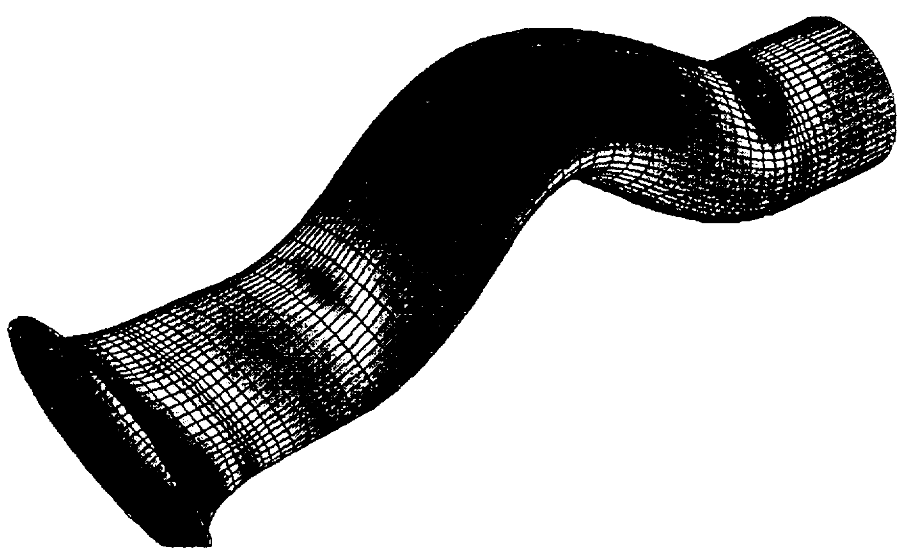




Offset Diffuser Analysis

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

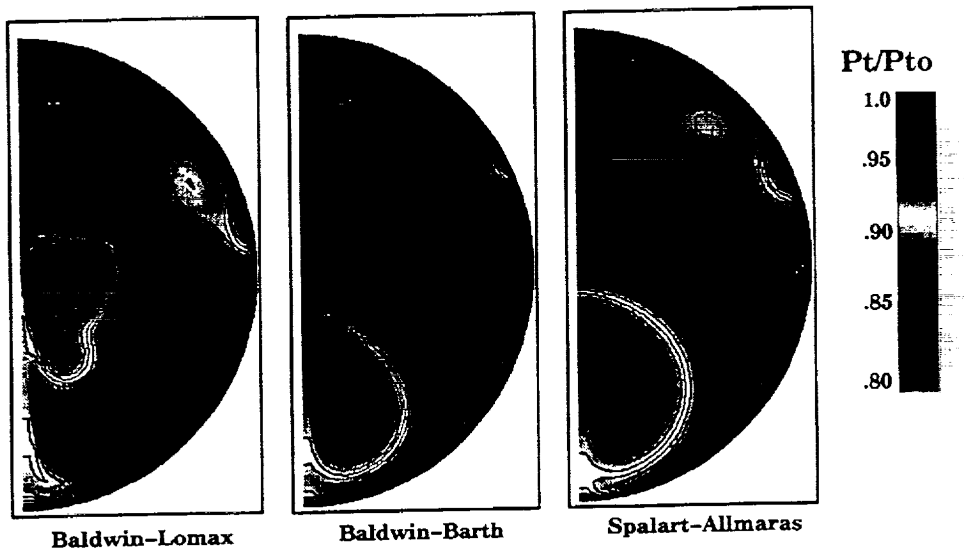
Surface Pressure and Computational Mesh



Offset Diffuser Analysis

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

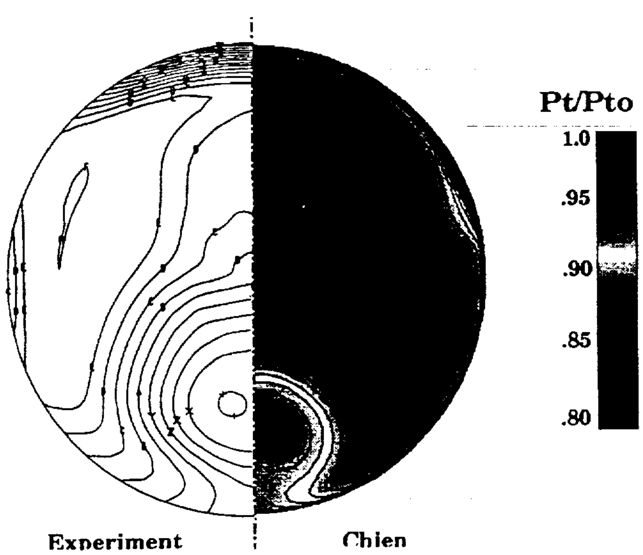
Comparison of Engine Face Total Pressures



Offset Diffuser Analysis

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

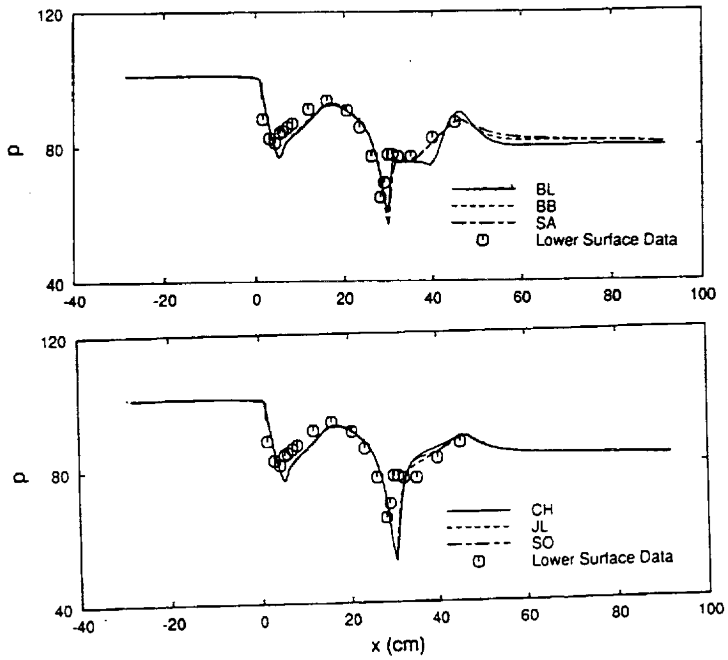
Comparison of Engine Face Total Pressures



Offset Diffuser Analysis

Lower Centerline Surface Static Pressure

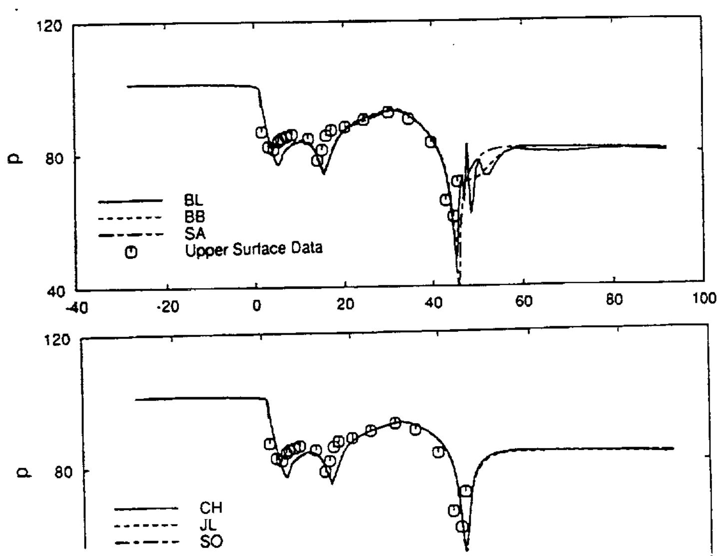
$A_e/A_t = 1.6$, $L/D = 4.5$, Design Pressure Ratio



Offset Diffuser Analysis

Upper Centerline Surface Static Pressure

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



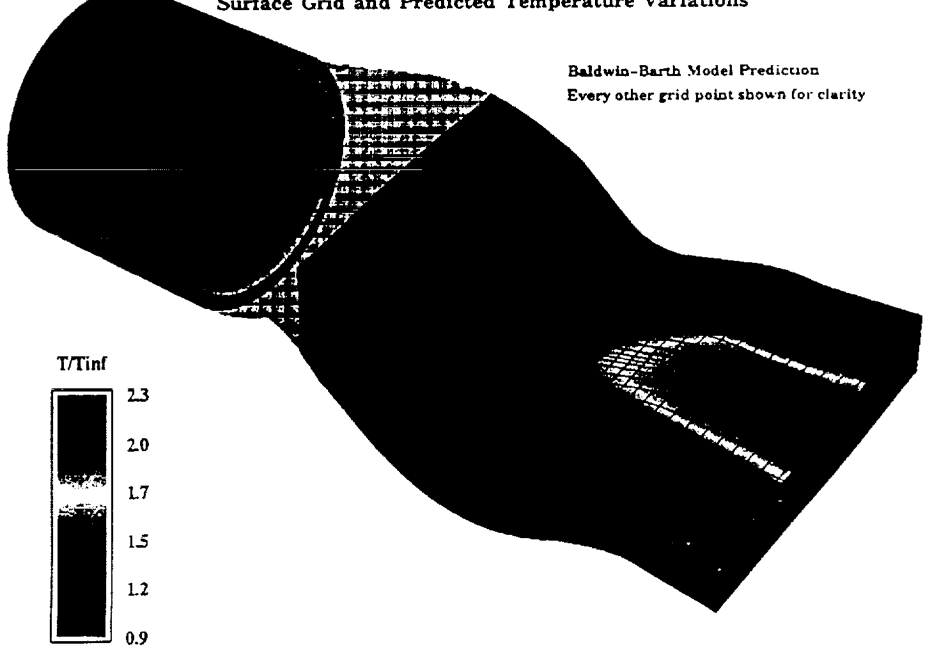
Three-Dimensional Highly Offset Diffuser

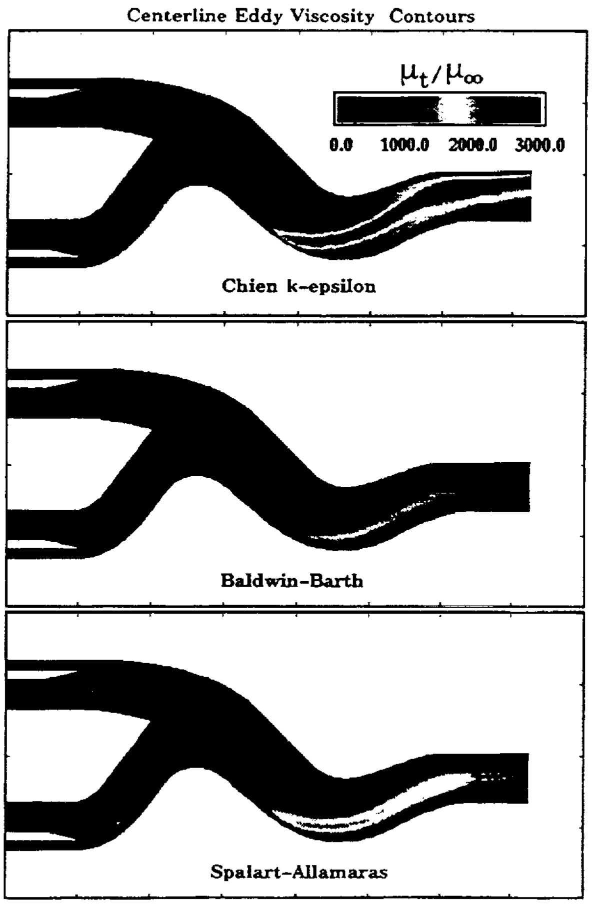
$A_e/A_t = 1.6$, $L/D = 4.5$, Design Pressure Ratio

Comparison of Engine Face Parameters

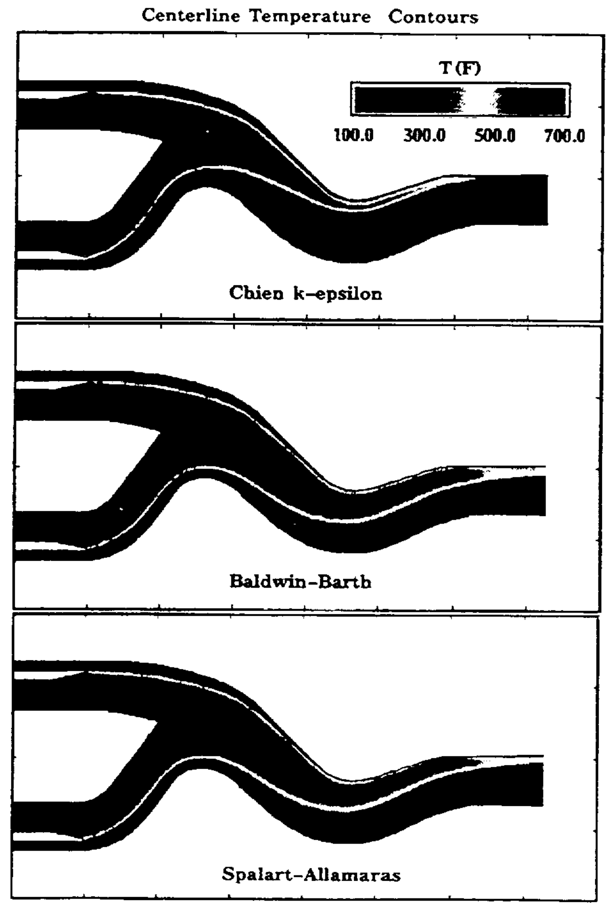
Model	$P_{t_{avg}}/P_{t_\infty}$	$P_{t_{min}}/P_{t_\infty}$	$\frac{P_{t_{max}} - P_{t_{min}}}{P_{t_{avg}}}$
Experiment	0.958	0.890	0.114
Baldwin-Lomax	0.936	0.708	0.292
Baldwin-Barth	0.944	0.735	0.265
Spalart-Allmaras	0.955	0.860	0.140
Chien $k-\epsilon$	0.970	0.894	0.106
Jones-Launder $k-\epsilon$	0.966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

G.E. Confluent Mixer
Surface Grid and Predicted Temperature Variations



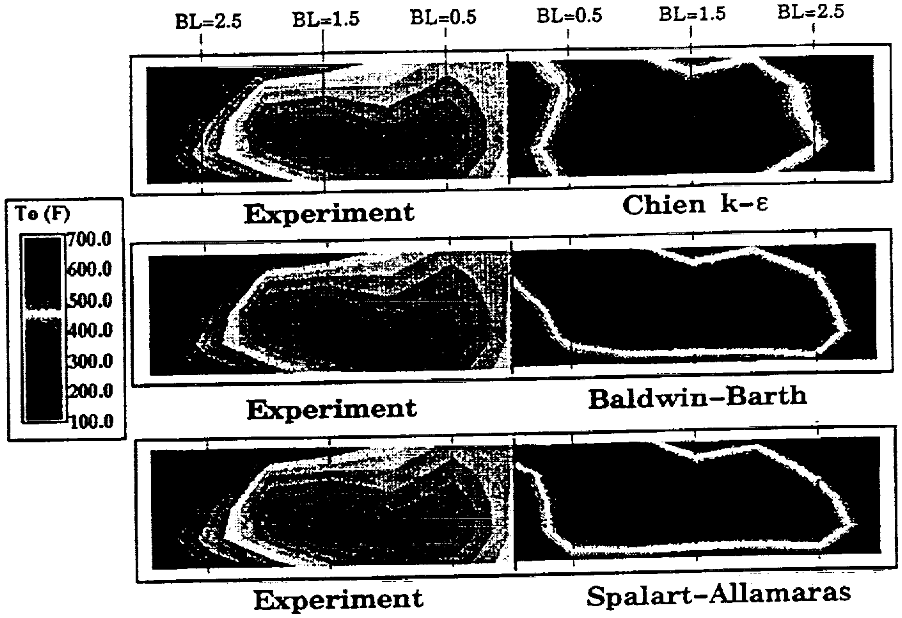


Centerline Eddy Viscosity Contours
μ_t/μ_∞
0.0 1000.0 2000.0 3000.0
Chien k-epsilon
Baldwin-Barth
Spalart-Allamaras



Centerline Temperature Contours
T (F)
100.0 300.0 500.0 700.0
Chien k-epsilon
Baldwin-Barth
Spalart-Allamaras

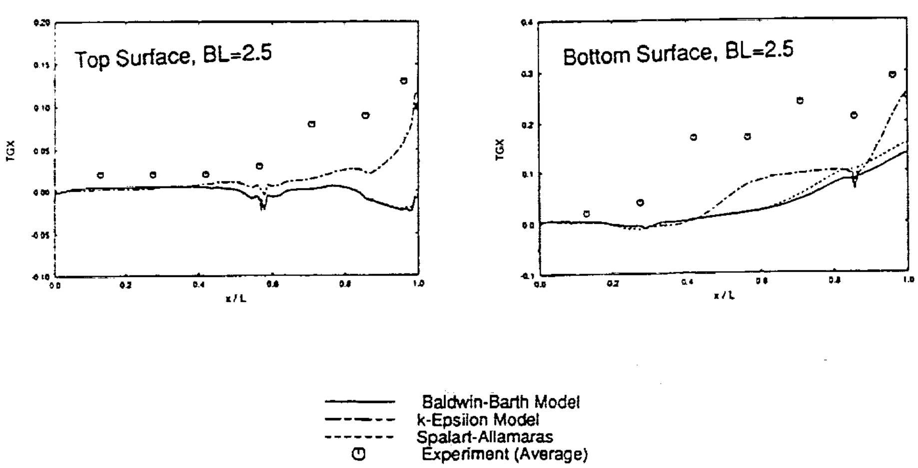
Comparison of Throat Total Temperatures



GE Slot Cooled Nozzle, Confluent Mixer

Surface Temperature Distributions,

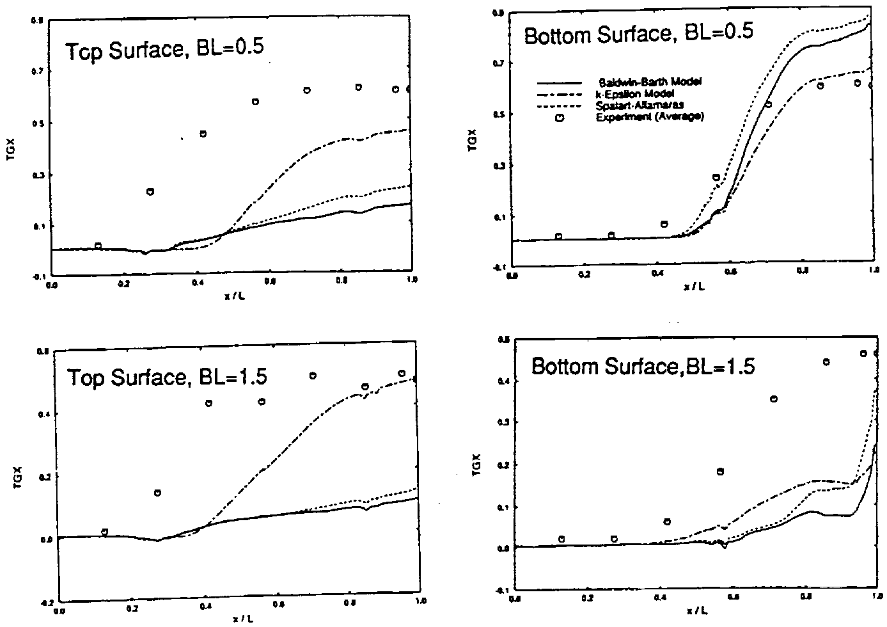
$$TGX = (T_I - T_{I_{cold}})/(T_{I_{hot}} - T_{I_{cold}})$$



GE Slot Cooled Nozzle, Confluent Mixer

Surface Temperature Distributions,

$$TGX = (T_I - T_{I_{cold}})/(T_{I_{hot}} - T_{I_{cold}})$$



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.

EXPERIENCE WITH k-ε TURBULENCE MODELS FOR HEAT TRANSFER COMPUTATIONS IN ROTATING

N95- 27887

Prabhat Tekriwal
GE Corporate Research and Development
Schenectady, New York

OUTLINE

- Geometry and flow configuration
- Effect of y+ on heat transfer computations
- Standard and Extended k-ε 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

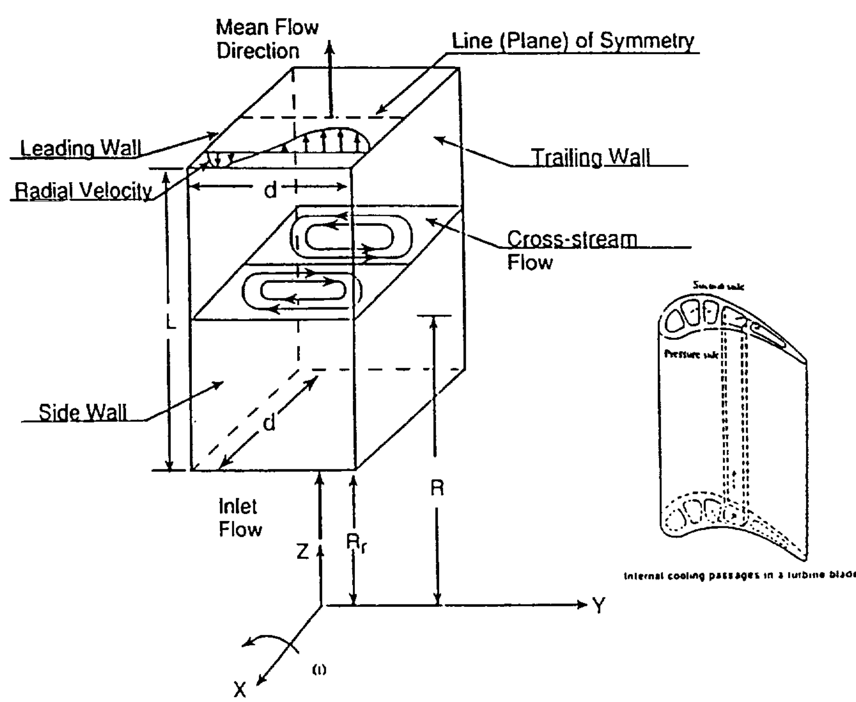


Fig. 1-Illustration of geometry and physics of flow

PRECEDING PAGE BLANK NOT FILMED

$$\mu_t = f_\mu C_\mu \rho k^2/\varepsilon$$

$$\frac{D(\rho k)}{Dt} = \frac{\partial}{\partial x_i}\left(\frac{\mu_t}{Pr_k}\frac{\partial k}{\partial x_i}\right) + \rho(G_k - \varepsilon)$$

$$\frac{D(\rho \varepsilon)}{Dt} = \frac{\partial}{\partial x_i}\left(\frac{\mu_t}{Pr_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right) + f_1 C_1 \frac{\varepsilon}{k}\rho G_k - f_2 C_2 \rho \frac{\varepsilon^2}{k} + C_3 \rho \frac{G_k^2}{k}$$

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

Standard k-ε model:
$Pr_k = 1.0$, $Pr_\varepsilon \cong 1.3$, $C_1=1.44$, $C_2=1.92$, $C_3=0.0$, $f_1=1.0$, $f_2=1.0$, and $f_\mu=1.0$

Extended k-ε model:
$Pr_k = 0.89$, $Pr_\varepsilon=1.15$, $C_1=1.15$, $C_2=1.9$, $C_3=0.25$, $f_1=1.0$, $f_2=1.0$, and $f_\mu=1.0$

Lam-Bremhorst low-Re model:
$Pr_k = 1.0$, $Pr_\varepsilon \cong 1.3$, $C_1=1.44$, $C_2=1.92$, $C_3=0.0$, $f_1=(1+0.05/f_\mu)^3$, $f_2=1-e^{-R_t^2}$, and $f_\mu=(1-e^{-0.0165R_k})^2(1+20.5/R_t)$, where $R_k=k^{1/2} y \rho/\mu$ and $R_t=k^2 \rho/\mu\varepsilon$

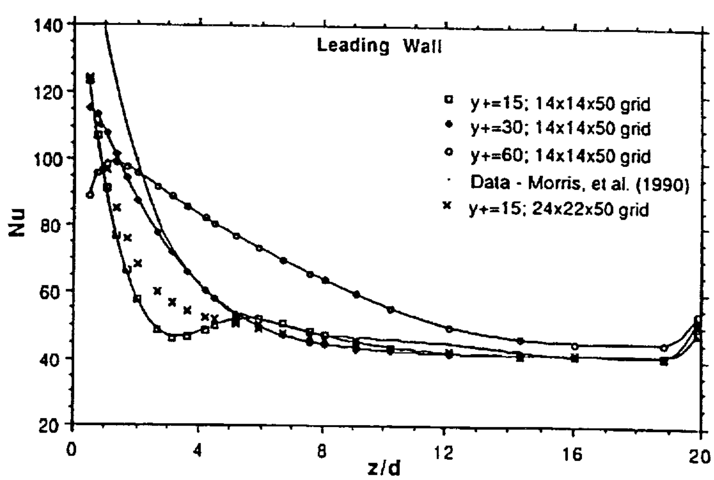


Fig. 2(a)-Effect of y+ and grid size on Nu computation

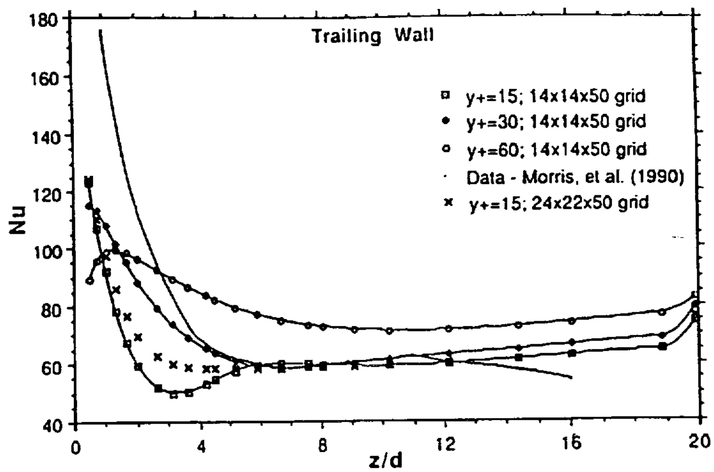


Fig. 2(b)-Effect of y+ and grid size on Nu computation

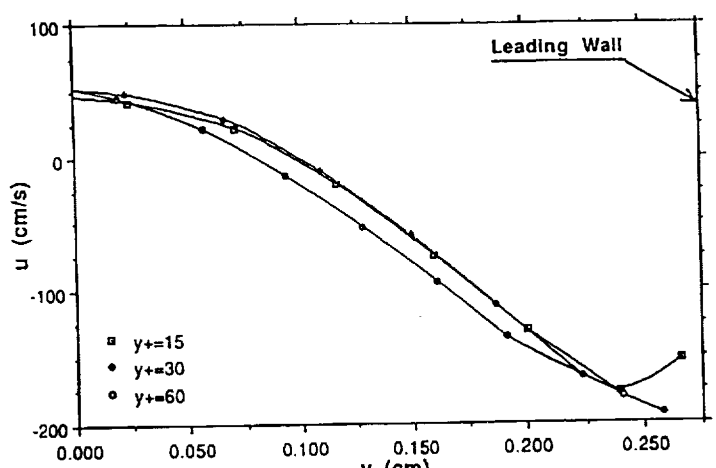


Fig. 2(c)-Effect of y+ on the cross-stream velocity near leading wall

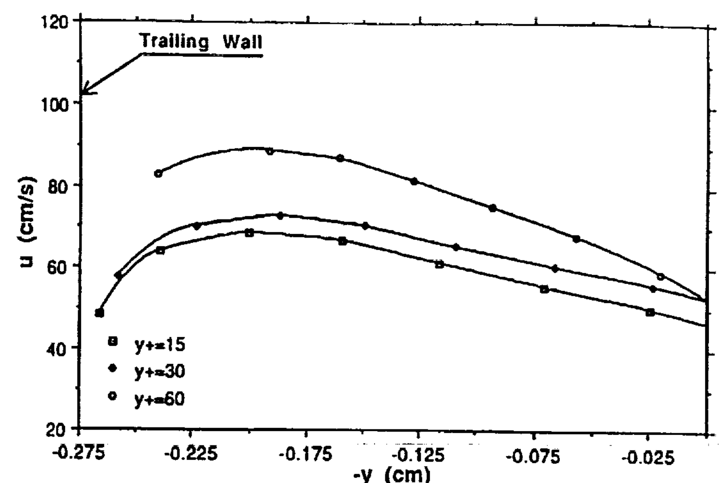


Fig. 2(d)-Effect of y+ on the cross-stream velocity near trailing wall

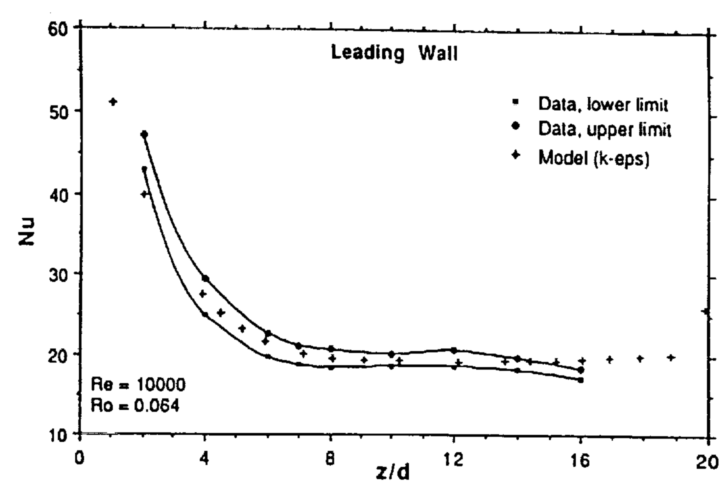


Fig. 3(a)-Comparison of model results with data (Morris & Ghavami-Nasr,1991)

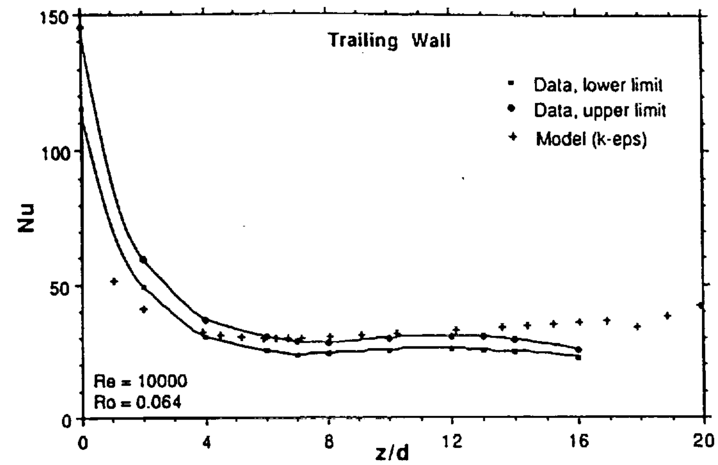


Fig. 3(b)-Comparison of model results with data (Morris & Ghavami-Nasr, 1991)

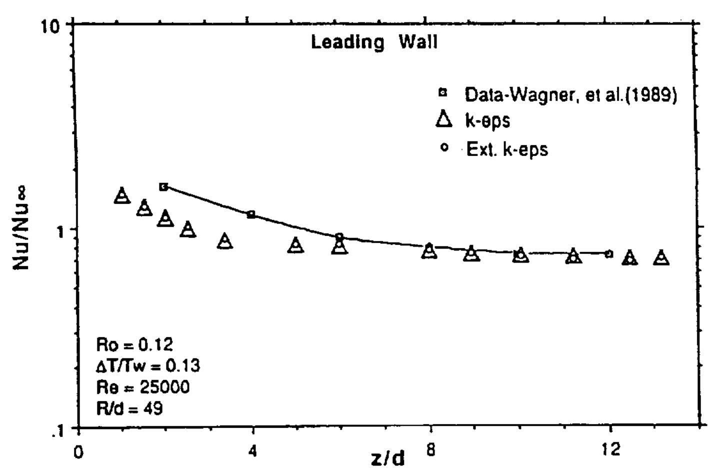


Fig. 4(a)-Comparison of model results with data on leading wall

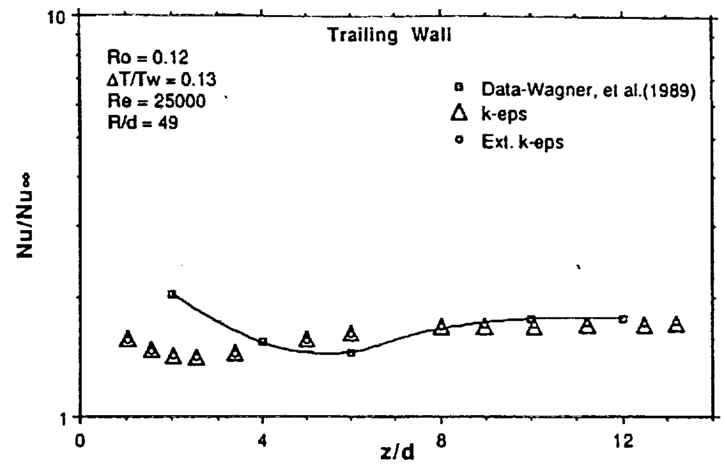


Fig. 4(b)-Comparison of model results with data on trailing wall

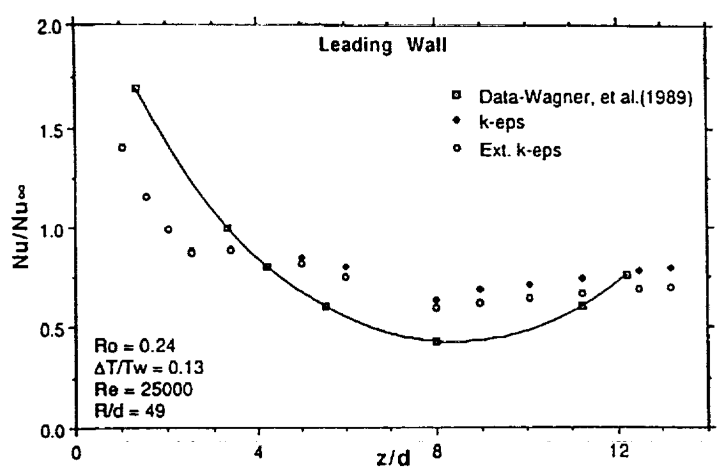


Fig. 4(c)-Comparison of model results with data on leading wall

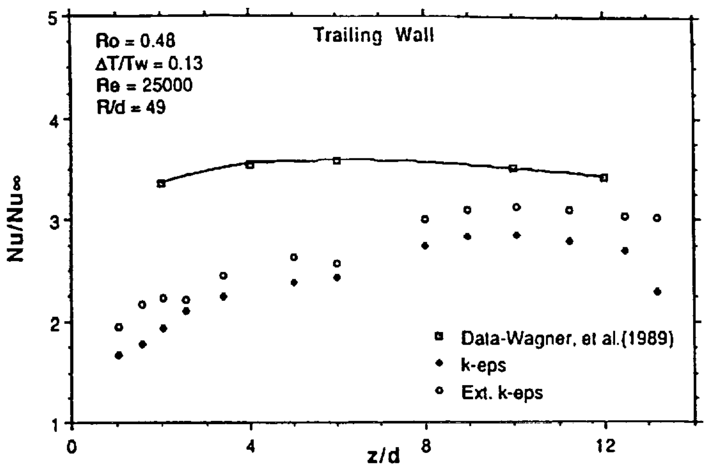


Fig. 4(d)-Comparison of model results with data on trailing wall

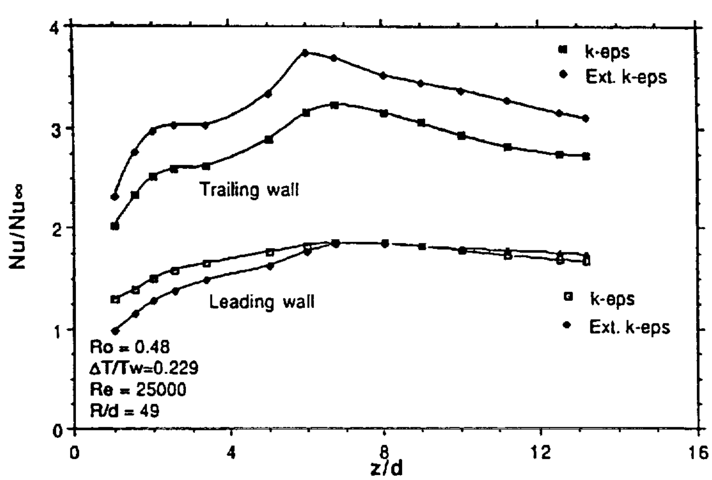


Fig. 5-Comparison of the two model results at high Ro and high density ratio

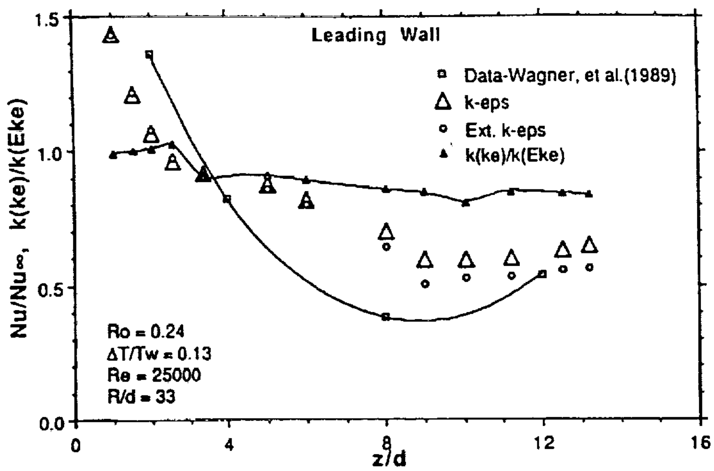


Fig. 6(a)-Plots of Nu-ratio and k-ratio from the two models

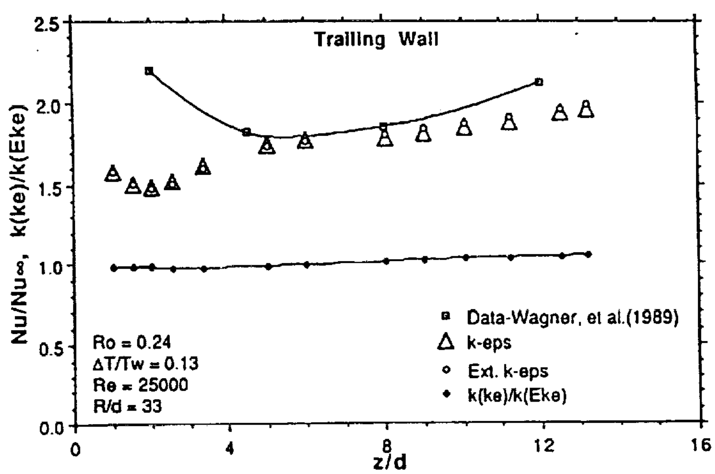


Fig. 6(b)-Plots of Nu-ratio and k-ratio from the two models

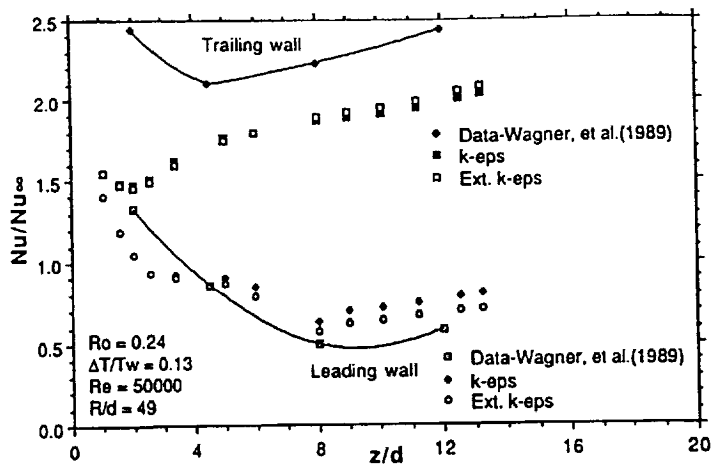


Fig. 7(a)-Comparison on both leading and trailing walls

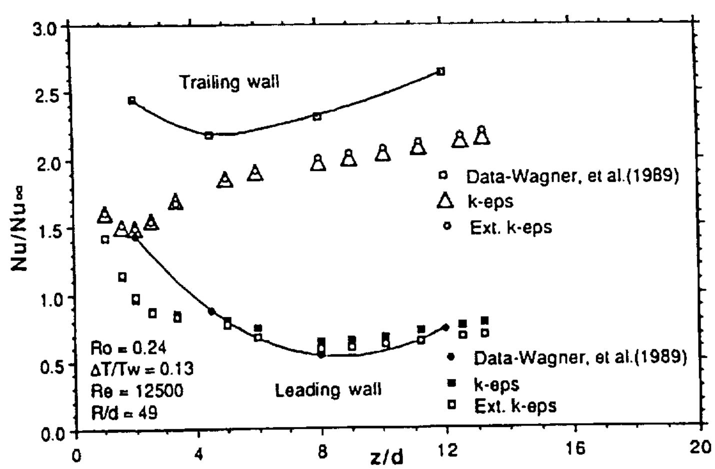


Fig. 7(b)-Comparison on both leading and trailing walls

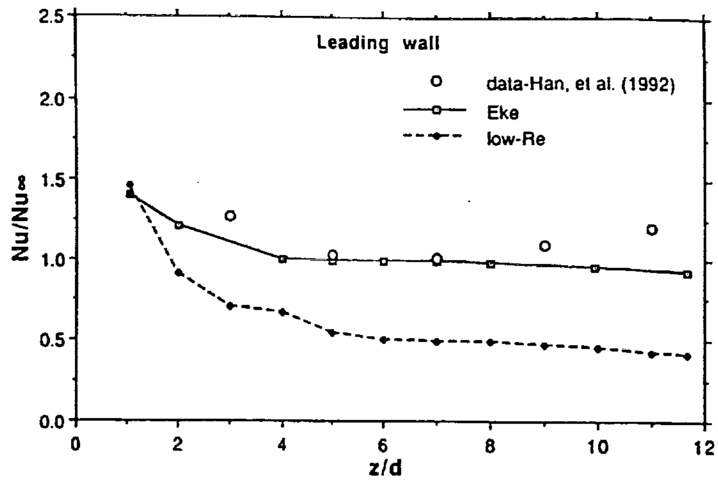


Fig. 8(a)-Comparison of Eke and low-Re results on leading wall
(Re=10000,Ro=.088,Case B)

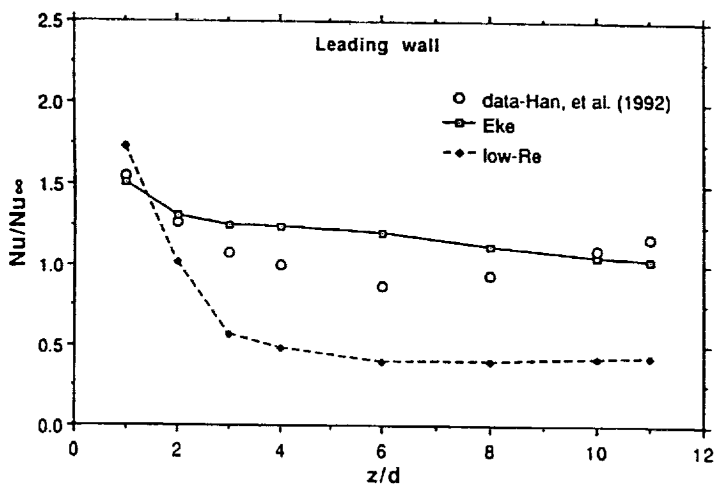


Fig. 8(b)-Comparison of Eke and low-Re results on leading wall
(Re=5000,Ro=.176,CaseB)

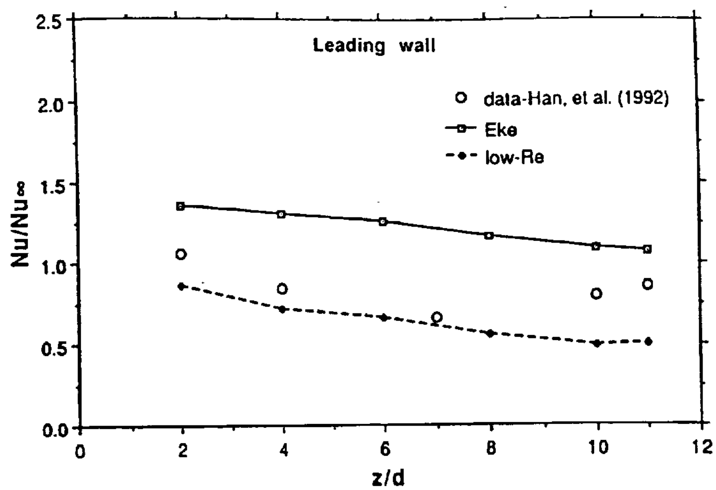


Fig. 8(c)-Comparison of Eke and low-Re results on leading wall (Re=5000,Ro=.176,CaseC)

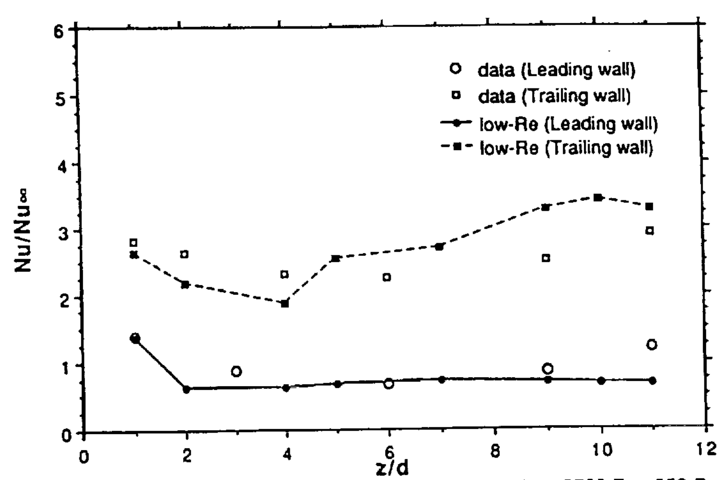


Fig. 8(d)-Comparison of low-Re results with data (Re=2500,Ro=.352,CaseB)

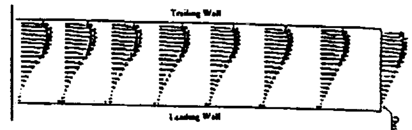


Fig. 9-Velocity vectors in the yz-plane

Table I Prediction of Flow Reversal Near the Leading Wall

Ro	ΔT/Tw	R/d	Re	Gr/Re^2	Flow Reversal ?
0.12	0.07	49	25000	0.05	No
	0.13			0.09	No
	0.23			0.16	No
	0.36			0.26	No
	0.48			0.34	Yes
	0.07	196		0.20	No
		300		0.30	Yes
0.24	0.07	33	25000	0.13	No
		49		0.20	No
		196		0.77	Yes
		300		1.18	Yes
	0.13	49		0.36	Yes
	0.16			0.45	Yes
	0.23			0.65	Yes
	0.07	.	12500	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 as 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

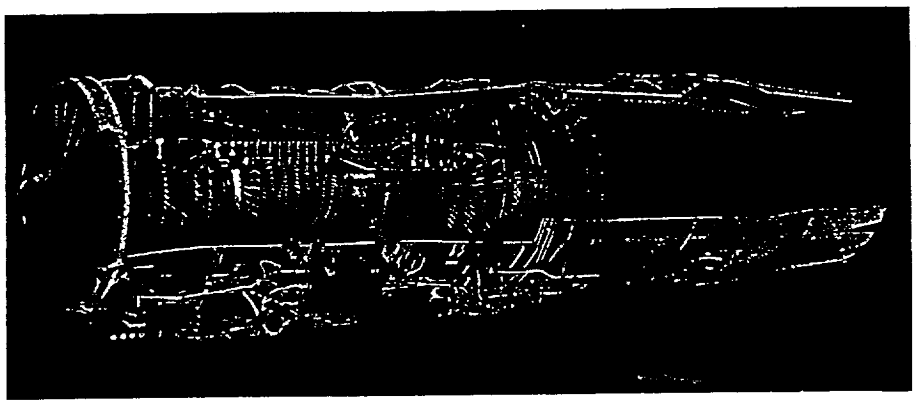
1. Han, J.-C., and Zhang, Y.M., 1992, "Effect of Uneven Wall Temperature on Local Heat Transfer in a Rotating Square Channel With Smooth Walls and Radial Outward Flow," *ASME J. Heat Transfer*, Vol. 114, pp. 850-858.
2. Morris, W.D., and Ghavami-Nasr, G., 1991, "Heat Transfer Measurements in Rectangular Channels with Orthogonal Mode Rotation," *ASME J. Turbomachinery*, Vol. 113, pp. 339-345.
3. Prakash, C., and Zerkle, R., 1991, "Prediction of Turbulent Flow and Heat Transfer in a Radially Rotating Square Duct," *ASME Heat Transfer in Gas Turbine Engines*, HTD-Vol. 188.
4. Tekriwal, P., 1992, "Heat Transfer Predictions with Extended k-ε Turbulence Model in Radial Cooling Ducts Rotating in Orthogonal Mode," ASME HTD - Vol. 226, *Fundamentals and Applied Heat Transfer Research for Gas Turbine Engines*, eds. D.E. Metzger and M.E. Crawford, pp. 41-50. Also in *ASME Journal of Heat Transfer*, Vol. 116, 1994, pp. 369-380.
5. Tekriwal, P., 1994, "Prediction of Heat Transfer for Turbulent Flow in Rotating Radial Duct," *Proc. of the 5th Intl. Symp. on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC 5)*, Vol. A, pp. 673-688.
6. Tekriwal, P., 1994, "Heat Transfer Predictions in Rotating Radial Smooth Channel: Comparative Study of k-ε Models with Wall Function and Low-Re Model," Paper No. 94-GT-196, Presented at *The ASME International Gas Turbine and Aeroengine Congress and Exposition*, The Hague, Netherlands, June 1994.
7. Tekriwal, P., 1994, "Centrifugal Buoyancy Driven Reverse Flow Near The Leading Wall of a Rotating Cooling Passage," To be presented at The ASME Winter Annual Meeting, Chicago, Illinois, November 1994.
8. Wagner, J.H., Johnson, B.V., and Hajek, T.J., 1989, "Heat Transfer in Rotating Passages with Smooth Walls and Radial Outward Flow," *ASME Gas Turbine and Aeroengine Congress and Exposition*, Paper 89-GT-272.

TURBULENCE MODELS FOR GAS TURBINE COMBUSTORS

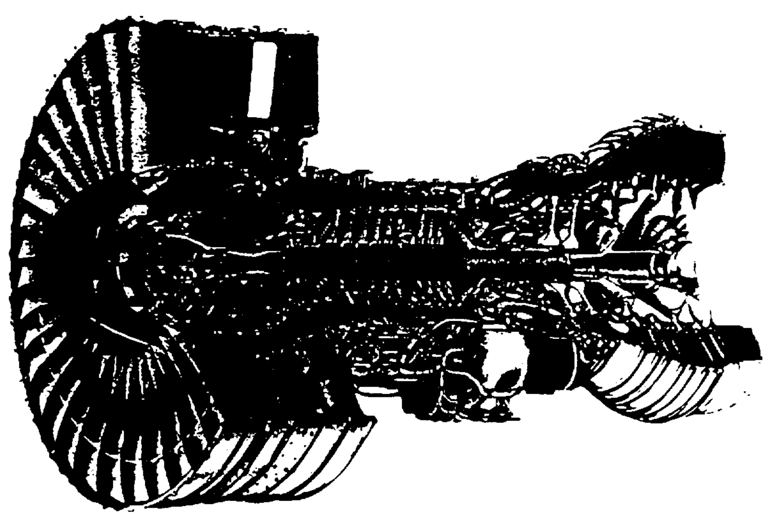
N95- 27888

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

F100-PW-200 TURBOFAN ENGINE



PW4000



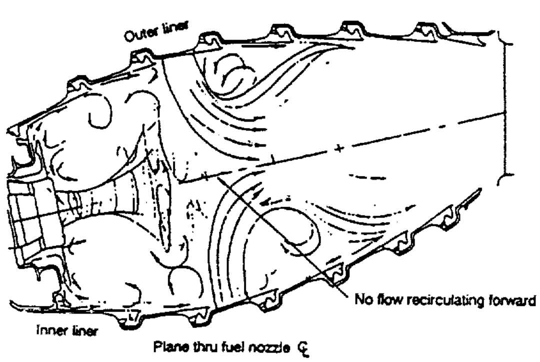
PRECEDING PAGE BLANK NOT FILMED

CONTENTS

- Gas Turbine Combustor Flow Physics
- Turbulence Model Investigations
- Turbulent Combustion Modeling
- Present Status and Future Needs

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.



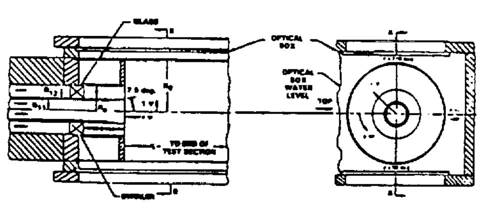
TURBULENCE MODELS SURVEYED

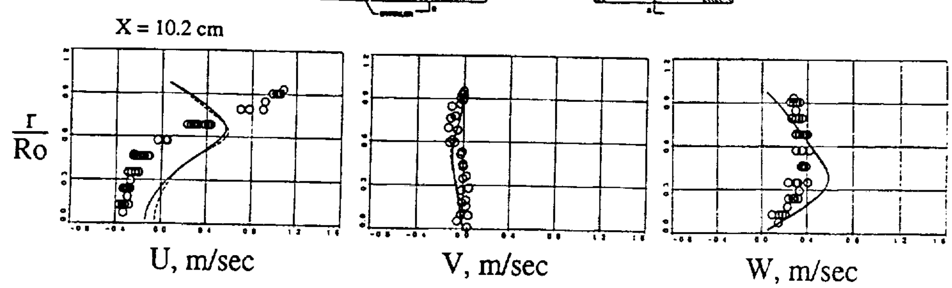
- Following models or modifications have been tested at P&W / UTRC using RANS solvers on building block flows:
  1. low-Re models (complex ducts);
  2. RSTM or SMC (complex ducts, swirling and non-swirling dump combustor);
  3. RNG (pipe, backstep, 180 deg duct);
  4. two-layer near-wall model (internal 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:





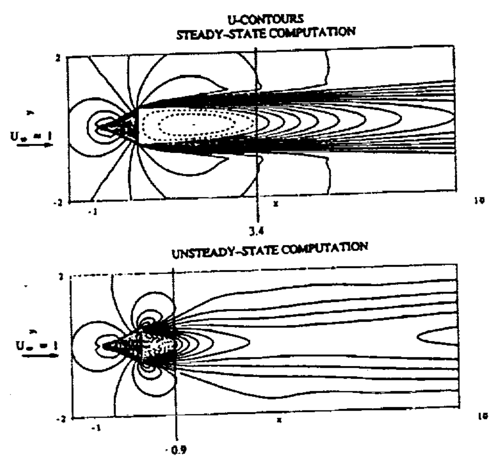
- Poor agreement of CFD and data highlights need for improved upstream BC specification (swirler geometry), 3-D, unsteady analysis. Even SMC models fail to reproduce downstream velocity profiles.

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):



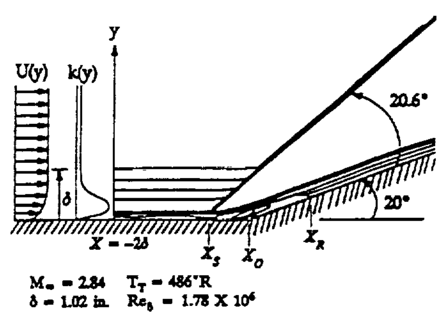
Ratio of characteristic frequences (estimated)

$$\frac{f_T}{f_o} \simeq 225$$

UNSTEADINESS AND FLOW FIELD RESOLUTION

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

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



Frequency ratio:

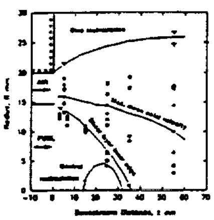
$$\frac{f_T}{f_o} \simeq 7.7$$

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

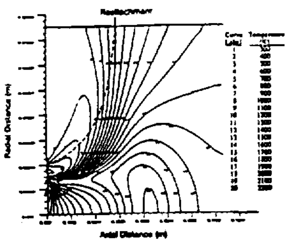
- Limitations of steady-state and unsteady-state RANS solvers set by flow characteristic time scales.
  True time-accurate solvers (LES, DNS) needed for prediction of all relevant phenomena

TURBULENT COMBUSTION MODELING

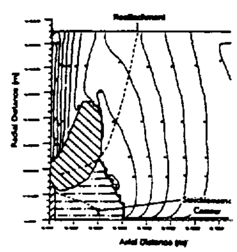
- Eddy Dissipation Concept Model, together with reaction exclusion regions, capable of prediction gross flow features at near LBO conditions (Sturgess et al., 94-GT-433)



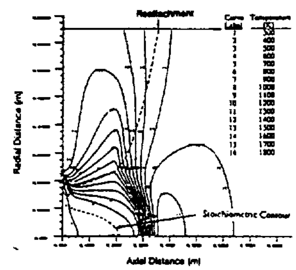
Near-field flow features



EBU-model Temperature field



Reaction exclusion regions



EDC-model Temperature field

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

TURBULENT COMBUSTION MODELING

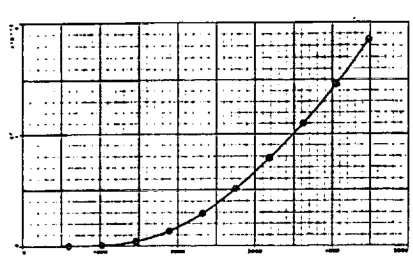
- Assumed-Pdf method of Girimaji (LaRC Workshop, 1991) used with non-equilibrium kinetics model.

$$\frac{K_{f\,turb}}{K_{f\,Lam}} = \frac{\int_a^b k_f(T)\ P(T)\ dT}{k_f\left(\tilde{T}\right)}$$

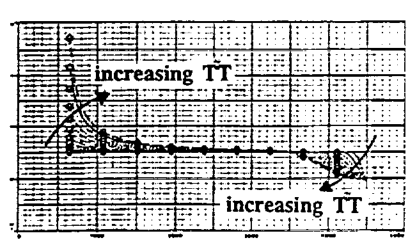
$$T_{min} = \max\left(\tilde{T} - \phi\ \sqrt{\widetilde{T''T''}}\ ,\ T_{low}\right)$$

$$T_{max} = \min\left(\tilde{T} + \phi\ \sqrt{\widetilde{T''T''}}\ ,\ T_{high}\right)$$

K_F



Temp, °R →

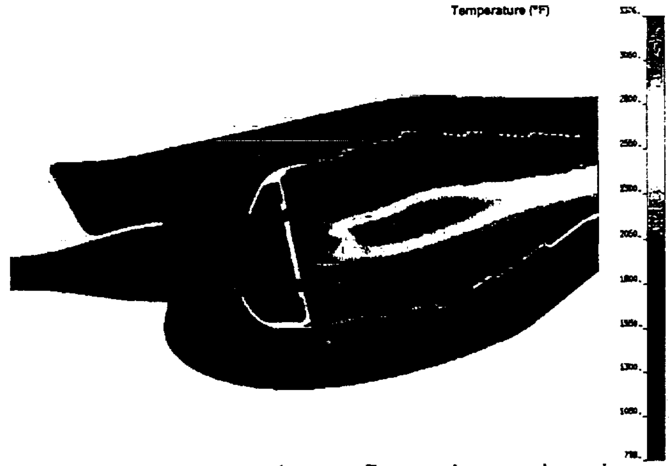


Temp, °R →

- **Example:** $N + O_2 \leftrightarrows NO + O$ in extended Zeldovich model
- Results dependent on T_{Low}, T_{High}, ϕ, modeling of $\tilde{hh}$ transport equation, etc.
- More testing needed

PRESENT STATUS OF COMBUSTOR MODELING

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

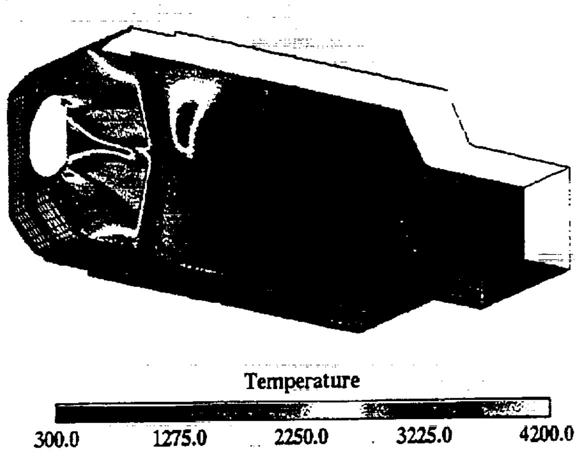


- **Example:** Time-dependent combustor flow using engineering boundary conditions, compressor exit to turbine inlet

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

PRESENT STATUS OF COMBUSTOR MODELING

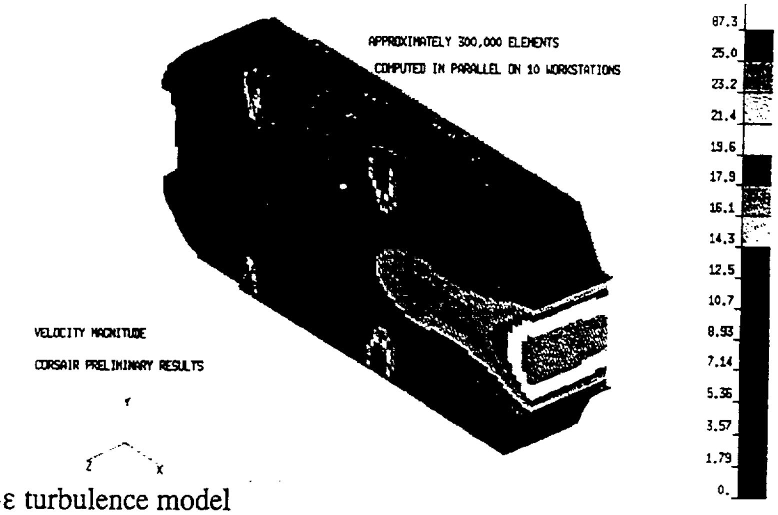
- **Example:** Structured flow solver solution of Task 200 LBO Research Combustor:



- k-ε 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:

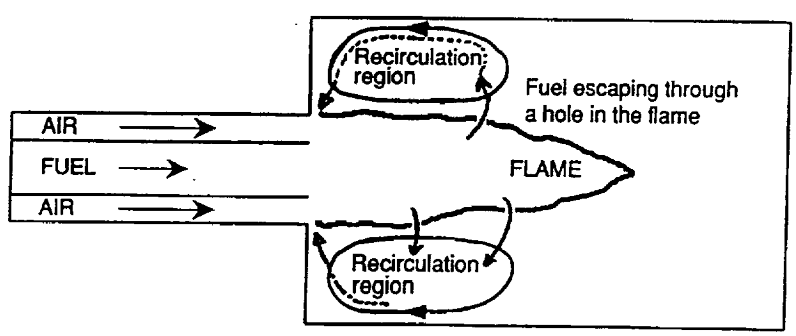


- k-ε turbulence model
- EBU combustion model for propane fuel
- Approx. 300,000 elements

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

COMBUSTION SYSTEM CFD MODELING AT GE AIRCRAFT ENGINES

D. Burrus and H. Mongia
GE Aircraft Engines
Cincinnati, Ohio

and

A. Tolpadi, S. Correa, and M. Braaten
GE Corporate Research and Development
Schenectady, New York

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

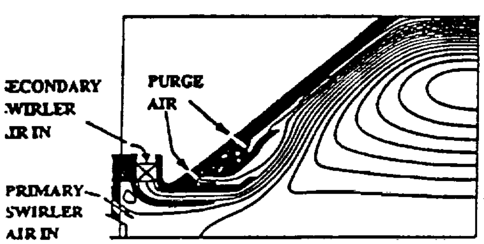
TYPICAL 3D MODEL OF A COMBUSTOR UTILIZING A MESH OF ~100,000 POINTS CAN BE GENERATED, RUN, AND POST PROCESSED WITHIN A SINGLE WORKING DAY!

HAS UNDERGONE CONTINUAL DEVELOPMENT TO IMPROVE AND ENHANCE MODELING CAPABILITIES

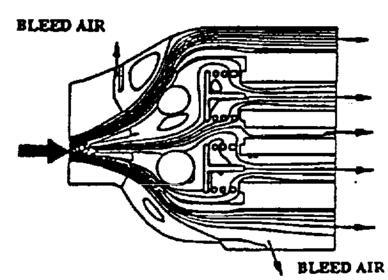
- CURRENTLY ON VERSION 3 RELEASE

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

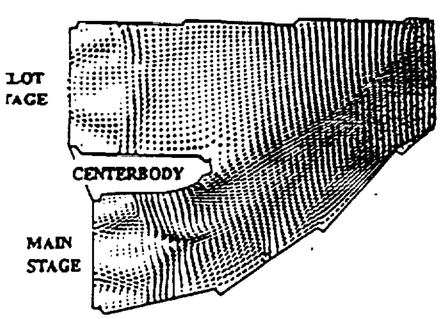
COMBUSTION SYSTEM CFD MODELING IN ACTION AT GEAE



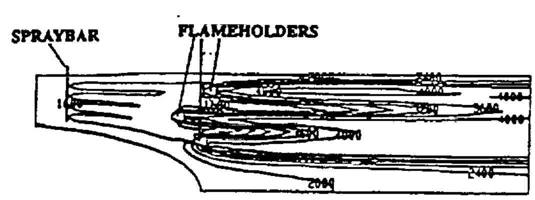
SWIRLCUP/SPRAY MODELING
- RECIRCULATION STRENGTH/SIZE
- FLOW FIELD CHARACTERISTICS
- SPRAY DROPLET TRAJECTORIES
- INPUTS FOR 3D COMBUSTOR MODEL



DIFFUSER FLOW MODELING
- Ps RECOVERIES AND Pt LOSSES
- FLOW FIELD CHARACTERISTICS



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



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

MODELING APPLIED FOR DESIGNING ENGINE COMBUSTION SYSTEMS

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

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

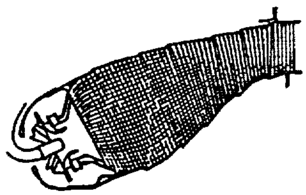
LM2500 CO EMISSIONS REDUCTION EFFORT

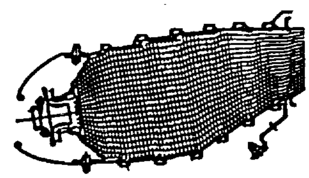
CF34 LINER COOLING MOD IMPACT ON CO EMISSIONS

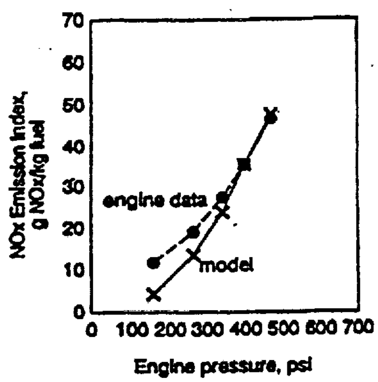
F110X AUGMENTOR MIXER, SPRAYBAR, FLAMEHOLDER INTERACTION OPTIMIZATION

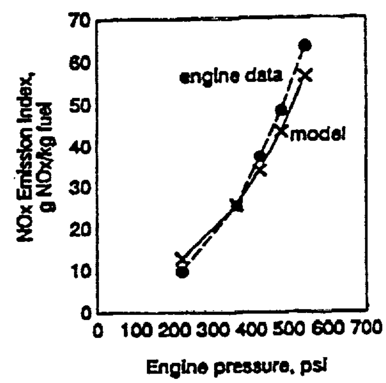
F110-400 AUGMENTOR EXHAUST DUCT LINER FAILURE AND FIX INVESTIGATION

CONCERT3D RESULTS FOR CURRENT PRODUCTION COMBUSTORS

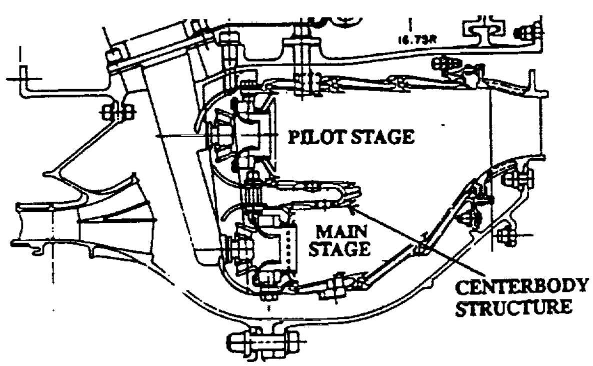


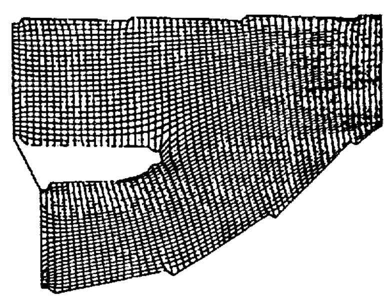




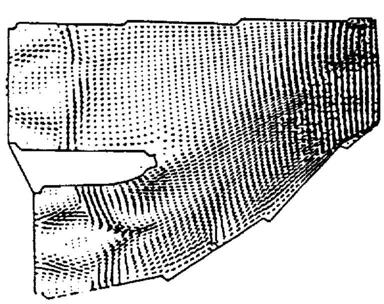


CONCERT3D MODEL OF NASA/GE E3 COMBUSTOR





57X57X25 GRID (81,225 TOTAL MESH POINTS)

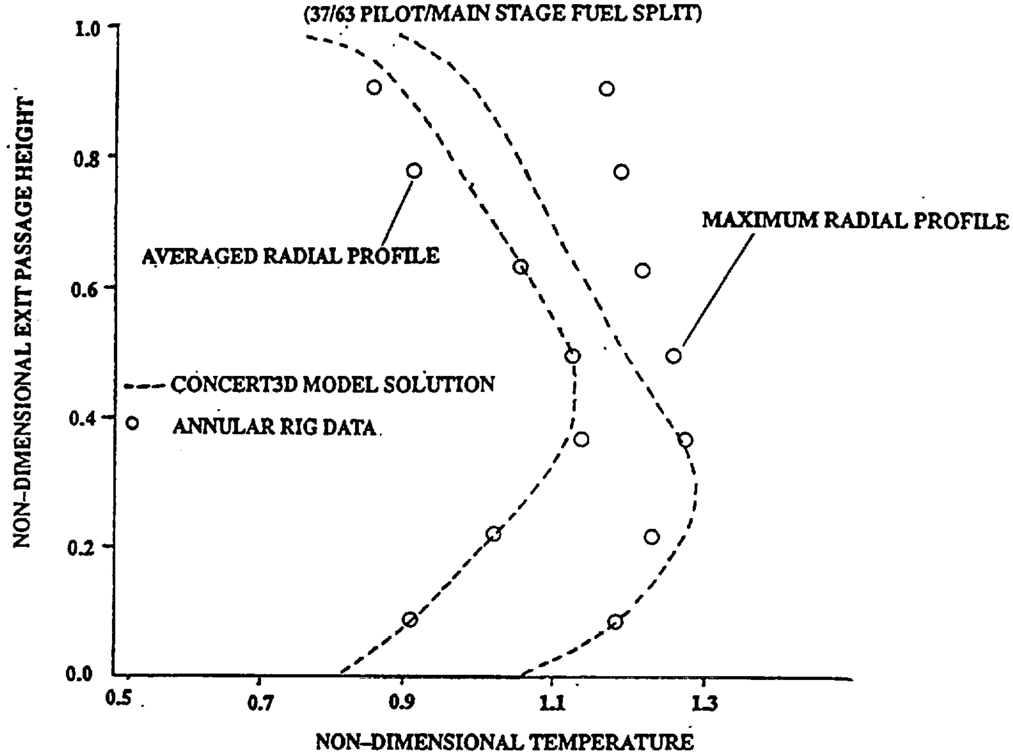


VELOCITY VECTORS

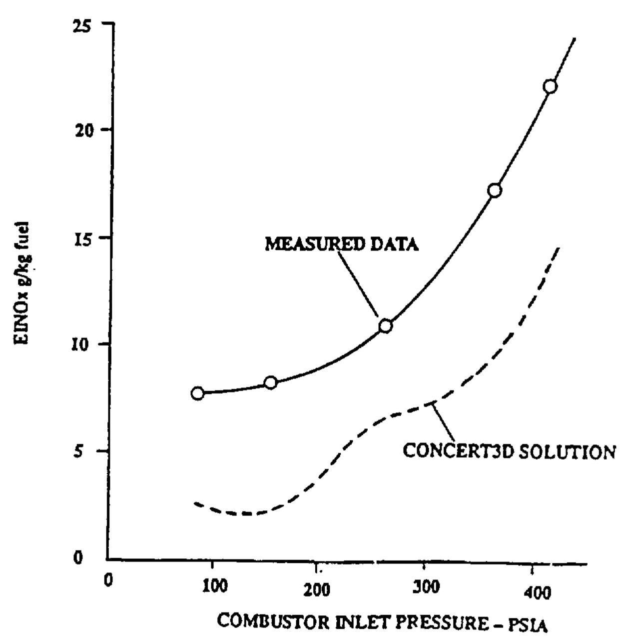
GAS TEMPERATURES (R)

CALCULATED FLOW FIELD IN PLANE IN LINE WITH SWIRLCUPS

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)



CONCERT3D vs. RIG DATA COMPARISON FOR NASA/GE E3 COMBUSTOR
(NOx EMISSIONS)



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

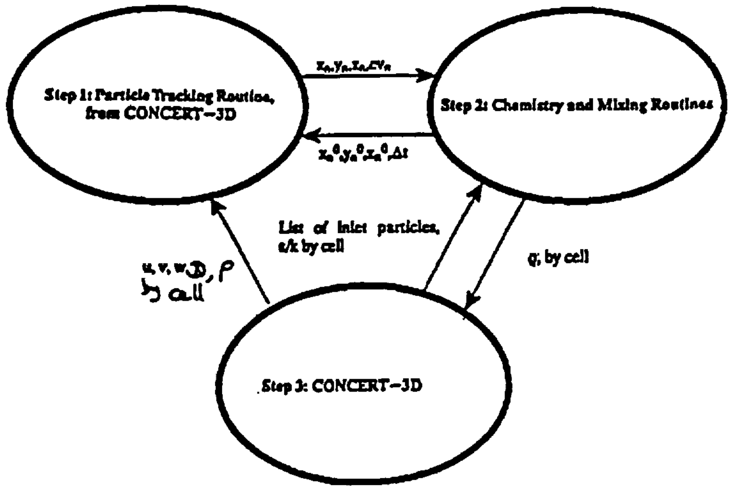
INTRODUCES;

- 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 APPROPIATE REDUCED MECHANISMS

DEVELOPMENT HAS BEEN UNDERWAY SINCE 1992

- 3D CODE DEVELOPMENT INITIATED IN MID YEAR 1993

HYBRID CONCERT CFD / MONTE–CARLO MODELING APPROACH



SCHEMATIC OF COMMUNICATIONS IN THE COMBINED CONCERT / MONTE–CARLO MODELING

HYBRID CONCERT CFD / MONTE–CARLO MODELING APPROACH

BETA TESTING INITIATED BEGINNING OF 1994

> FOCUSED ON FAST CHEMISTRY CALCULATIONS AND OPTIMIZING COMPUTATIONAL EFFICIENCY

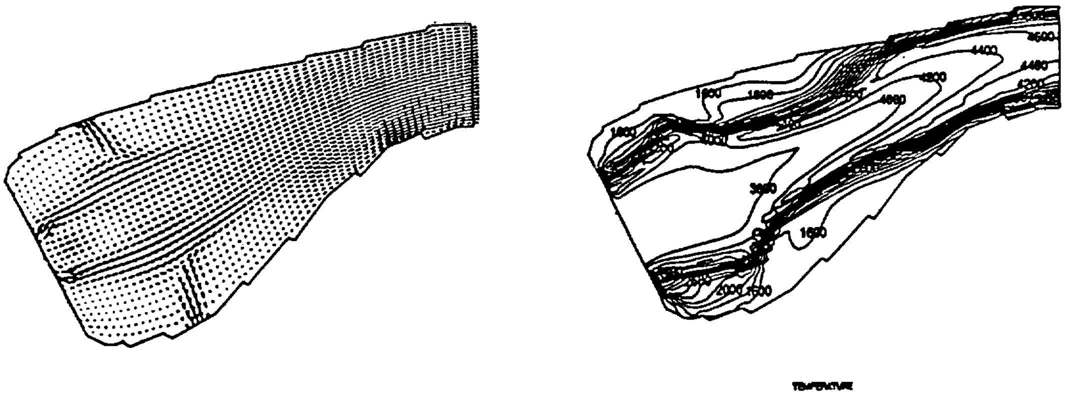
SIGNIFICANT IMPROVEMENT IN COMPUTATIONAL EFFICIENCY ACHIEVED

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

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

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.

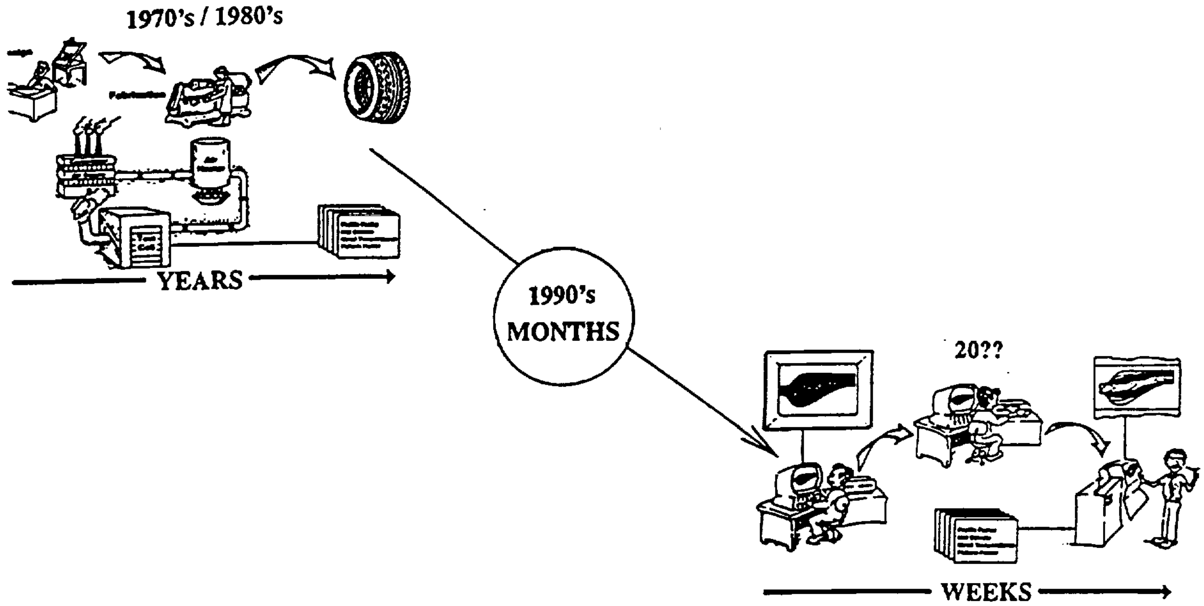
HYBRID CONCERT CFD / MONTE–CARLO MODELING APPROACH

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



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

Computational Turbulence 1994	1 ---- 6	Calculation of turbulent heat transfer in "cluttered spaces", by BRIAN SPALDING Topic 1: The WDIS & WGAP calculation.

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 quantitities) by solving:

$$\text{div grad } L = -1$$

with L fixed to zero in solids.

Computational Turbulence 1994	2 ---- 6	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 fucntion of the local values of L and its gradient.

Computational Turbulence 1994	3 ---- 6	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.

Computational Turbulence 1994	4 ---- 6	Calculation of turbulent heat transfer in "cluttered spaces", by BRIAN SPALDING Topic 2. The LVEL model.

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 plsces.

* 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 nuplus-versus-uplus relation which fits the laminar, transitional & full-turbulent ranges very well Only local velocity and WDIS (wall distance) are needed.

Computational Turbulence 1994	5 ---- 6	Outline of the theory

The u-plus versus y-plus formula of Spalding (1961) is employed namely:

```
        y+ = u+ + (1/E) * [ exp(K*u+) - 1 - K*u+ - (K*u+)**2/2
                                    - (K*u+)**3/6 - (K*u+)**4/24 ]
```

which implies the formula for dimensionless effective viscosity:

```
        v+ = 1 + (K/E) * [ exp(K*u+) - 1 - K*u+ - (K*u+)**2/2
                                              - (K*u+)**3/6 ]
```

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.

Computational Turbulence 1994	6 ---- 6	The results

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

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

SIMULATION ISSUES:

- o NUMERICS (accuracy, convergence)
- o GEOMETRY (body-fitted grids, unstructured grids)
- o COMPUTATIONAL RESOURCES (Time, Storage)

JOINT VELOCITY-SCALAR PDF METHOD

SIGNIFICANT MILESTONES AND RECENT PROGRESS

o 2-D and 3-D time dependent flows (with finite-volume method)
(Anand et al. 1987, Haworth & El Tahry 1989)

o Stochastic dissipation model development and validation
(Pope & Chen 1990, Pope 1991, Anand et al. 1993)

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

o Spray treatment
(Anand 1990)

o Manifold methods for reaction kinetics
(Maas & Pope 1992, 1994; Norris & Pope 1994; Norris & Hsu 1994)

o Solve Poisson equation for mean pressure: $\frac{\partial^2 \langle p \rangle}{\partial x_j \partial x_j} = -\frac{\partial^2}{\partial x_i \partial x_j} \langle \rho U_i U_j \rangle$

o Satisfy continuity by solving for velocity correction potential, velocity correction:

$$\frac{\partial^2 \phi}{\partial x_i \partial x_i} = -\frac{\partial}{\partial x_i} \langle \rho U_i \rangle \quad ; \quad \Delta U_i = \frac{1}{\langle \rho \rangle} \frac{\partial \phi}{\partial x_i}$$

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

o Same descretized form: $\underline{\underline{A}} \cdot \underline{s} = \underline{b}$

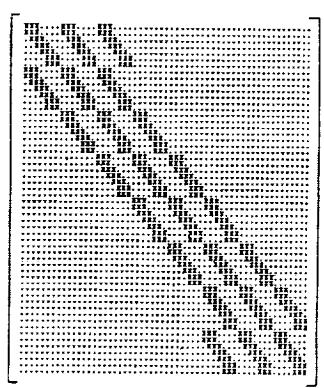
o $\underline{\underline{A}}$ is a banded matrix, constant and same for both <p> and ϕ

o LU decomposition only once

o Special band solver economizes storage and computational effort

o Judicious implementation of the algorithm results in significant economy in computer resource requirement

$\underline{\underline{A}}$ =



M = 7, N = 8, $\underline{\underline{A}}$ (56 x 56)

TURBULENT COMBUSTION MODELING ISSUES

(FOR GAS TURBINE COMBUSTORS)

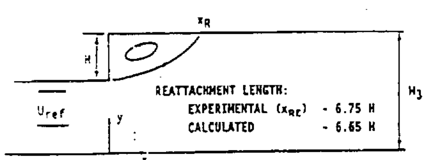
o Most promising method for turbulent reacting flows

ATTRIBUTES OF DIFFERENT PDF METHODS

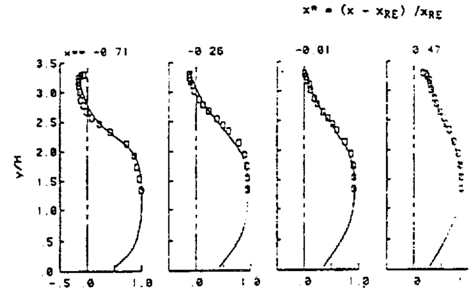
Method	**Attributes**	**Limitations/shortcomings**
Joint PDF of $\underline{\phi}$	Reaction treated exactly	Assumes gradient-diffusion, Does not give velocityfield (requires e.g, k-ε) Turbulence/chemistry interactions not fully simulated
Joint PDF of $\underline{U}$ and $\underline{\phi}$	Reaction exact, Convection (mean and turbulent) exact, Variable-density effects exact	Needs ε equation (or equivalent)
Joint PDF of $\underline{U}$, $\underline{\phi}$, and ω	... In addition Provides complete closure, Treats turbulent streams of different scales, Can account for effects of large scale structures	

PDF CALCULATIONS FOR A RECIRCULATING FLOW

(Anand et al. 1989)

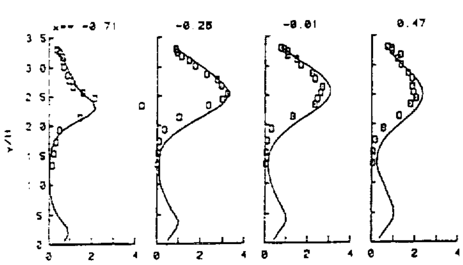


Backward-facing step
Pronchick and Kline (1983)

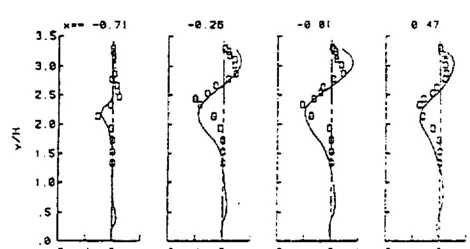


<U>/U ref

Storage: 1.3 Mwords
CPU Time: 6 min. Cray YMP



$10^2 \langle u'^2\rangle/U_{ref}^2$



$10^3 \langle u'^2v'\rangle/U_{ref}^3$

STOCHASTIC DISSIPATION MODEL

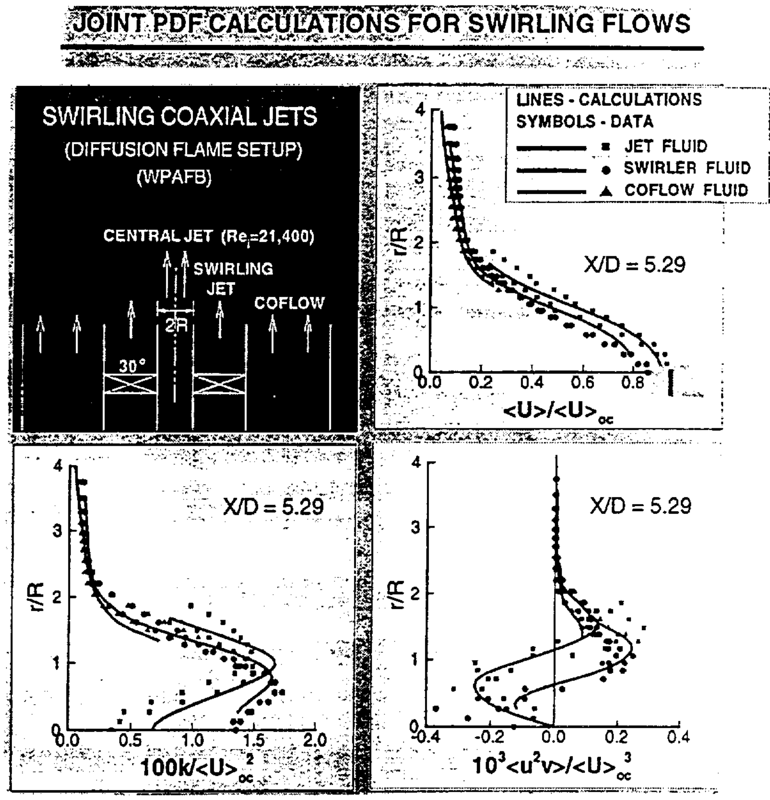
- o Provides complete closure of the PDF equation (joint velocity-frequency-scalar)
- o More realistic than a mean dissipation model. Dissipation (rather, turbulent frequency) is also a random variable and included in the joint PDF.
- o Treats multiple scales in the flow
- o Accounts for internal intermittency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

$$\boxed{d\omega^* = -\omega^* <\omega> (S_\omega + C_\chi \Omega)\, dt + <\omega>^2 h\, dt + \omega^* (2C_\chi <\omega> \sigma^2)^{1/2}\, dW}$$

$$\boxed{dU_i^* = -\frac{1}{\rho}\frac{\partial <P>}{\partial x_i} dt + D_i dt + (C_o \tilde{k}\, \omega^*)^{1/2}\, dW_i}$$

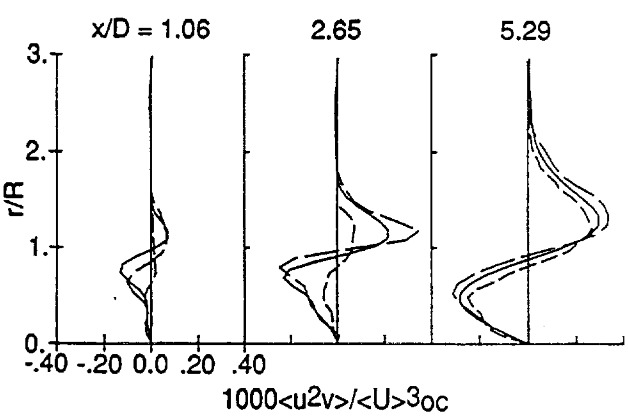
SWIRLING FLOWS

- o No theoretical limitations
- o Additional production terms due to non-zero mean swirl velocity
- o 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
- o 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:

$$<u^2v> = -C_s \frac{k}{<\varepsilon>} <v^2> \frac{\partial <u^2>}{\partial r}$$

$C_s = 0.22$

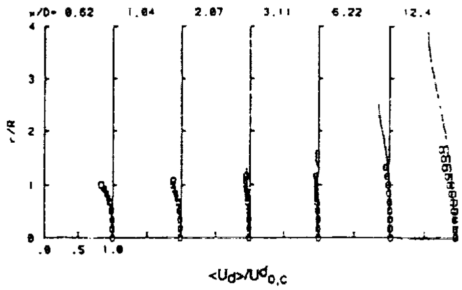
—— PDF CALCULATIONS
– – – RS MODEL WITH RSM RESULTS
— — RS MODEL WITH PDF RESULTS

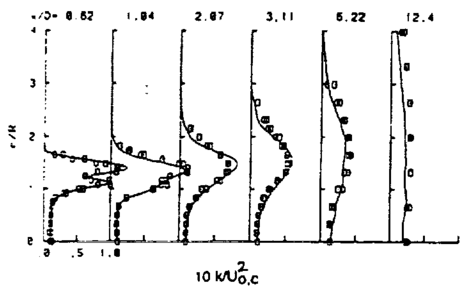
SPRAY CALCULATIONS

(Anand 1990)

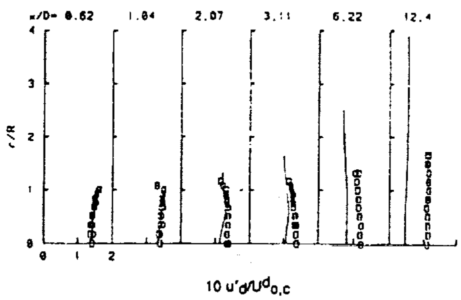
- o Advanced spray models (stochastic Lagrangian, Monte Carlo) naturally compatible with the joint PDF method
- o Assumptions about turbulent kinetic energy partition avoided
- o Effects of gas phase turbulence structure (velocity cross-correlation) included

105 micron glass beads, NASA HOST C data



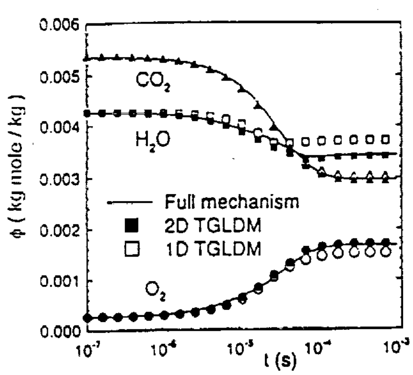


Computed profiles of normalized turbulent kinetic energy of air compared against data.

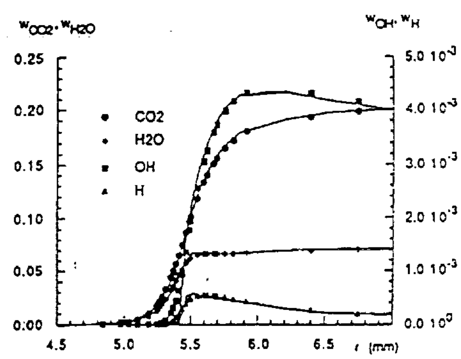


REDUCED KINETICS / MANIFOLD METHODS

- o Low dimensional manifold methods (ILDM, TGLDM)
  - Given detailed kinetcs, they provide low-dimensional description (e.g., 1-D, 2-D, 3-D) in multidimensional composition/scalar space
  - Use dynmical 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

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

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

o 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

o 3-D Flows, Improved solution algorithms

o Parallel processing

o Reduced kinetics / Low Dimensional Manifolds

o Evaporating / reacting sprays

o Emphasis on emissions and performance predictions

OVERVIEW OF TURBULENCE MODEL DEVELOPMENT AND APPLICATIONS AT ROCKETDYNE

N95- 27891

A.H. Hadid, E.D. Lynch, and M.M. Sindir
Rocketdyne Division
Rockwell International
Canoga Park, California

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_t
  - 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-BREMHORST 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 C_{k2} AS A FUNCTION OF TURBULENT MACH NUMBER $\sqrt{\rho k/\gamma p}$
    - ZEMAN MODIFICATION (1990)
    - SARKAR (1990, 1991) AND WILCOX (1991) PROPOSALS
  - MODIFICATIONS OF ZEMAN AND SARKAR NOT RECOMMENDED
- **HEAT TRANSFER OVER PREDICTED NEAR SHOCK WAVES**
  - LIMIT TURBULENT LENGTH SCALE L_t TO $\mathrm{MIN}\left(\frac{k^{3/2}}{\varepsilon}, \frac{Ky}{C_\mu^{3/4}}\right)$ (VUONG AND COAKLEY, 1987)
- **SEPARATION UNDERPREDICTED IN RAPID COMPRESSION OR STRAIN REGIONS**
  - INCREASE α_ε OR α_ω UNDER RAPID COMPRESSION (VUONG AND COAKLEY)
- **HEAT TRANSFER OVER PREDICTED FOR VERY COLD WALLS $T_w/T_{aw}<0.1$ (COAKLEY)**
  - CEBECI-SMITH ~ 60%, k-ω ~ 40%, q-ω ~ 10%, k-ε ~ 30%

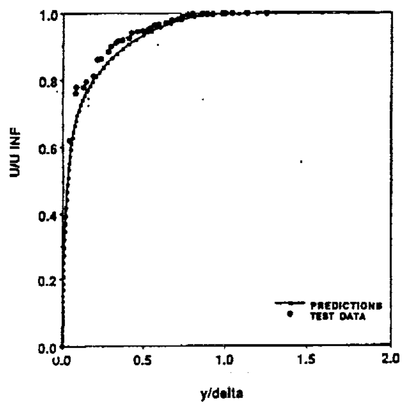
TURBULENCE MODELS ADAPTED TO USA CODE

ALGEBRAIC	DAMPING WALL	DAMPING LOCAL	BOUNDARY CONDITIONS	TRANSITION MODEL: HIGH ORDER POLYNOMINAL	TRANSITION MODEL: ARNAL	COMPRESSIBILITY EFFECTS: MIXING LAYER SPREADING	COMPRESSIBILITY EFFECTS: SEPARATION EXTENT	COMPRESSIBILITY EFFECTS: REATTACHMENT HEAT TRANSFER
Baldwin-Lomax	X	3 Versions		X	X			
k-ε								
1. Myong-Kasagi	X	X	$k=0$, $\varepsilon=\nu_1 u^2/18y^2$	X		1. Sarkar (1991) 2. Zeman (1990) 3. Wilcox (1991)	4. Yuong & Coakley (1987)	5. Vuong & Coakley (1987)
2. Chien (1982)	X	X	$k=0$, $\varepsilon=0$	X		1., 2., 3.	4.	5.
3. Jones-Launder (1972)	X	X	$k=0$, $\varepsilon=0$	X		1., 2., 3.	4.	5.
4. Launder-Sharma (1974)	X	X	$k=0$, $\varepsilon=0$	X		1., 2., 3.	4.	5.
5. Huang-Coakley (1992)	X	X	$k=0$, $\varepsilon=2\nu_1 k_1/y^2$	X		1., 2., 3.	4.	5.
6. Speziale-So-Zhang (1993)	X	X	$k=0$, $\varepsilon=2\nu_1 k_1/y^2$	X		1., 2., 3.	4.	5.
7. Lam-Bremhorst (1981)	X	X	$k=0$, $\varepsilon_y=0$	X		1., 2., 3.	4.	5.
8. High Re	X	X	Wall Function	X				
k-ω								
1. High Re Wilcox (1991a)	–	–	$k=0$, $\omega_0=10\omega_1$	X		1., 2., 3.	4.	5.
2. Low Re Wilcox (1991b)	–	–	$k=0$, $\omega=7.2\nu_1/y^2$	X		1., 2., 3.	4.	5.
q-ω								
Coakley (1987)			$k=0$, $\omega_y=0$	X				
One-Equation (Goldberg, Two-Time Scale 1992)	–		$k=0$					
One-Equation R_T (Goldberg 1993, 1994)		X	$\frac{d(k^2/\varepsilon)}{dy}=\frac{d(\nu_1 R_1)}{dy}=0$	X				

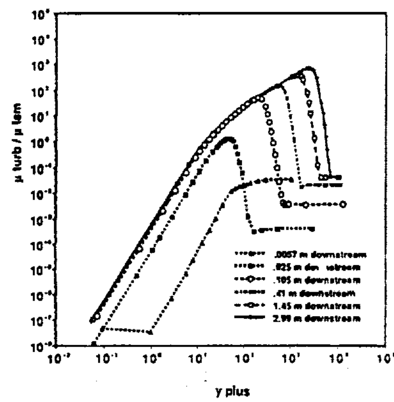
M = 9.2 FLAT PLATE FLOW

- CHIEN k-ε MODEL WITH RAPID COMPRESSION AND LENGTH SCALE COMPRESSIBILITY MODIFICATIONS

VELOCITY PROFILE



TURBULENT VISCOSITY

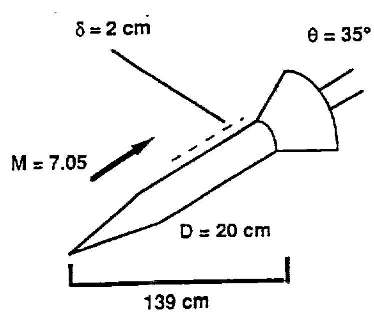


REF: G.T. COLEMAN AND J.L. STOLLERY, JFM 56: 741, "HEAT TRANSFER FROM A HYPERSONIC TURBULENT FLOW AT A WEDGE COMPRESSION CORNER"

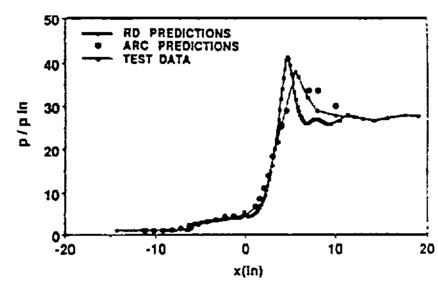
MACH 7.05 FLOW OVER AXISYMMETRIC FLARE

CHIEN k-ω MODEL WITH RAPID COMPRESSION AND LENGTH SCALE COMPRESSIBILITY MODIFICATIONS

AXISYMMETRIC FLARE



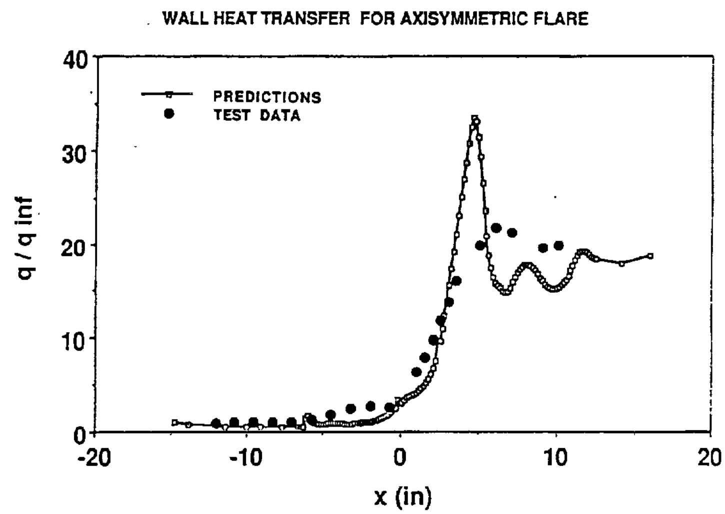
WALL PRESSURE FOR AXISYMMETRIC FLARE



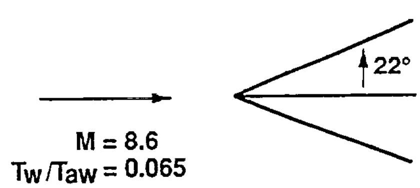
REF: M.I. KUSSOY AND C.C. HORSTMAN, "DOCUMENTATION OF TWO- AND THREE-DIMENSIONAL HYPERSONIC SHOCK-WAVE TURBULENT BOUNDARY LAYER INTERACTION FLOW," NASA TM 1-01075.

MACH 7.05 FLOW OVER AXISYMMETRIC FLARE

CHIEN k-ω MODEL WITH RAPID COMPRESSION AND LENGTH SCALE COMPRESSIBILITY MODIFICATIONS



MACH 8.6 FLOW OVER COLD WALL WEDGE



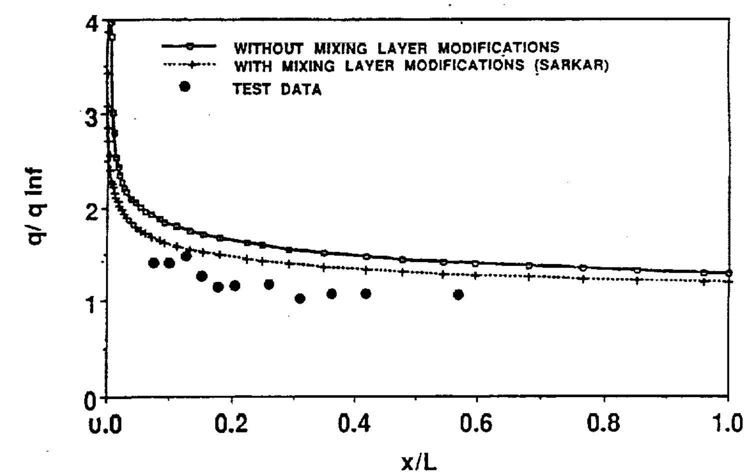
THREE STUDIES

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

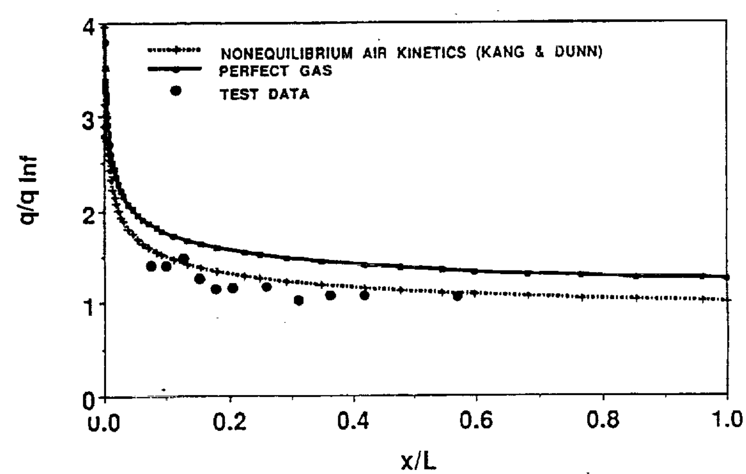
HEAT FLUX CALCULATIONS



MACH 8.6 FLOW OVER COLD WALL WEDGE

HIGH-Re k-ω MODEL WITH VARIOUS AIR CHEMISTRY MODELS

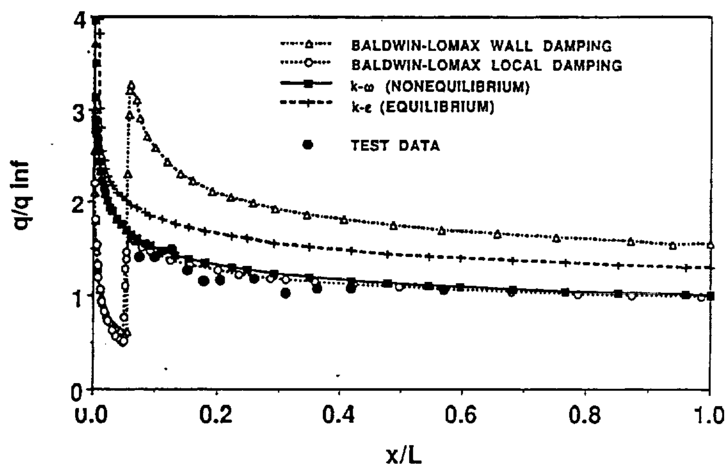
HEAT FLUX CALCULATIONS



MACH 8.6 FLOW OVER COLD WALL WEDGE

BALDWIN LOMAX, k-ε, k-ω MODEL COMPARISONS

HEAT FLUX CALCULATIONS

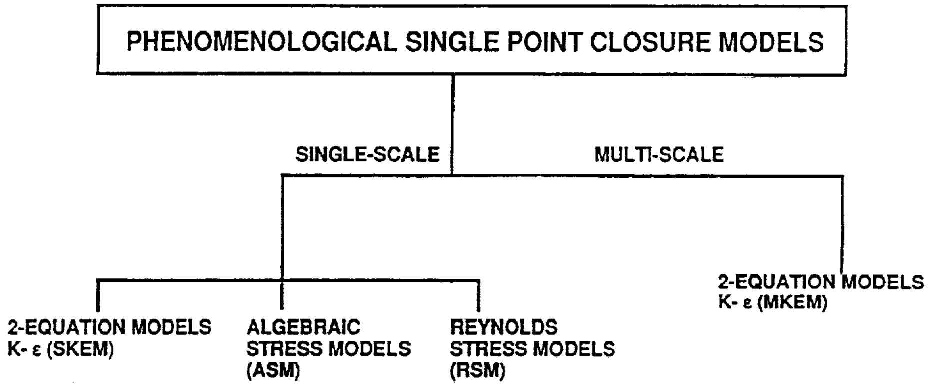


LOW SPEED TURBULENCE MODELING

- **EMPHASIS IS ON THE DEVELOPMENT OF ENGINEERING TURBULENCE MODELS FOR**
  - ROTATING MACHINERY
  - FLOW IN DUCTS AND MANIFOLDS
  - REACTING FLOWS
- **APPROACH TAKEN IS TO**

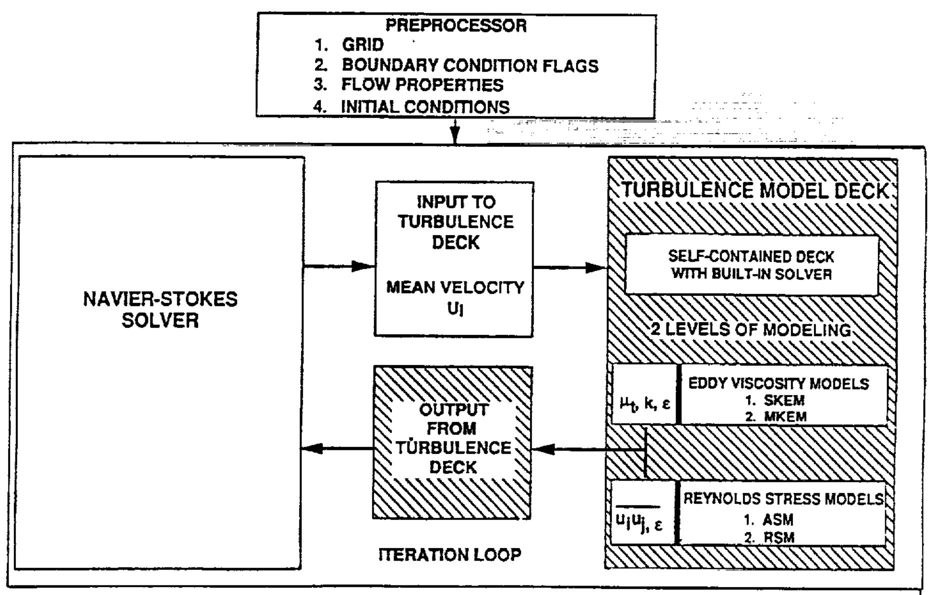
1. SYSTEMATICALLY ASSESS EXISTING PHENOMENOLOGICAL MODELS USING COMMON NAVIER-STOKES SOLVER
2. IDENTIFY, DEVELOP AND VALIDATE MODEL UPGRADES COMMENSURATE WITH OBSERVED FLOWPHYSICS
3. DEVELOP SELF-CONTAINED TURBULENCE MODEL DECKS (MODULES) THAT CAN BE INTEGRATED WITH NAVIER-STOKES SOLVERS
4. PROVIDE GUIDANCE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

TURBULENCE MODELS BEING ASSESSED



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



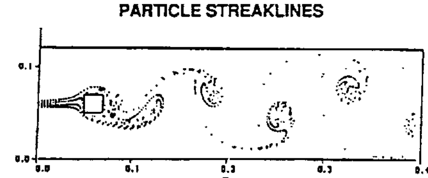
PROJECT WELL UNDERWAY

- **TEAM**
  - MODELS PROVIDED BY UMIST, LERC/ICOMP, ARC/CTR
  - MODULE DEVELOPMENT BY ROCKETDYNE
  - MODULE TESTING BY ROCKETDYNE (**REACT**, **USA**) AND UAH (**MAST**)
  - MODEL UPGRADES BY ROCKETDYNE, UMIST, ARC/CTR
  - APPLICATION BY ROCKETDYNE 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**

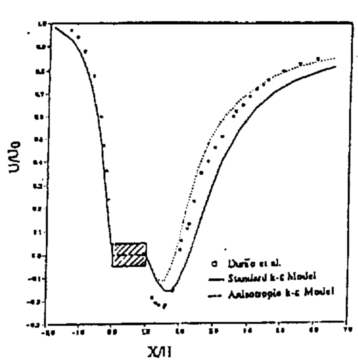
NONLINEAR ALGEBRAIC-STRESS MODEL

VORTEX SHEDDING FROM RECTANGULAR CYLINDERS (DURAO, et al)

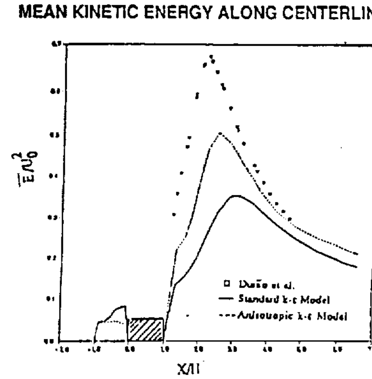
PARTICLE STREAKLINES



MEAN AXIAL VELOCITY ALONG CENTERLINE



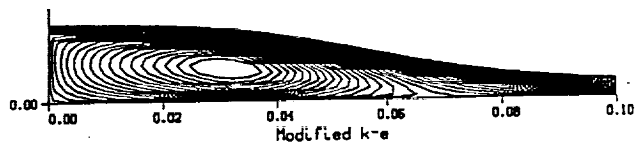
MEAN KINETIC ENERGY ALONG CENTERLINE



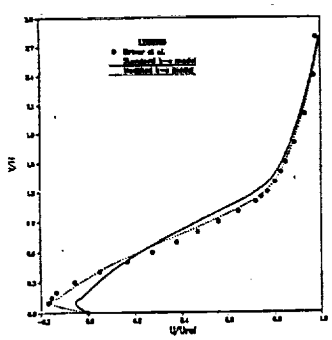
ROTATION MODIFIED k-ε MODEL

BACKWARD FACING STEP (DRIVER AND SEEGMILLER)

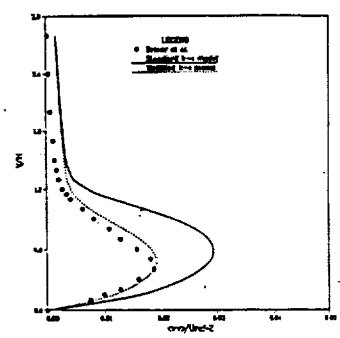
STREAMLINE CONTOURS



MEAN AXIAL VELOCITY AT X/M=4



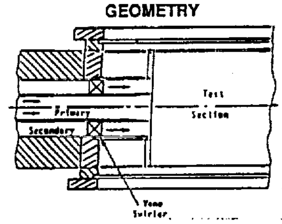
RADIAL TURBULENT INTENSITY ($\overline{vv}/u^2_{ref}$)



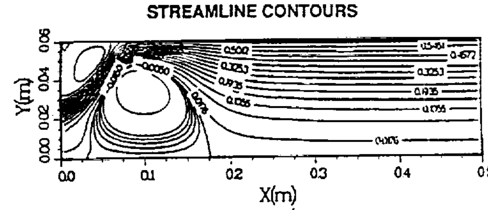
ALGEBRAIC STRESS MODEL

CONFINED COAXIAL SWIRLING JET FLOW (ROBACK AND JOHNSON)

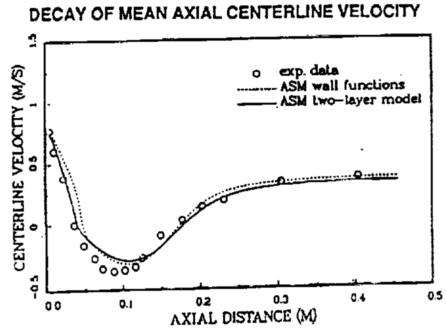
GEOMETRY



STREAMLINE CONTOURS

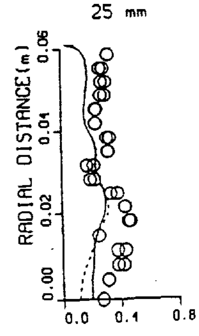


DECAY OF MEAN AXIAL CENTERLINE VELOCITY



RADIAL PROFILES OF $\overline{uu}$

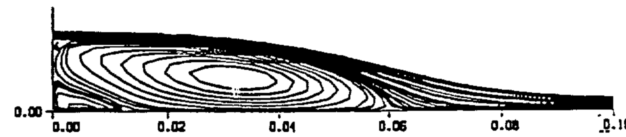
25 mm



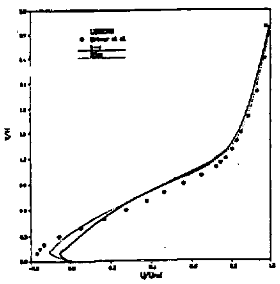
REYNOLDS STRESS MODEL (LRR – MODEL)

BACKWARD FACING STEP (DRIVER AND SEEGMILLER)

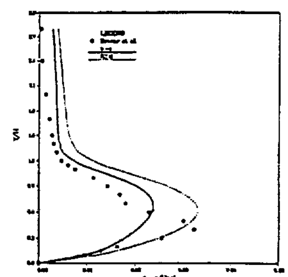
STREAMLINE CONTOURS



MEAN AXIAL VELOCITY AT X/H=4



AXIAL TURBULENT INTENSITY AT X/H=4



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

RECENT ADVANCES IN PDF MODELING OF TURBULENT REACTING FLOWS

A.D. Leonard and F. Dai
CFD Research Corporation
Huntsville, Alabama

N95- 27892

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

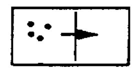
- Computational Requirements Increases Exponentially With Dimensions for Finite Difference Methods
- Computational Requirements Increase Linearly With Dimensions for Monte Carlo Methods

COMPOSITION PDF SOLUTION

Stochastic Lagrangian Particle Simulation

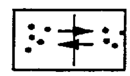
Particle Composition and Position Changed to Model Transport of Joint PDF

- Mean Convection
  - Move Particles Between Cells



- Chemical Reactions
  - Lookup Table Holds Composition Change

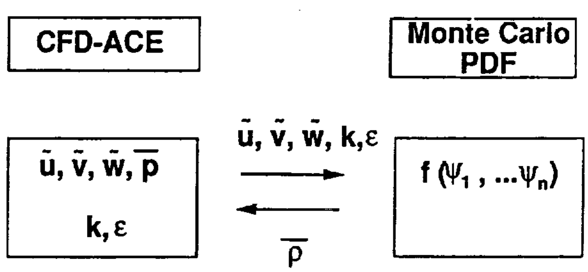
- Turbulent Diffusion
  - Exchange Particles Between Cells



- Molecular Mixing
  - Particle Interaction Changes Composition

COUPLING

PDF Solution is Separate Module



CHEMICAL KINETICS

Reduced Models are Used

Hydrogen: $2H_2 + O_2 \Leftrightarrow 2H_2O$

CO:

$CO + H_2O \Leftrightarrow CO_2 + H_2$

$2H_2 + O_2 \Leftrightarrow 2H_2O$

Methane:

$CH_4 + 2H + 2H_2O \rightarrow CO + 4H_2$

$CO + H_2O \Leftrightarrow CO_2 + H_2$

$2H_2 + O_2 \rightarrow 2H_2O$

$3H_2 + O_2 \Leftrightarrow 2H_2O + 2H$

Hydrocarbon:

$C_nH_{2n+2} + (\frac{n}{2})O_2 \rightarrow n\ CO + (n+1)\ H_2$

$C_nH_{2n+2} + n\ H_2O \rightarrow n\ CO + (2n+1)\ H_2$

$CO + H_2O \Leftrightarrow CO_2 + H_2$

$2H_2 + O_2 \Leftrightarrow 2H_2O$

Thermal NO:

$N_2 + O \Leftrightarrow NO + N$

$N + O_2 \Leftrightarrow NO + O$

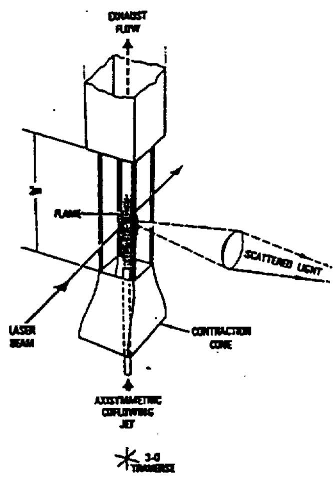
$N + OH \Leftrightarrow NO + H$

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



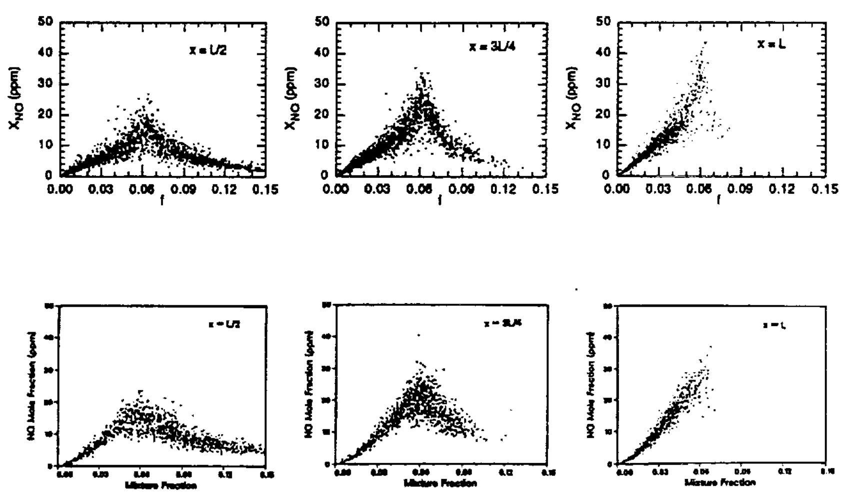
Re ≈ 10^4

Fuel

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

60% HYDROGEN FLAME

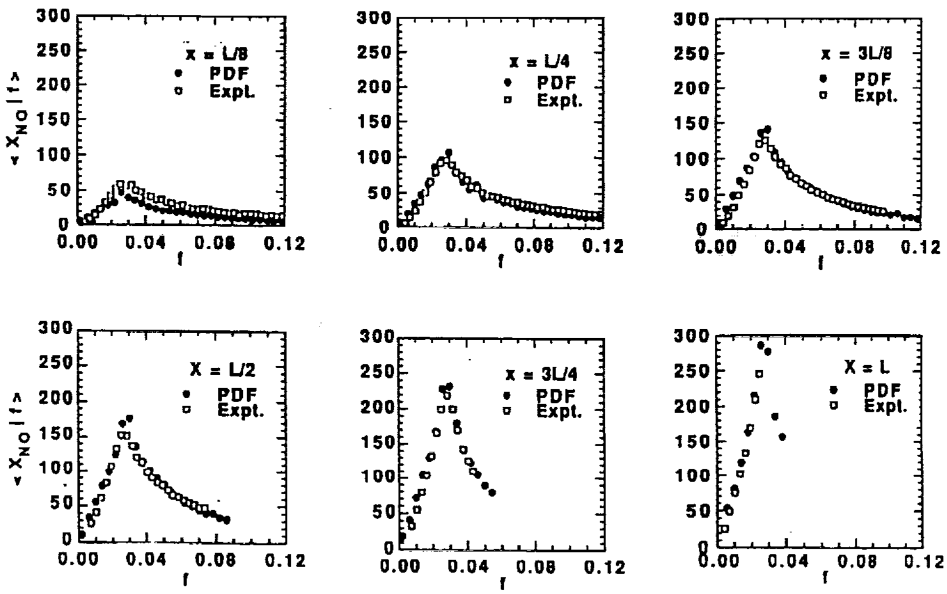
Scatter Plots of Mixture Fraction and NO Mole Fraction



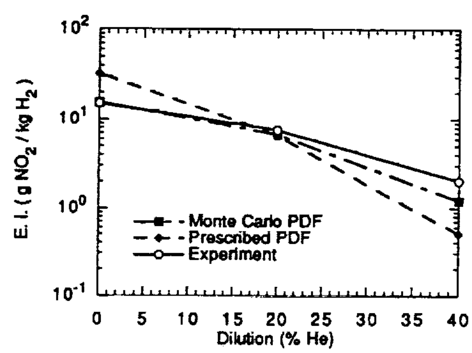
PDF Results

Experimental Data

UNDILUTED HYDROGEN FLAME
Conditional Averged NO Mole Fraction

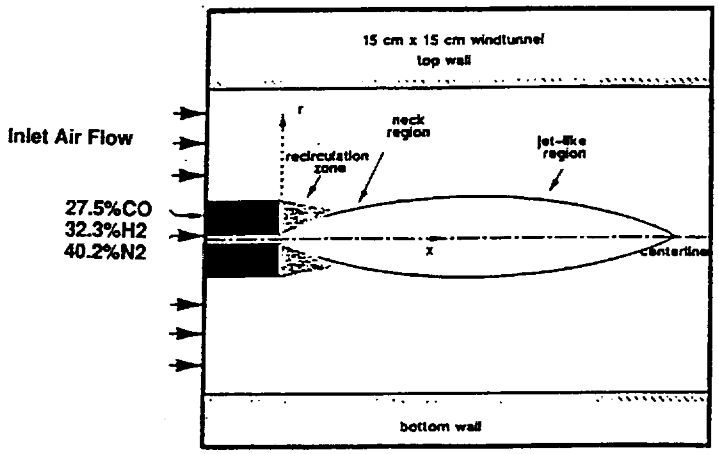


HYDROGEN DIFFUSION FLAME
Dilution Effects on Emmisions Index



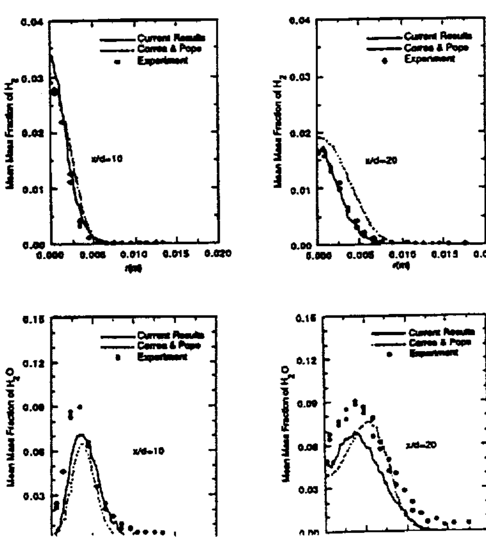
BLUFF BODY STABILIZED DIFFUSION FLAME

Illustration of Experiment of Correa and Gulati



BLUFF BODY STABILIZED DIFFUSION FLAME

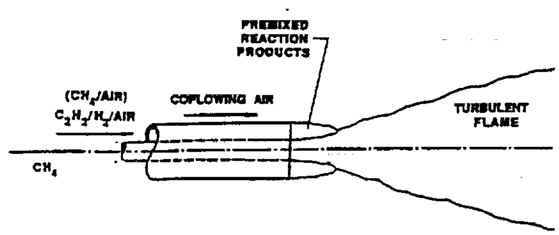
Composition PDF Predicts Mean Values as well as Velocity-Composition PDF



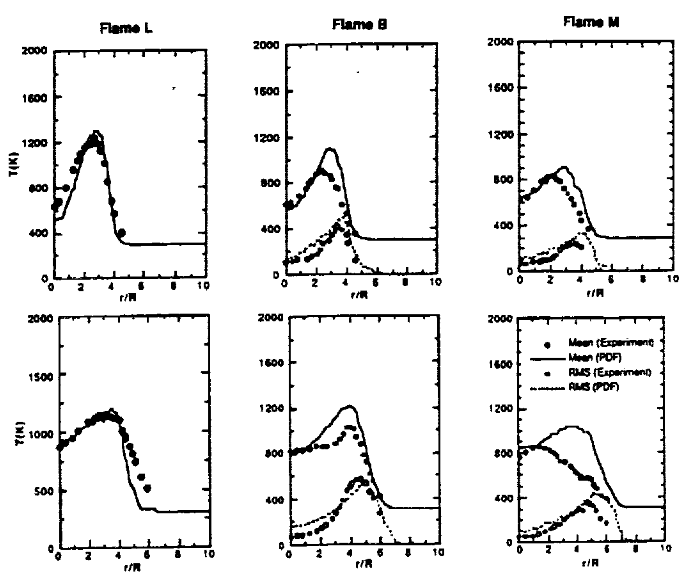
PILOTED JET DIFFUSION FLAME
Illustration of Experiment of Masri et.al.

Experimental Conditions

Flame	Fuel Jet Velocity
L	41 m/s
B	48 m/s
M	55 m/s



PILOTED JET DIFFUSION FLAME
Good Agreement with Experimental Data



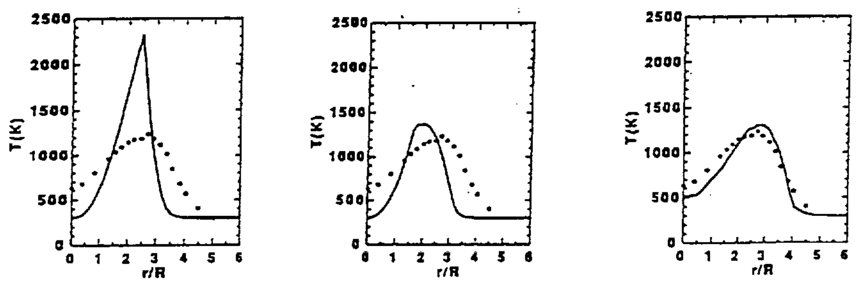
x/D = 20

x/D = 30

PILOTED JET DIFFUSION FLAME

More Accurate Prediction with Monte Carlo PDF

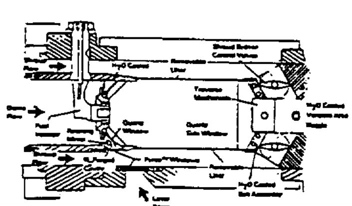
No PDF Prescribed PDF Monte Carlo PDF

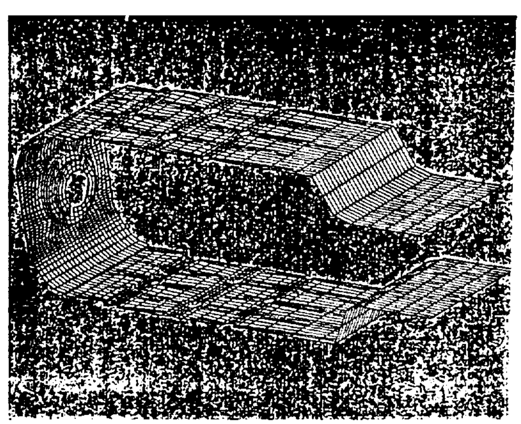


Flame B at x = 20D

GENERIC GAS TURBINE COMBUSTOR

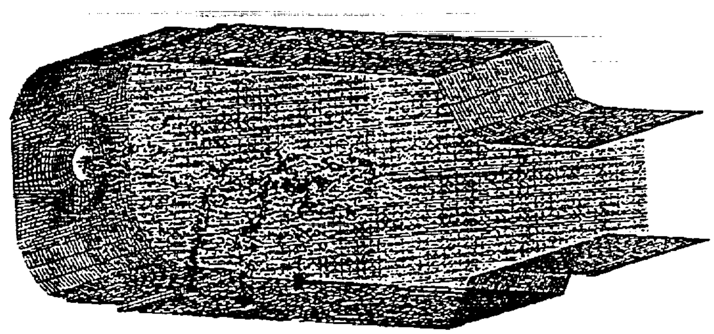
Pratt & Whitney Four-Nozzle Sector Combustor Tested at Wright Laboratory



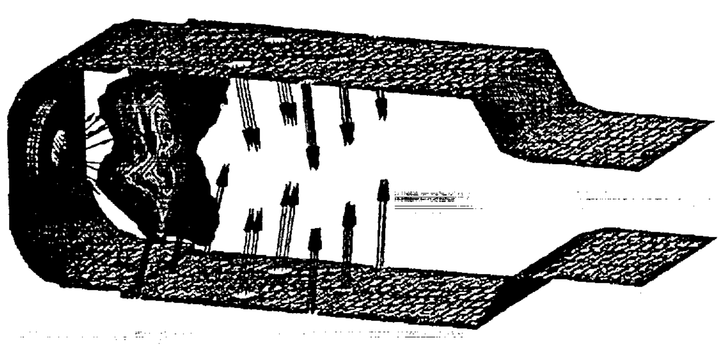


67,840 Cells

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)

Conventional CFD

CPU Time	20 hours
Memory	80 MBytes

Monte Carlo PDF

CPU Time	100 hours
Memory	120 MBytes

Parallel PDF (Projected)

CPU Time	25 hours	25 hours	25 hours	25 hours
Memory	30 MBytes	30 MBytes	30 MBytes	30 MBytes

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

EXPERIENCE WITH TURBULENCE INTERACTION AND TURBULENCE-CHEMISTRY MODELS AT FLUENT INC.

D. Choudhury, S.E. Kim, D.P. Tselepidakis,
and M. Missaghi
Fluent Inc.
Lebanon, New Hampshire

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, FLUENTand 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, ϵ.
  - 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-ε model adequate for simple flows with no significant strain rates.
- RNG k-ε 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-ε 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 ε-equation, making their fidelity limited.
- The constants in the modeled equations are calibrated against simple benchmark experiments.
- As a result, the k-ε model performs poorly in flows with curvature, swirl, rotation, separated flows, low-Reynolds number flows, strongly anisotropic flows, etc.

Renormalization Group (RNG) Based k-ε 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-ε equations remains largely the same with the standard k-ε model. But, the constants in the model equations are derived explicitly from theory.
- The ε-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:

$$
\begin{aligned}
\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} &= P_k - \varepsilon + \frac{\partial}{\partial x_i}\left(\frac{\nu_T}{\sigma_k}\frac{\partial k}{\partial x_i}\right) \\
\frac{\partial \varepsilon}{\partial t} + U_i \frac{\partial \varepsilon}{\partial x_i} &= 1.42\,\frac{\varepsilon}{k}P_k - 1.68\,\frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_i}\left(\frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right) - \Re
\end{aligned}
$$

where:

$$
\begin{aligned}
\sigma_k &= \sigma_\varepsilon = 0.7179 \\
P_k &= 2\nu_T S_{ij} S_{ij} \text{ is the kinetic energy production} \\
S_{ij} &= \tfrac{1}{2}\left(\tfrac{\partial U_i}{\partial x_j} + \tfrac{\partial U_j}{\partial x_i}\right) \text{ is the mean rate of strain tensor} \\
\nu_T &= C_\mu \tfrac{k^2}{\varepsilon} \\
\Re &= \frac{C_\mu \eta^3 \,(1-\frac{\eta}{\eta_0})}{1+\beta\,\eta^3}\,\frac{\varepsilon^2}{k}
\end{aligned}
$$

$\eta = Sk/\varepsilon,\ S = (2\,S_{ij}\,S_{ij})^{\frac{1}{2}}$

$\eta_0 = 4.38,\ \beta = 0.015$

RNG-Based k-ε Model (Cont'd)

- In the low-Re RNG model, a differential relationship exists between $\frac{k}{\sqrt{\varepsilon}}$ and ν_{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 $\hat{\nu} \longrightarrow 1, \alpha \longrightarrow \alpha_0$ (the low-Re limit) and as $\hat{\nu} \longrightarrow \infty$, $\sigma = \alpha^{-1} \longrightarrow 0.7179$ (the high-Re limit). Here:

  $\hat{\nu} = \nu_{\text{eff}}/\nu_0$, where $\nu_{\text{eff}} = \nu_0 + \nu_T$

  $\alpha =$ inverse turbulent Prandtl number (σ^{-1})

  $\alpha_0 =$ inverse molecular Prandtl number (σ_0^{-1})

- In the low-Re regions, σ_k and σ_ε 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 $\hat{\nu} >> 1$:

$$\nu_{\text{eff}} = \nu_T = 0.085\,\frac{k^2}{\varepsilon}$$

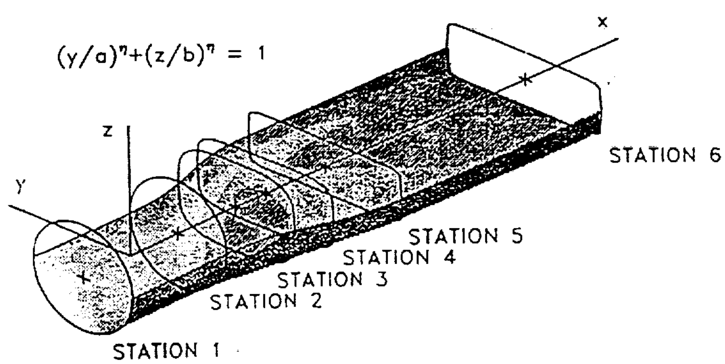
  and the effective viscosity varies smoothly from the molecular viscosity to the turbulent viscosity.
- The low-Re eddy-viscosity formula does not explicitly involve any geometric length scale, i.e., the distance from a solid wall used in the damping functions adopted by most low-*Re* near-wall models, 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-ε model also works well with conventional and enhanced (non-equilibrium) wall functions available in Fluent Inc.codes.

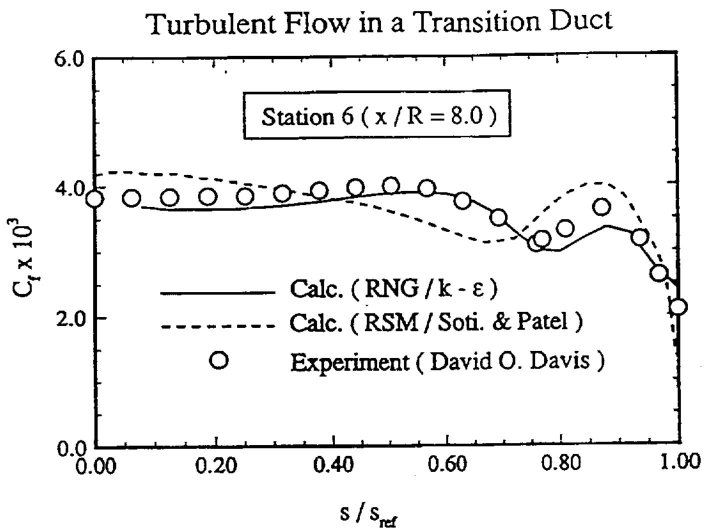
The Reynolds-Stress Model in FLUENT

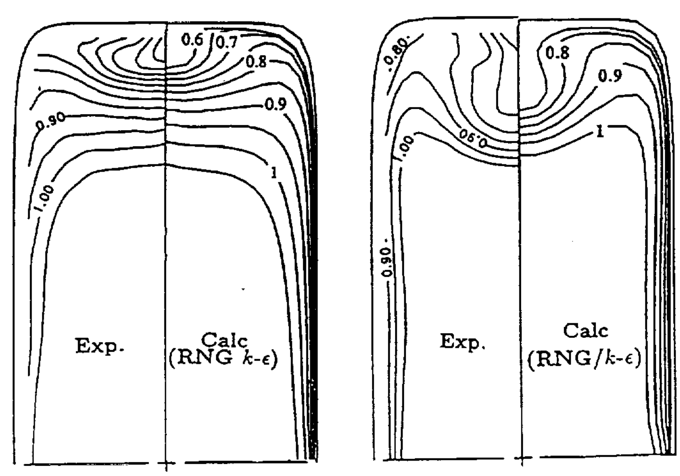
- RSM solves transport equations for the Reynolds stresses: $\overline{u_i' u_j'}$ (4 equations in 2D problems, 6 equations in 3D problems).
- RSM is the level of modeling that has a well established track record of out-performing eddy-viscosity models in complex flows.
- It is computationally more expensive 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

- Measured by Davis (1991).
- $Re_D = 3.9 \times 10^5$.
- Solution Domain.
  - Upstream Inlet Boundary: $x/D = -1.0$
  - Downstream Exit Boundary: $x/D = 8.0$
  - A Quadrant of the duct modeled.



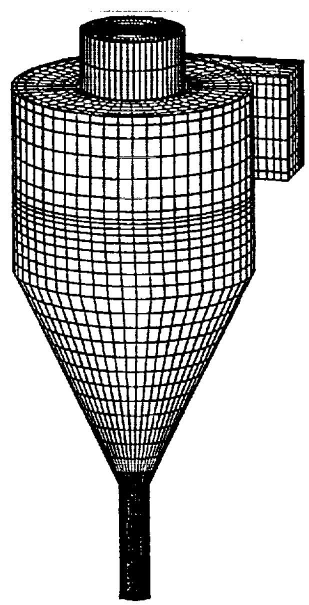


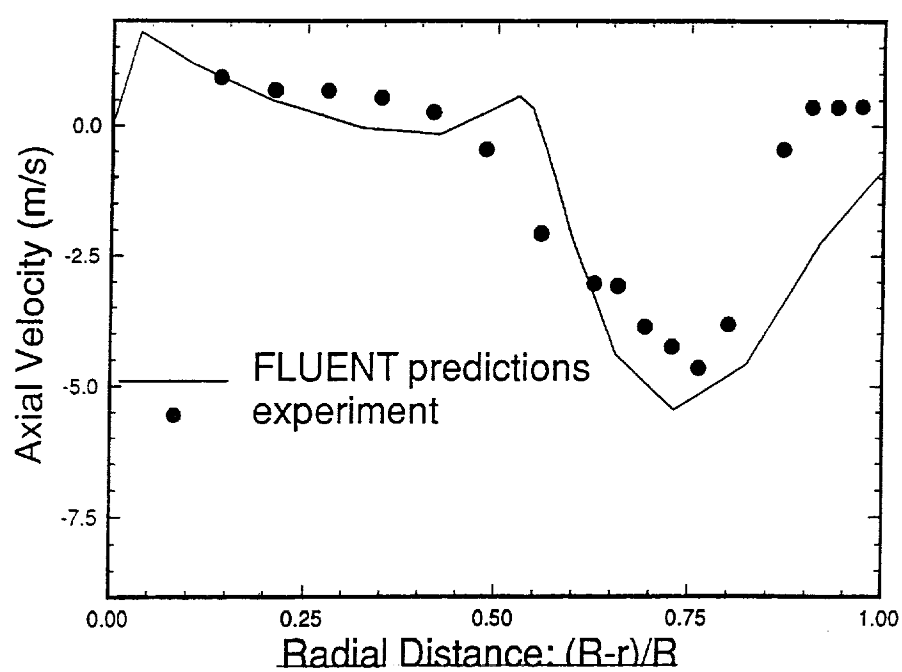


Contours of computed streamwise velocity
(RNG-based k-ϵ model)

Example 2:
Cyclone Sparator

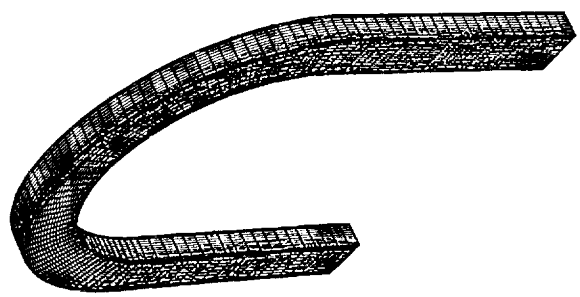
- Measured by Qing (1983).
- RSM is used.
- Cylindrical 55 × 23 × 41 grid.

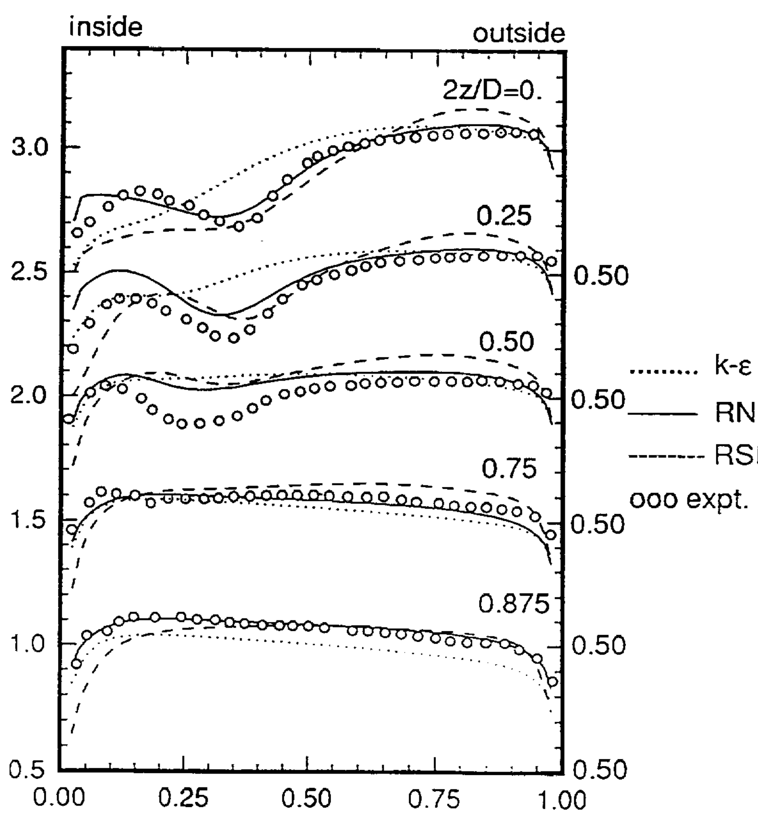




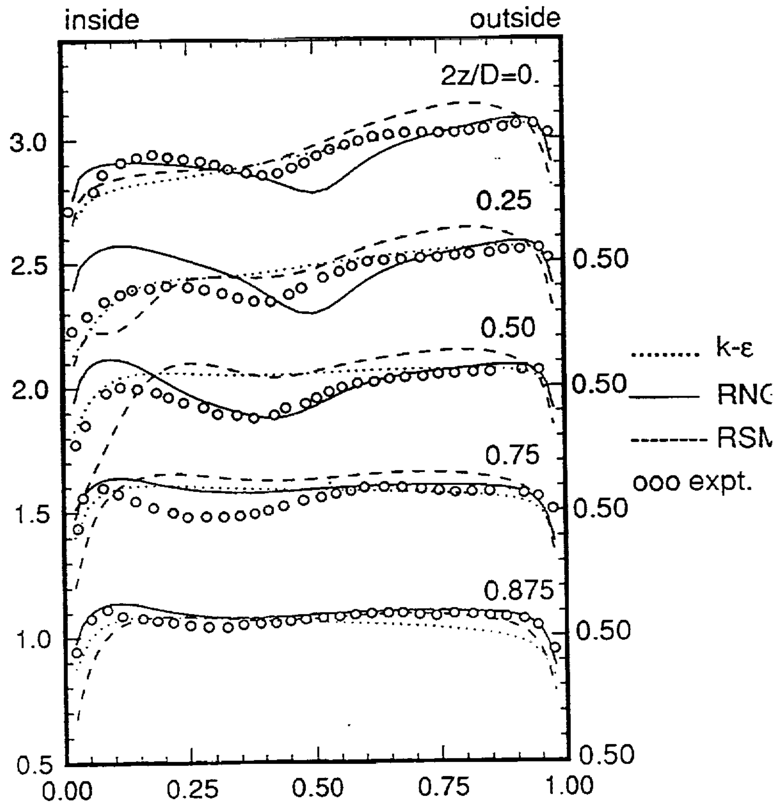
Example 3:
180° Bend of Square-Cross Section

- Solution Domain
  - Upstream Boundary: $5.0D_H$ from the start of the bend
  - Downstream Boundary: $5.0D_H$ from the end of the bend
  - A symmetric half of the duct modeled.
- Mesh
  - Orthogonal $101 \times 47 \times 27$
  - Distance from the wall $\approx 0.01D_H$





90° Station



135° Station

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:

$$\bar{r}(\theta_1, \theta_2, ...) \neq r(\bar{\theta}_1, \bar{\theta}_2, ...)$$

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

$$Da = \frac{\tau_F}{\tau_r} = \frac{\ell}{\ell_f} \frac{U_\ell}{U'}$$

- When $Da << 1$ 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:

$$\bar{\theta} = \int_0^1 \theta(\xi) P(\xi) d\xi$$

- 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:

$$\bar{r} = \int_0^1 r(c) p(c) dc$$

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:

$$\frac{\partial}{\partial x_i}(\rho u_i \overline{\xi}) = \frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_t}\frac{\partial \overline{\xi}}{\partial x_i}\right)$$

$$\frac{\partial}{\partial x_i}(\rho u_i \overline{\xi'^2}) = \frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_t}\frac{\partial \overline{\xi'^2}}{\partial x_i}\right) + \frac{2\mu_t}{\sigma_t}\left(\frac{\partial \overline{\xi}}{\partial x_i}\right)^2 - C_d \rho \frac{\varepsilon}{k}\overline{\xi'^2}$$

- 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

N95- 27894

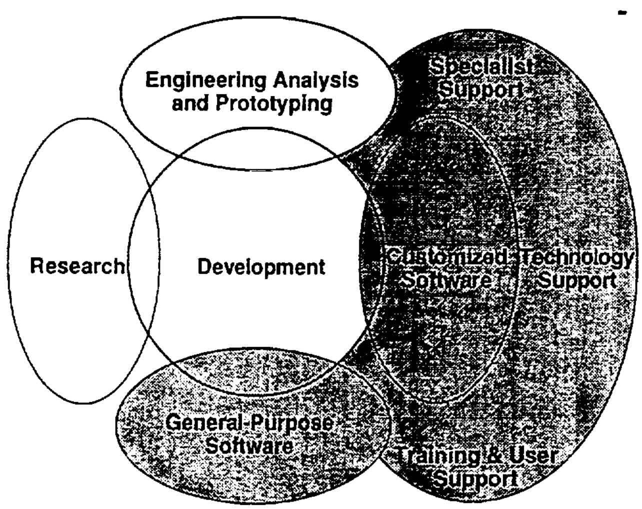
Ashok K. Singhal, Yong G. Lai, and Ram K. Avva
CFD Research Corporation
Huntsville, Alabama

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.
  - Stanford Endeavor: "Collaborative Testing of Turbulence Models" 1989-1993
  - 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-ε Model (Launder & Spalding, 1974)
- Low-Re k-ε Model (Chien, 1982)
- Extended k-ε Model (Chen & Kim, 1987)
- Multiscale k-ε Model (Kim & Chen, 1988)
- RNG-Based k-ε Model (Yakhot et. al. 1993)
- 2-Layer k-ε Model (Rodi, 1991)
- k~ε^{++} Models
- k-ω Model (Wilcox, 1991)

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

NUMERICAL DIFFICULTIES

- Positivity of k & ε (or ω) Is Not Guaranteed in Iterative Algorithms
- Strong Nonlinearity of Source Terms and Coupling Causes Numerical Difficulties
- Inappropriate Specifications of ε (or ω) at Boundaries or in Initial Conditions May Also Cause Divergence
- Non-orthogonaltiy 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° 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

Model/Expt.	Time Period	Strouhal Number
Expt.	7.25	0.138
Standard k-ε	7.1	0.141
2-Layer k-ε	7.1	0.141
RNG k-ε	7.6	0.132

Notes:

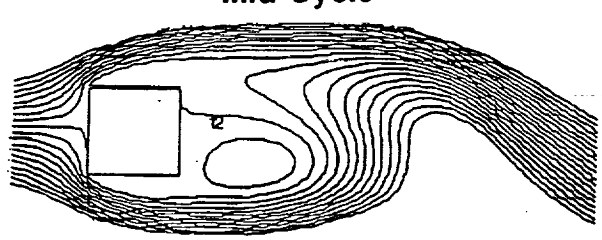
1. Experiments By Durao, Heitor, and Pereira (1988)
2. Computations with CFD-ACE
   Inlet: 78H Upstream; Outlet: 22H Downstream
   Grid: 120 x 80
   Time Steps: Over 70 Per Time Period

   Ref.: Avva, R.K., Singhal, A.K., Lai, Y.G., "Numerical Simulation Of Periodic and 3-Dimensional, Turbulent Flows With CFD-ACE," ASME Fluid Dynamics Conference, Lake Tahoe, NV, June 19-23, 1994.

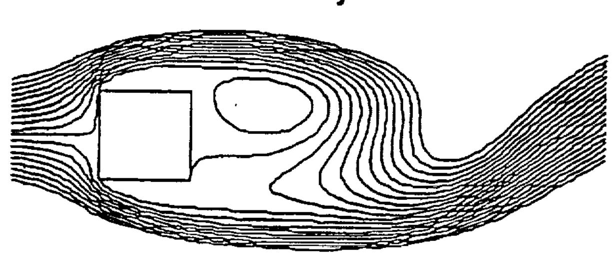
FLOW AROUND A SQUARE CYLINDER

Instantaneous Streamlines

Mid-Cycle

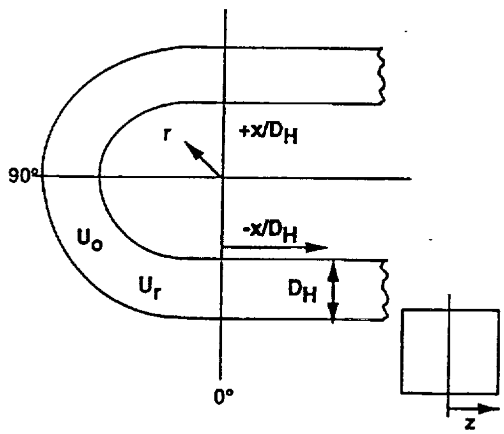


End of Cycle

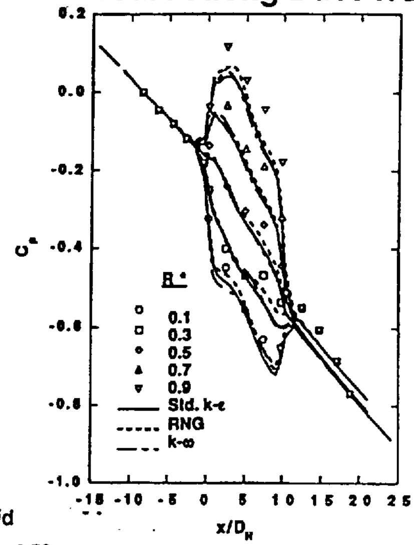


FLOW IN A 180° SQUARE DUCT

Computational Domain



Static Pressure Along Duct Walls

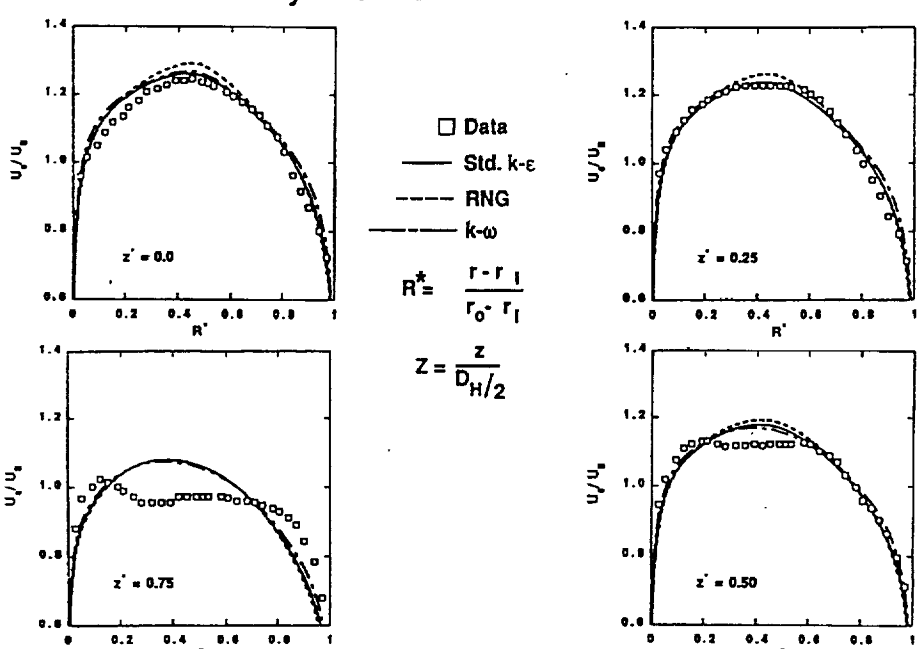


- Experiment by Chang, Humphrey and Modavi (1983)
- Computations Done with CFD-ACE on a 40x40x20 Grid

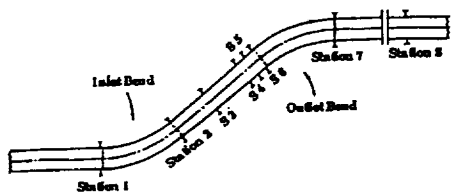
Ref.: Avva, R.K., Singhal, A.K., Lai, Y.G., "Numerical Simulation Of Periodic and 3-Dimensional, Turbulent Flows With CFD-ACE," ASME Fluid Dynamics Conference, Lake Tahoe, NV, June 19-23, 1994.

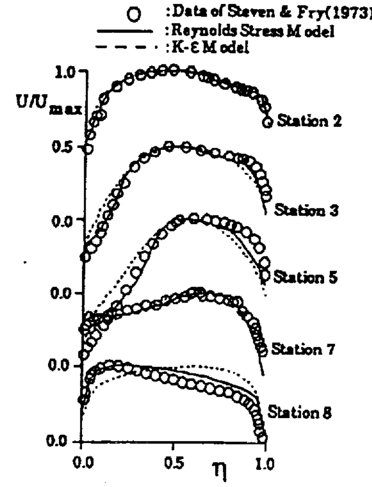
FLOW IN A 180^0 SQUARE DUCT

Mean Axial Velocity at θ = 3^0



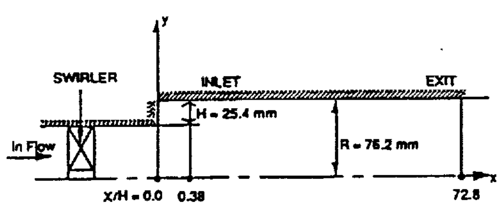
S-SHAPED ANNULAR DIFFUSER

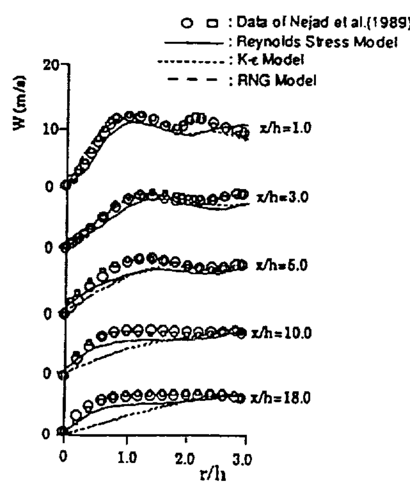




* k-ε Model and RNG Model Failed to Predict the Correct Location of the Maximum Velocity Downstream
* Computations with CFD-ACE; Publication Under Preparation

Confined Swirling Flow for a Dump Combustor

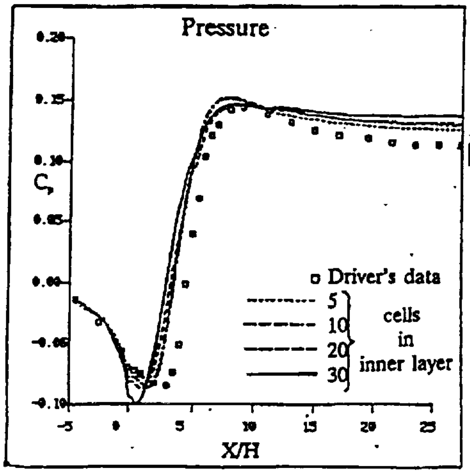


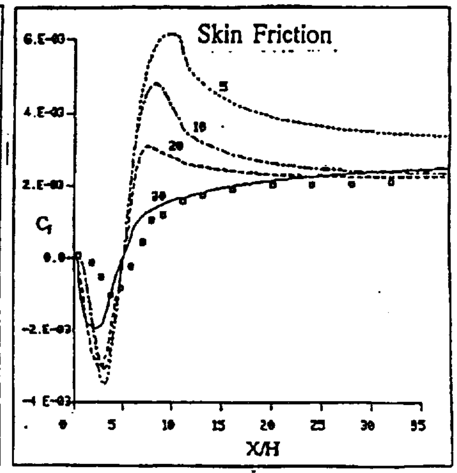


* K-ε model failed to preserve the vortex core strength near center (see x/h=10 & 18)
* Computational results to be presented at 1994 ASME Winter Annual Meeting (Chicago)

BACKWARD-FACING STEP

Sensitivitv to Grid Refinement





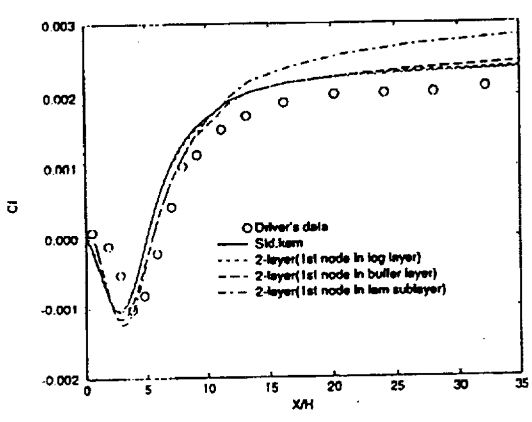
- **Low-Re Model Requires >30 Nodes in the Inter Layer**

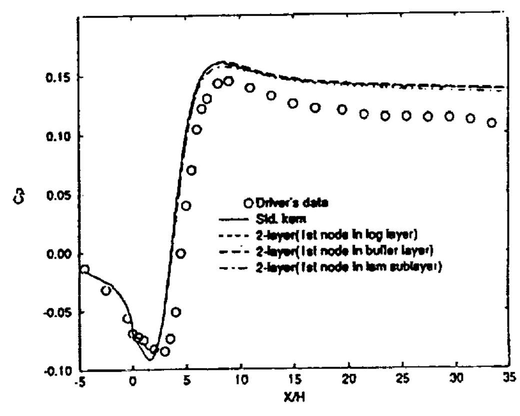
Ref. : "Comparative Study of High and Low Reynolds Number Versions of k-e Models," R.K. Avva, C.E. Smith, A.K. Singhal, AIAA-90-0246.

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 $k\sim\varepsilon$ Model, with Wall Functions
  - Wall Functions, Oh No!, Never!!
    Use Low-Re $k\sim\varepsilon$,..: 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 $k\sim\varepsilon^{++}$
    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?

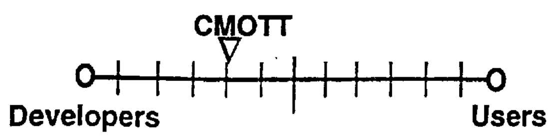
- Using What is Available, in Best Possible Ways
- Listening to Both Sides (Model Developers and Users)
- Trying to Resist Peer Pressures
- Struggling to Find Resources for Mundane Goals Such as Developing Guidelines for Correct Use of Turbulence Models

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 $k\sim\varepsilon$ 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



PROGRESS IN SIMULATING INDUSTRIAL FLOWS USING TWO-EQUATION MODELS: CAN MORE BE ACHIEVED WITH FURTHER RESEARCH?

N95-27895

Vahé Haroutunian
Fluid Dynamics International, Inc.
Evanston, Illinois

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-ε model.
- ► We have embarked on extensive study of TEM's over wide range of flows:
  - ▹ Identify true strengths and limitations of standard k-ε 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.

○ 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 $\rho\overline{u_i'u_j'}, \rho\overline{u_i'\vartheta'}, \rho\overline{u_i'c_\alpha'}$.
   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-ε MODEL**

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

where,

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

and,

$$c_\mu=0.09,\ c_1=1.44,\ c_2=1.92,\ \sigma_k=1.0,\ \sigma_\varepsilon=1.3$$

► **Remarks on standard k-ε model:**

▷ Use is made of Boussinesq's "isotropic" viscosity model.
▷ Fine scale isotropy is assumed in modeling ε 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.

LENGTH SCALE DETERMINING EQUATION

O THE EXTENDED k-ε MODEL OF CHEN AND KIM

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

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

$$c_\mu=0.09, c_1=1.15, c_2=1.9, c_3=0.25, \sigma_k=0.75, \sigma_\varepsilon=1.15$$

► Remarks on extended k- ε 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_s = 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.

LENGTH SCALE DETERMINING EQUATION

O THE RNG k-ε MODEL

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

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

where

$$R=\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\overline{\frac{\partial u_k'}{\partial x_i}\frac{\partial u_k'}{\partial x_j}}\approx\frac{c_\mu\eta^3(1-\eta/\eta_0)}{1+\beta\eta^3}\frac{\varepsilon^2}{k}$$

$$\eta\equiv s\frac{k}{\varepsilon};\quad s=\sqrt{G/\mu_t}$$

and

$$c_\mu=0.085, c_1=1.42, c_2=1.68, \sigma_k=\sigma_\varepsilon=0.7179, \eta_0=4.38, \beta=0.015$$

- 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. $c_\mu=0.0865, c_1=1.45, c_2=1.83, \sigma_k=0.8, \sigma_\varepsilon=1.15, \eta_0=4.618, \beta=0.17$
- 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.

LENGTH SCALE DETERMINING EQUATION

○ **Additional Remarks on RNG k-ε 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 ε eq'n of extended k-ε model of Chen and Kim.
- Latest model does not predict von Karman constant.

- **The most notable fact about the RNG k-ε model of YOTGS is that it challenges the notion of fine scale isotropy of turbulence**
  - Thus ε (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 η.
  - 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_{ij}
  - Anisotropic eddy-viscosity models can provide estimates of a_{ij} 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 $\rho\overline{u_i'u_j'}$, 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 <u>accurate</u> unbounded scheme in <u>mean flow equations</u>.
  - ▹ Use <u>inaccurate</u> numerically diffuse scheme in <u>turbulence equations</u>.
  - ▹ 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.

- ► <u>Accurate schemes must be used in both mean flow and turbulence eq's.</u>

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

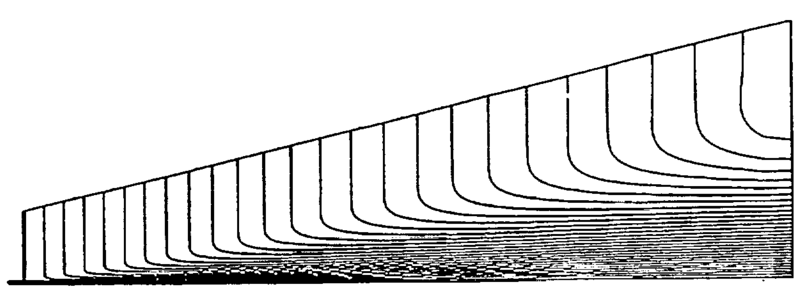
REMARKS:

- ► 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

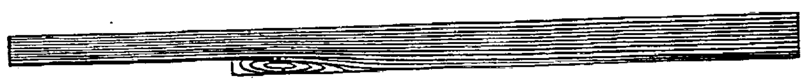
	Plane Jet		Round Jet	
	$d\delta/dx$	% error	$d\delta/dx$	% error
Experiment	≈0.105		≈0.095	
Standard k-ε model	0.104	−1	0.112	18
Extended k-ε model (original)	0.10	−5	0.10	5
Extended k-ε model (revised)	0.102	−3	0.104	9.5
RNG k-ε model (original)	0.131	25	0.157	65
RNG k-ε model (revised)	0.101	−4	0.113	19



TURBULENT FLOW OVER BACKWARD FACING STEP

Kim et al Test Case: Re = 45000

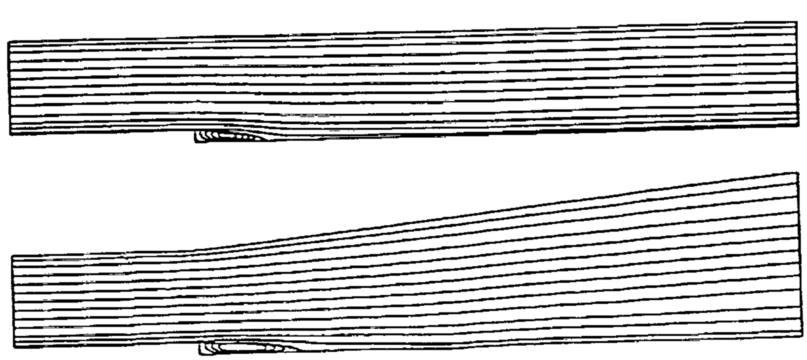
	X_R	% error
Experiment	7.0 ±0.5	
Standard k-ε model	6.5	–7.1
Extended k-ε model (original)	8.4	20.0
Extended k-ε model (revised)	7.1	1.4
RNG k-ε model (original)	7.5	7.1
RNG k-ε model (revised)	7.46	6.6



TURBULENT FLOW OVER STEP IN CHANNEL WITH DIFFUSER WALL

Driver and Seegmiller Test Case: Re = 36000

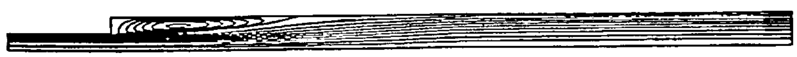
	Angle			
	0 degrees		6 degrees	
	X_R	% error	X_R	% error
Experiment	6.2		8.1	
Standard k-ε model	5.3	–14.5	6.6	–18.5
Extended k-ε model (original)	6.6	6.5	9.55	17.9
Extended k-ε model (revised)	5.76	–7.1	7.4	–8.6
RNG k-ε model (original)	6.17	–0.5	8.33	2.8
RNG k-ε model (revised)	6.11	–1.5	8.33	2.8



TURBULENT FLOW IN PIPE EXPANSION

Szczepura Test Case: Re = 890,000

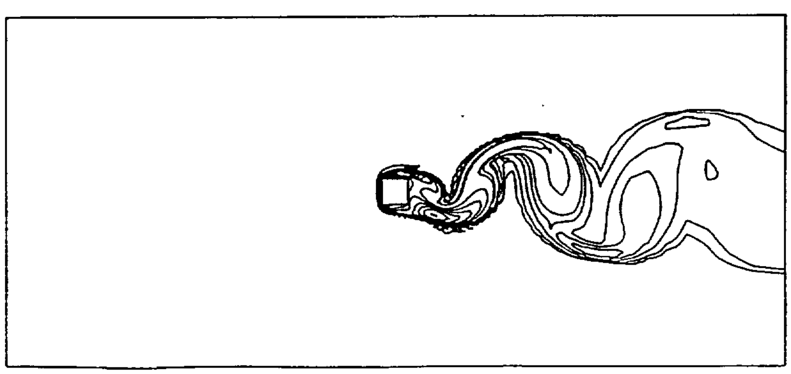
	X_R	% error
Experiment	9.51	
Standard k-ε model	9.59	0.9
Extended k-ε model (original)	12.44	30.8
Extended k-ε model (revised)	10.6	11.5
RNG k-ε model (original)	11.35	19.5
RNG k-ε model (revised)	11.39	20

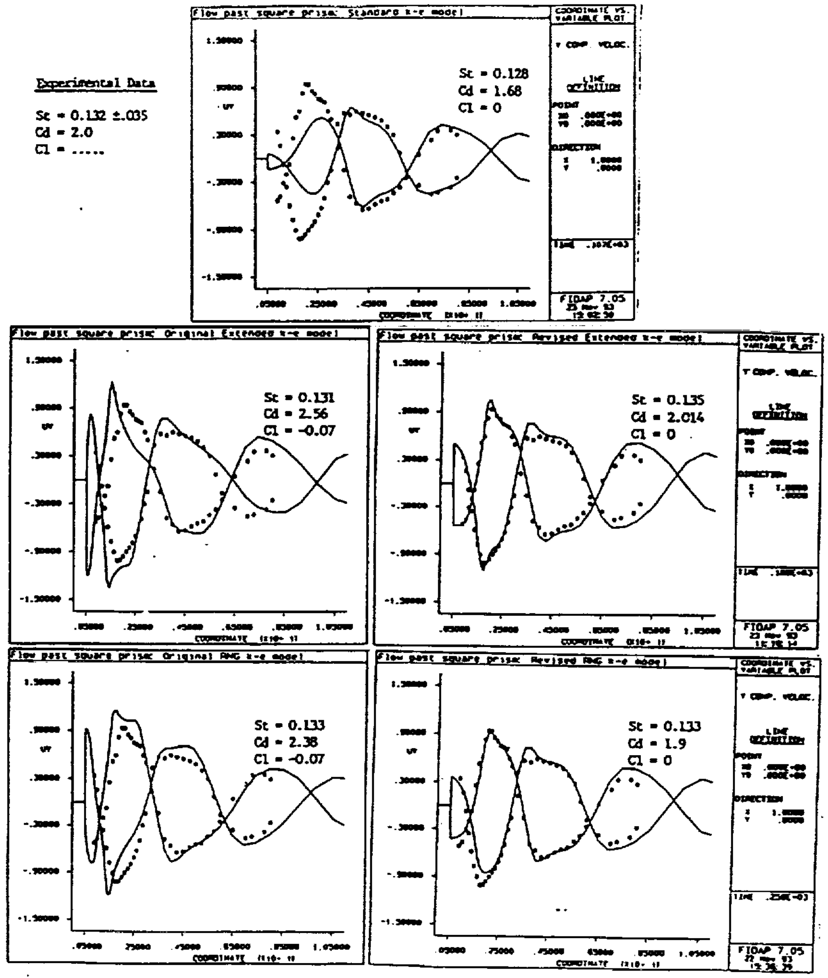


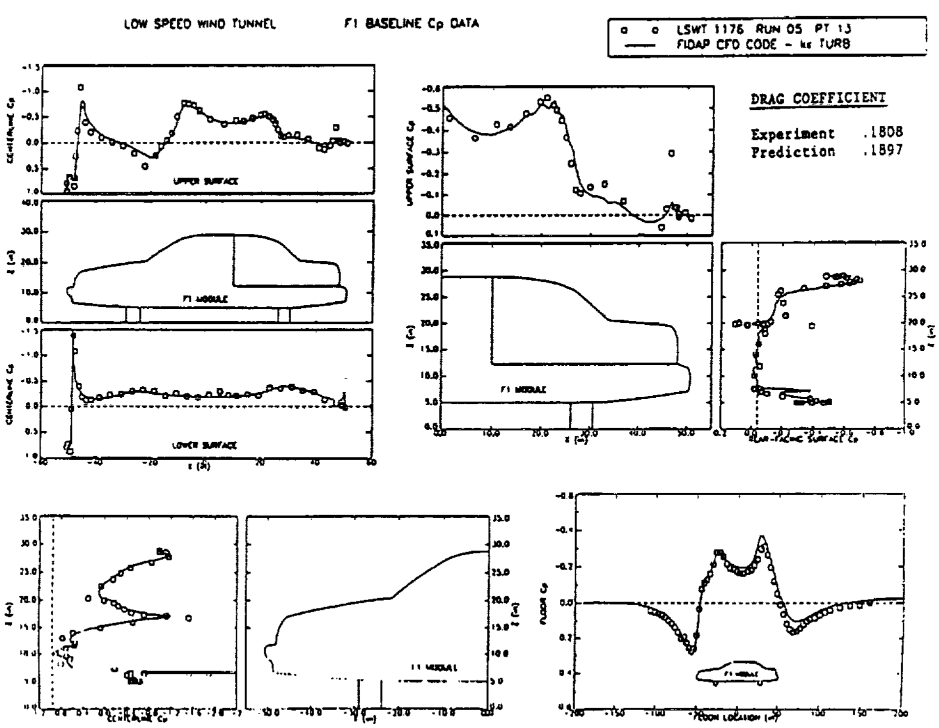
TURBULENT FLOW PAST SQUARE PRISM

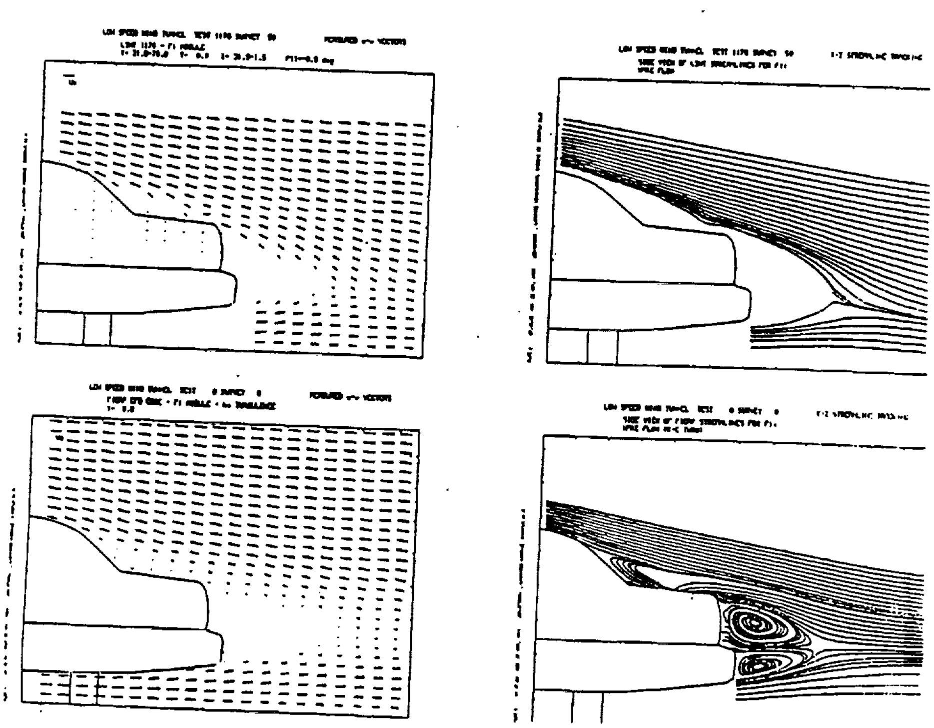
Lyn's Test Case: Re = 21400

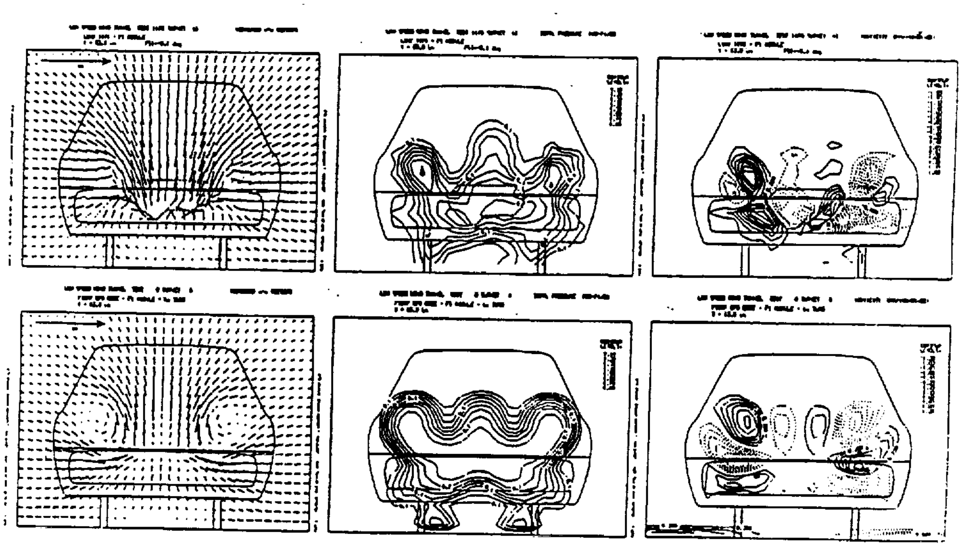
	Strouhal No.	C_d	C_l
Experiment	0.132 ± 0.035	≈ 2.0	N.A.
Standard k-ε model	0.128	1.68	0
Extended k-ε model (original)	0.131	2.56	–0.07
Extended k-ε model (revised)	0.135	2.014	0
RNG k-ε model (original)	0.133	2.38	–0.07
RNG k-ε model (revised)	0.133	1.9	0

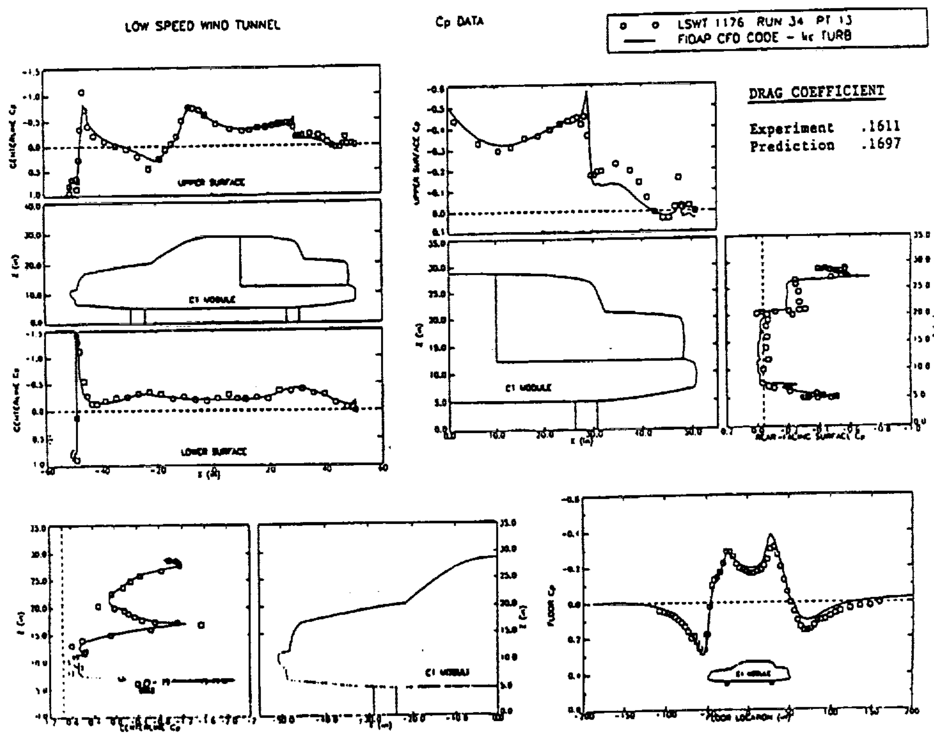


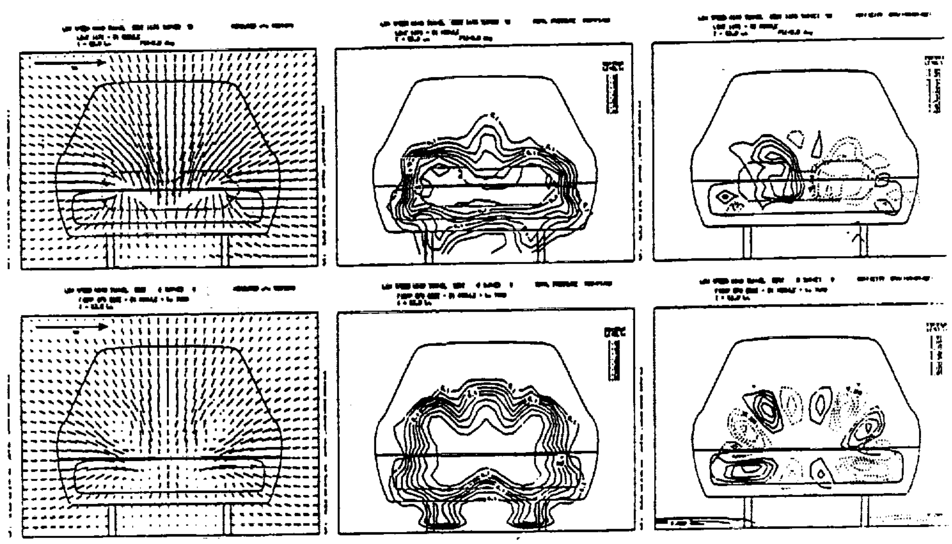

Experimental Data
St = 0.132 ±.035
Cd = 2.0
Cl = ...
Flow past square prism: Standard k-e model
St = 0.128
Cd = 1.68
Cl = 0
Flow past square prism: Original Extended k-e model
St = 0.131
Cd = 2.56
Cl = -0.07
Flow past square prism: Revised Extended k-e model
St = 0.135
Cd = 2.014
Cl = 0
Flow past square prism: Original RNG k-e model
St = 0.133
Cd = 2.38
Cl = -0.07
Flow past square prism: Revised RNG k-e model
St = 0.133
Cd = 1.9
Cl = 0
FIDAP 7.05

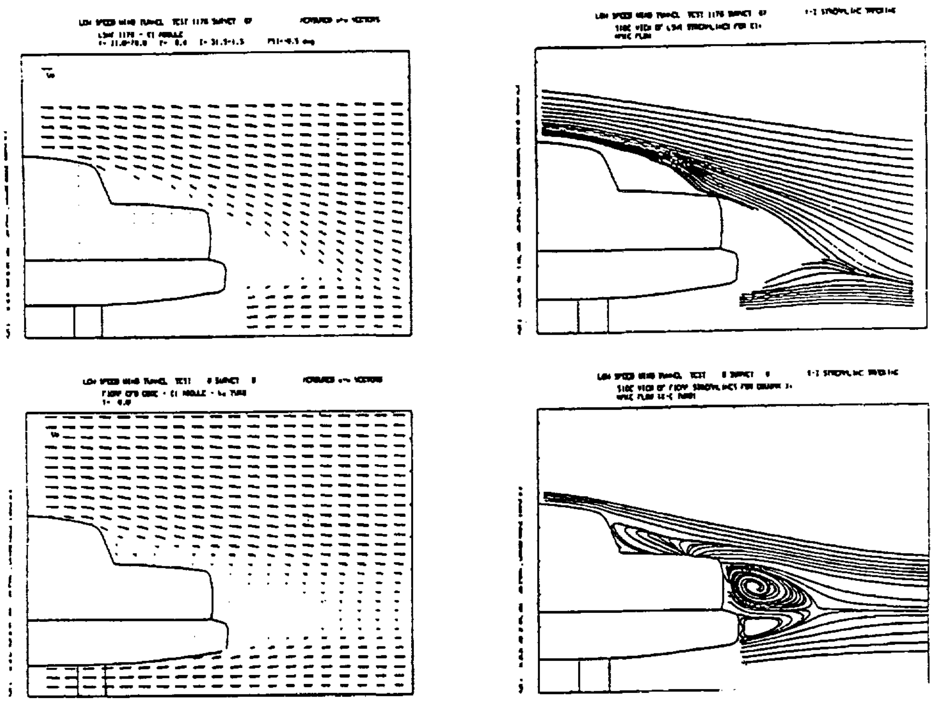

LOW SPEED WIND TUNNEL
F1 BASELINE Cp DATA
LSWT 1176 RUN 05 PT 13
FIDAP CFD CODE - ke TURB
DRAG COEFFICIENT
Experiment .1808
Prediction .1897
UPPER SURFACE
F1 MODULE
LOWER SURFACE

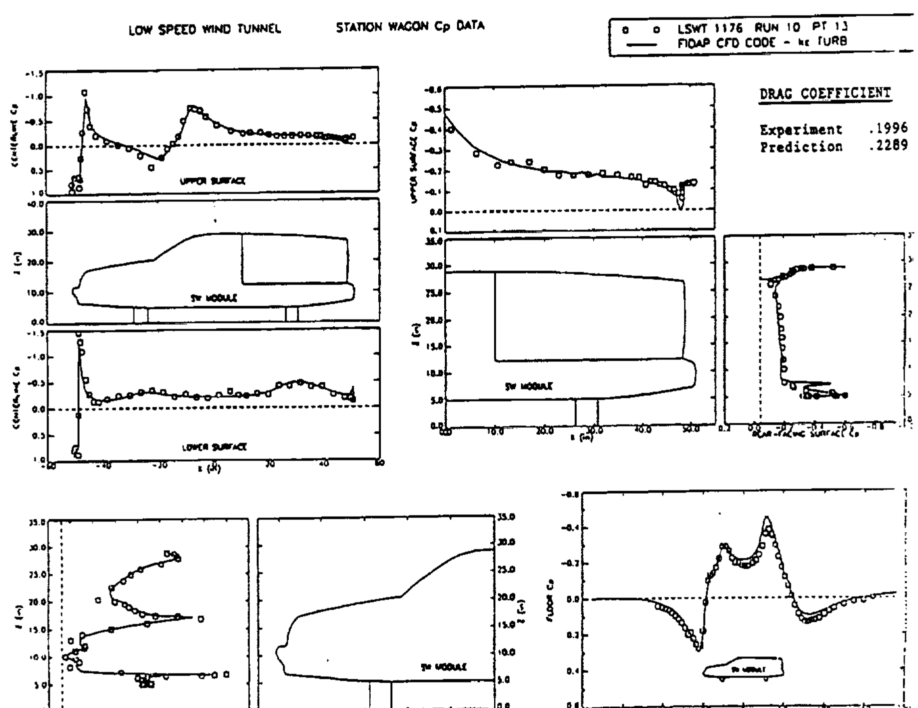


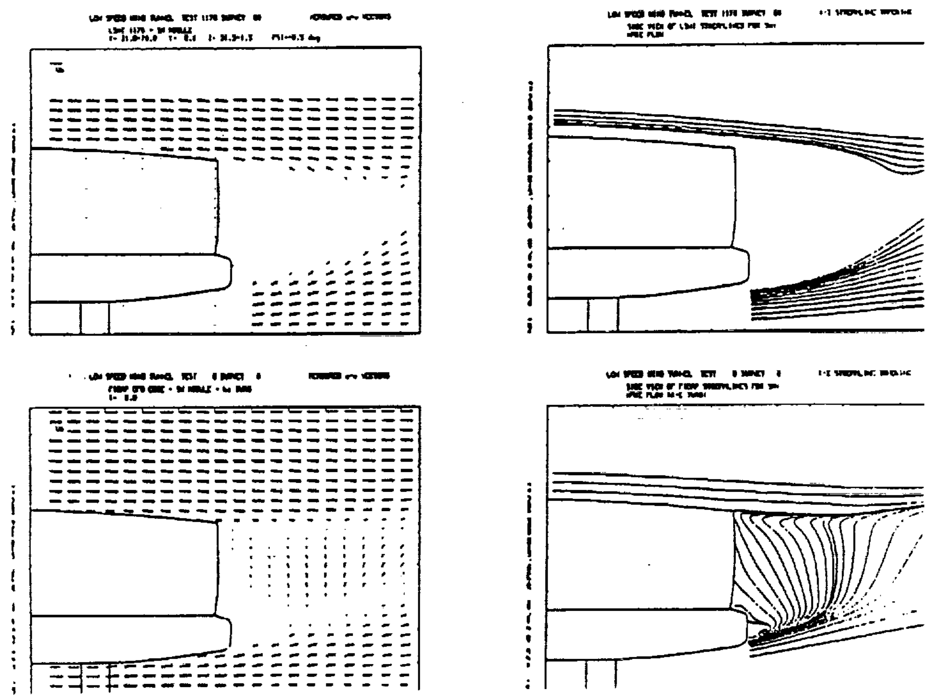


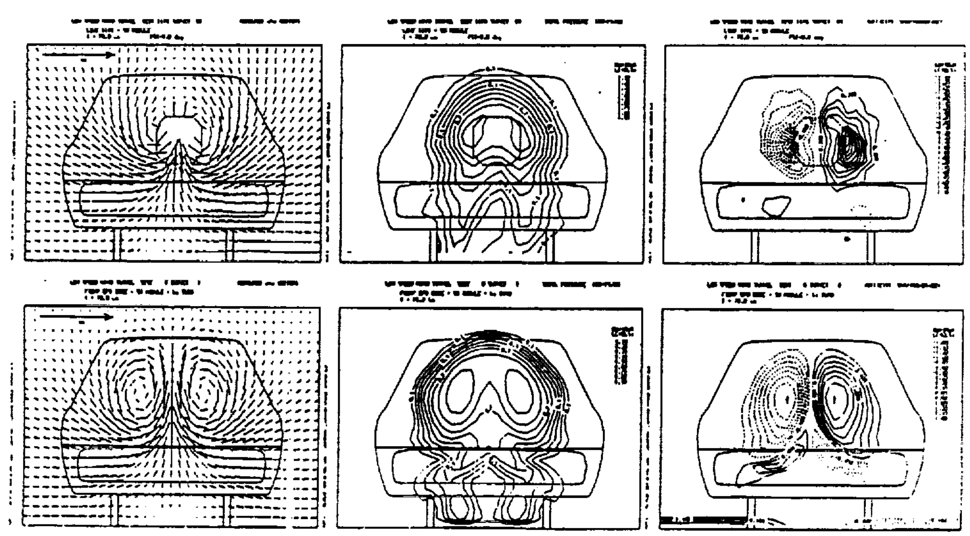

LOW SPEED WIND TUNNEL
Cp DATA
LSWT 1176 RUN 34 PT 13
FIDAP CFD CODE - kε TURB
DRAG COEFFICIENT
Experiment .1611
Prediction .1697
UPPER SURFACE
C1 MODULE
LOWER SURFACE
FLOOR LOCATION






LOW SPEED WIND TUNNEL
STATION WAGON Cp DATA
LSWT 1176 RUN 10 PT 13
FIDAP CFD CODE - kε TURB
DRAG COEFFICIENT
Experiment .1996
Prediction .2289
UPPER SURFACE
SW MODULE
LOWER SURFACE
SW MODULE
SW MODULE





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:
    - AEVM'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

N95- 27896

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

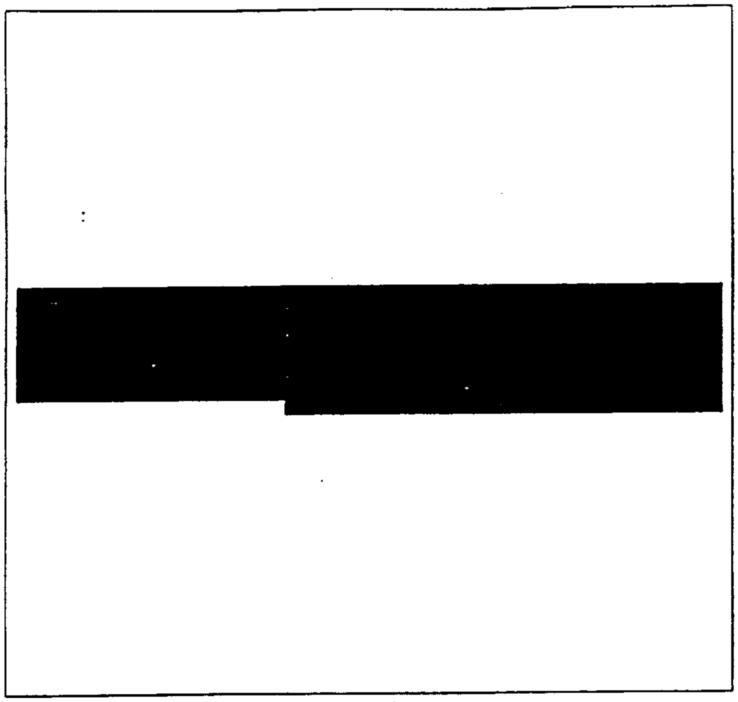
- BODY-FITTED NON-ORTHOGONAL COORDINATE SYSTEM
- UNSTRUCTURED COMPUTATIONAL MESH, DIFFERENT CELL TOPOLOGIES, IMBEDDED MESH REFINEMENT, DISCONTINUOUS MESH INTERFACE, MOVING BOUNDARY AND INTERNAL INTERFACES
- PRIMITIVE VARIABLE, SELF-ADAPTIVE ELLIPTIC-HYPERBOLIC PRESSURE CORRECTION METHOD
- COLLOCATED-VARIABLE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

STAR-CD: TURBULENCE MODELS

- TWO-EQUATION MODEL
  - STANDARD k-ε WITH CORRECTIONS FOR BULK DILATATION AND BUOYANCY
  - HIGH REYNOLDS NO. RNG BASED k-ε MODEL
- TWO-ZONE (TWO-LAYER) MODEL
  - HIGH REYNOLDS NO.: k-ε VARIANTS
  - LOW REYNOLDS NO.: k-ℓ VARIANTS, PRANDTL MIXING LENGTH MODEL

STAR-CD: TURBULENCE MODELS

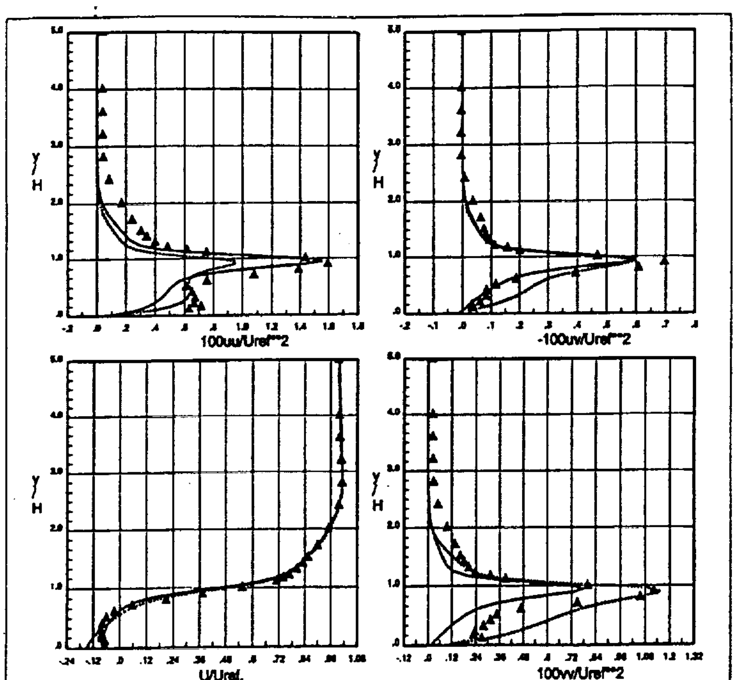
- REYNOLDS STRESS TRANSPORT MODEL
  - TRANSPORT EQUATIONS FOR CARTESIAN STRESS TENSOR IN NON-ORTHOGONAL COORDINATE SYSTEM, ON NON-STRUCTURED MESH
  - LAUNDER, RODI, REECE (1975) FORMULATION WITH LAUNDER (1989) MODEL CONSTANTS
  - GIBSON & LAUNDER (1978) WALL REFLECTION MODEL



Sep 94
VIEW
.000
.000
1.000
ANGLE
.000
DISTANCE
.340
CENTER
.076
.057
.003
EHIDDEN PLOT

Y
Z X

Driver & Seegmiller Backward Facing Step
Flow Domain = -20*H to 32*H
Mesh = 105 (Axial) x 45 (Radial)



GRAPH PLOT
FRAMES

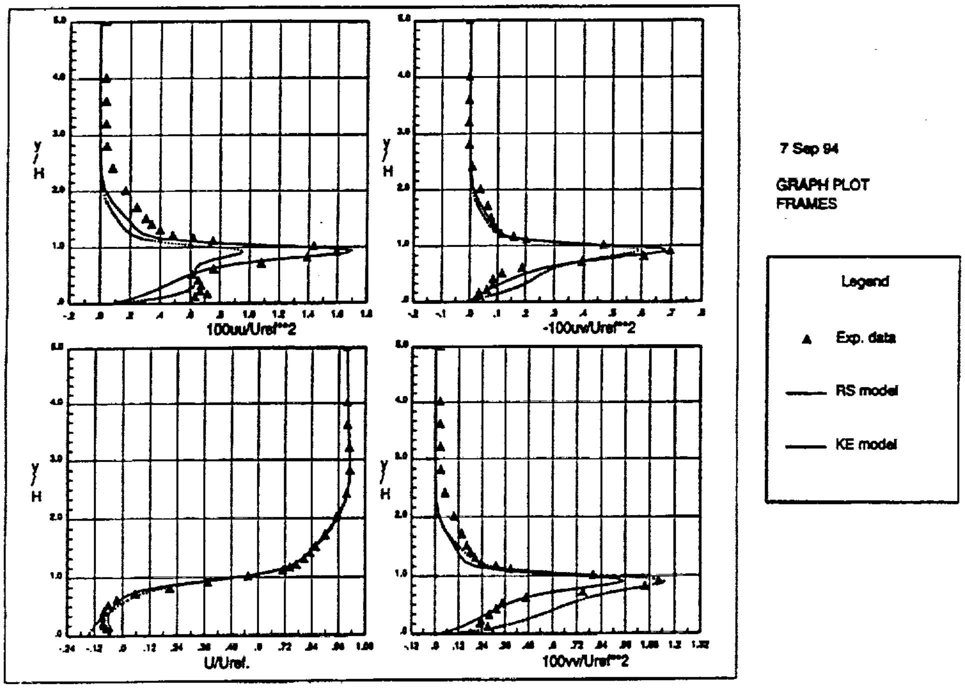
Legend

▲ Exp. data

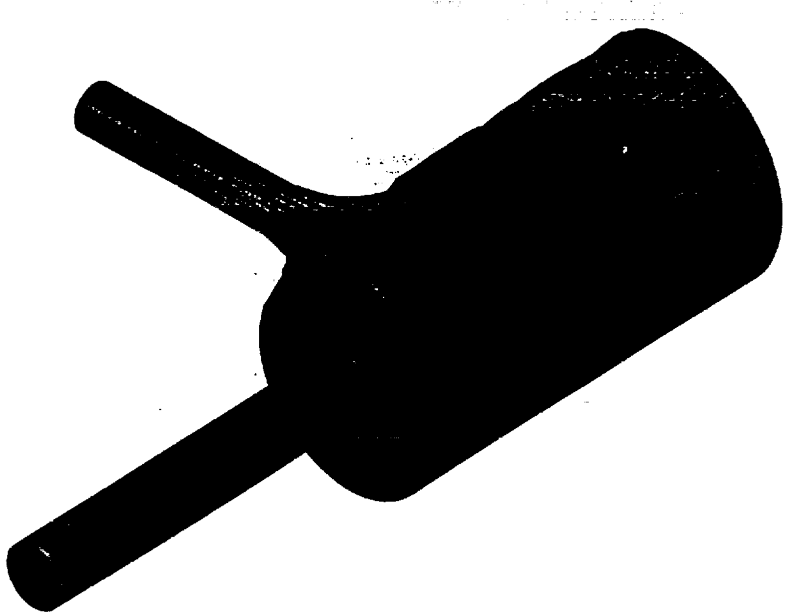
—— RS model

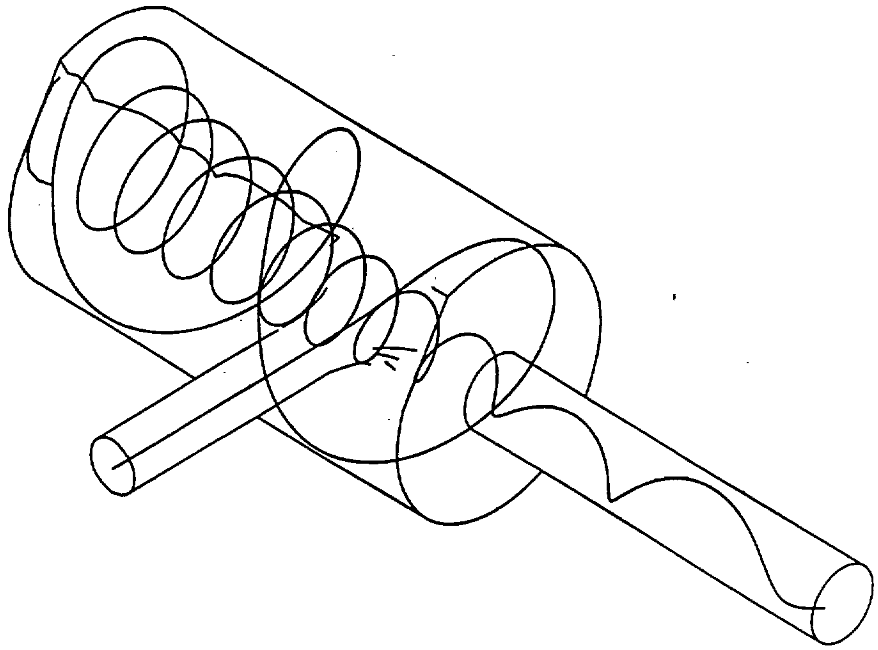
—— KE model

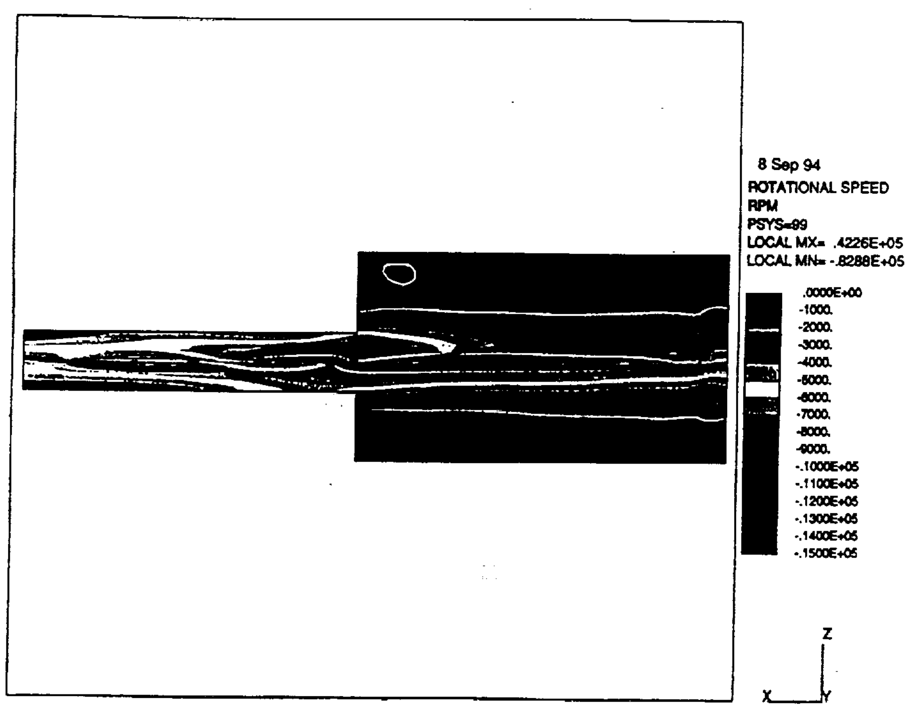
Driver & Seegmiller Backward Facing Step
Data Inlet B.C.
Location X/H = 1.5



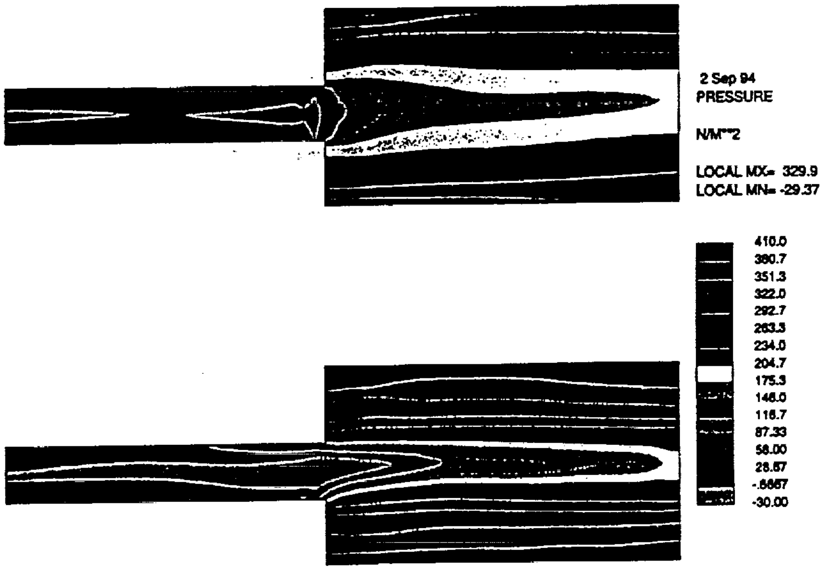
Driver & Seegmiller Backward Facing Step
Data Inlet B.C.; No Wall Damping Funct.
Location X/H = 1.5







8 Sep 94
ROTATIONAL SPEED
RPM
PSYS=99
LOCAL MX= .4226E+05
LOCAL MN= -.8288E+05
.0000E+00
-1000.
-2000.
-3000.
-4000.
-5000.
-6000.
-7000.
-8000.
-9000.
-.1000E+05
-.1100E+05
-.1200E+05
-.1300E+05
-.1400E+05
-.1500E+05
Z
X
Y



0.5 LUD DIFFERENCING
TURBULENCE MODEL KE - UPPER, RSM - LOWER

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(10^5 - 10^6) 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 (BENCH-MARK EXPERIMENTAL MODELS)
  - DESIGN/PERFORMANCE OPTIMIZATION OBJECTIVES
  - NUMERICAL AND TURBULENCE MODEL ACCURACY IMPORTANT
  - REQUIREMENTS
    - HEAT TRANSFER
    - LOW REYNOLDS NO. FLOW
    - BODY FORCE FIELDS

ORIGINAL PAGE IS
OF POOR QUALITY

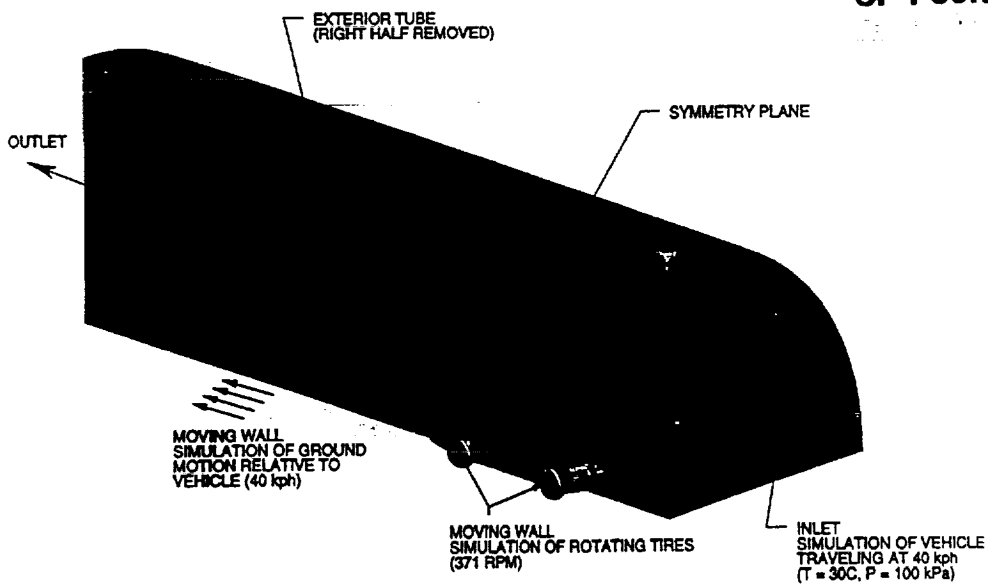
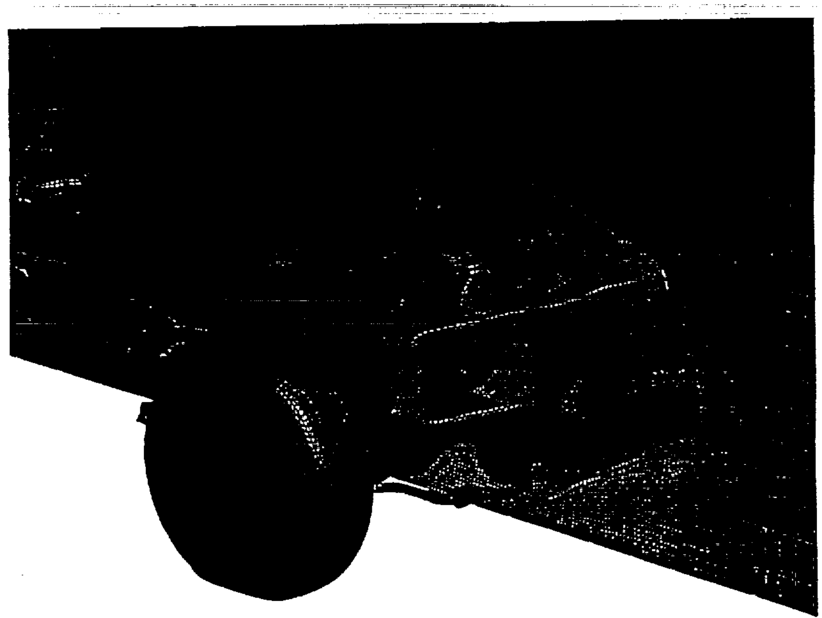
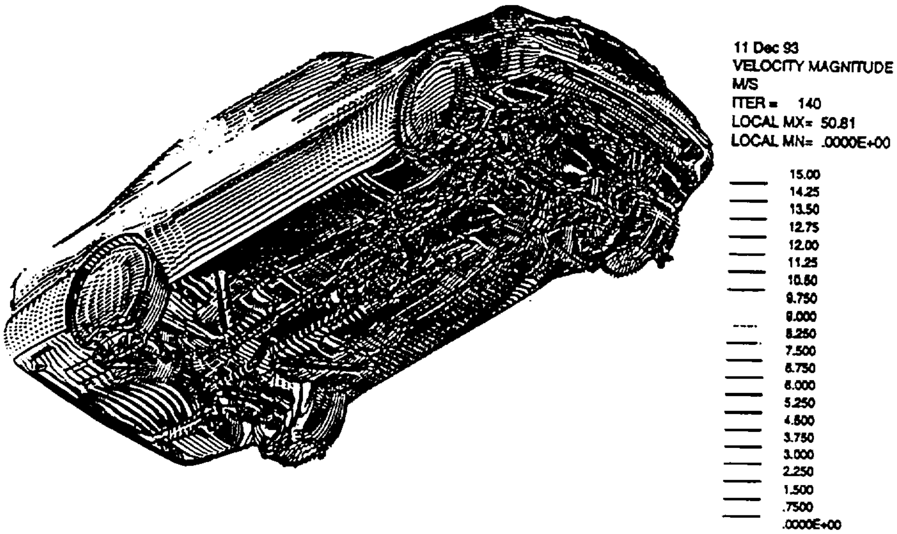


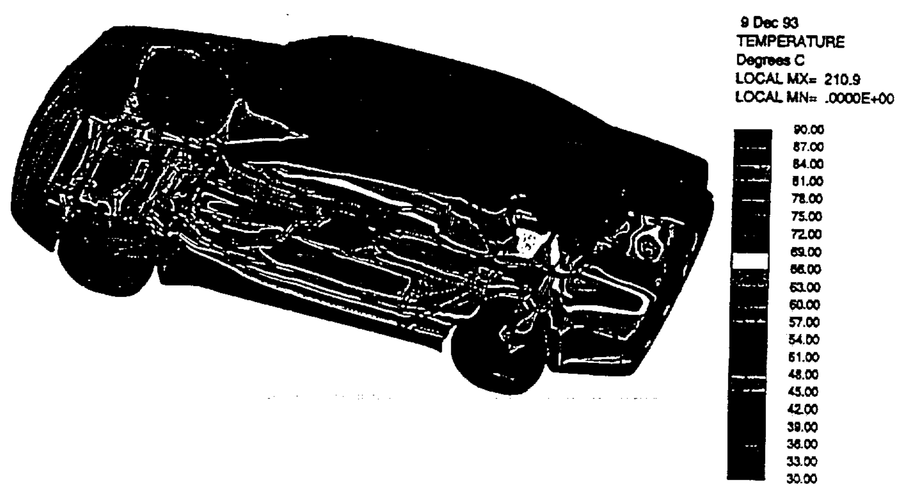
FIGURE 1: EXTERIOR BOUNDARY CONDITIONS FOR W202 40 kph ANALYSIS



ORIGINAL PAGE IS
OF POOR QUALITY



W202 UNDERHOOD FLOW ANALYSIS
CASE 3: 40 kph SIMULATION
Velocity near the surface of the vehicle.



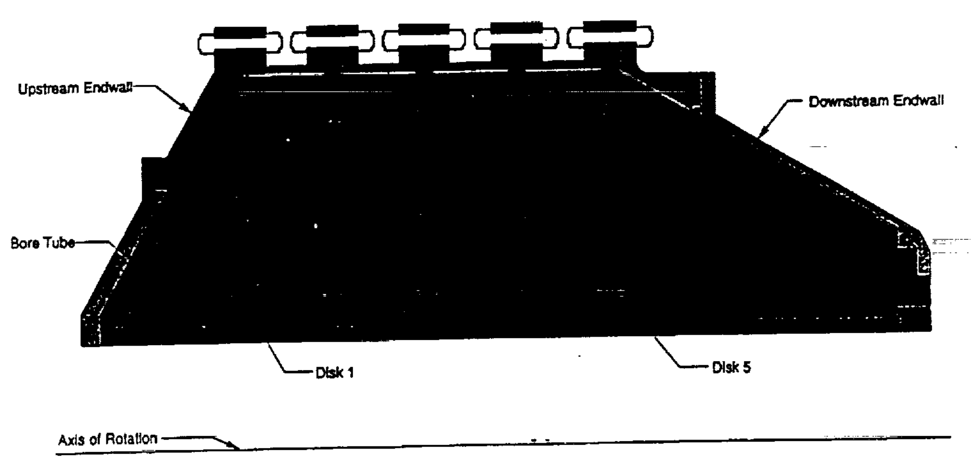
W202 UNDERHOOD FLOW ANALYSIS
CASE 3: 40 kph SIMULATION

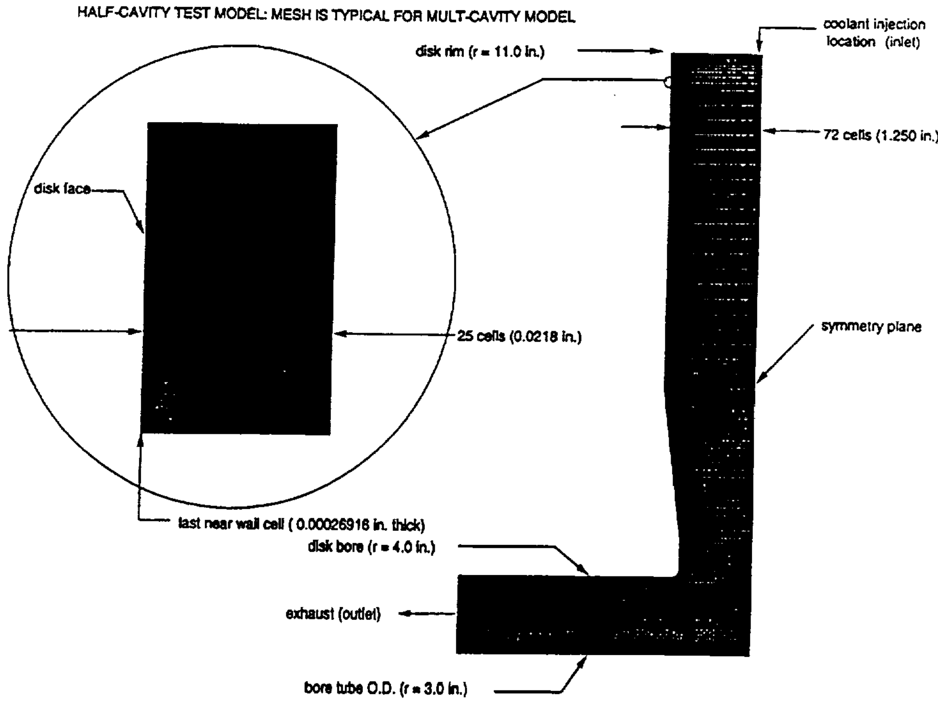
APPLICATIONS & EXPERIENCES

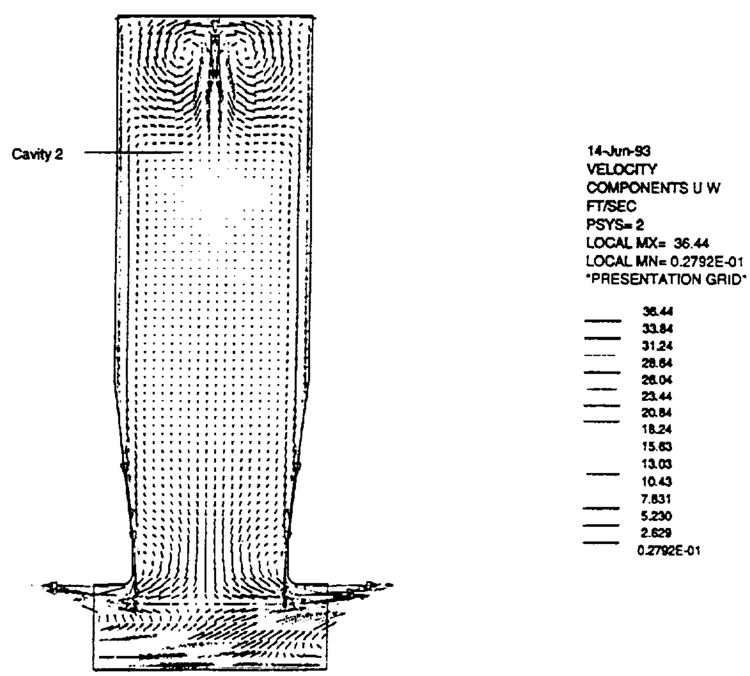
APPLICATION (DATA)	FLOW COMPLEXITY	TURBULENCE MODEL	FINDINGS	T.M. NEEDS
ROTATING DISC CAVITY[1]	• FORCE FIELD • WALL EFFECT	• k-ε • 2 LAYER k-ℓ	• EKMAN LAYER RESOLVED • FAIR PRESSURE DROP • EXCESSIVE E.V.	• RSTM + SUITABLE 2 LAYER • LOW Re RSTM
DIFFUSION HOLE FILM COOLING[2]	• JET-CROSS FLOW • WALL ANISOTROPY	• k-ε • RNG, k-ε • 2 LAYER k-ℓ	• JET SEPARATION SENSITIVE TO MESH TOPOLOGY/ RESOLUTION • POOR SPANWISE SPREAD	• RSTM + SUITABLE 2 LAYER • LOW Re RSTM

[1] GRABER et al (1987)
[2] GOLDSTEIN et al (1968), LIGRANI et al (1992)

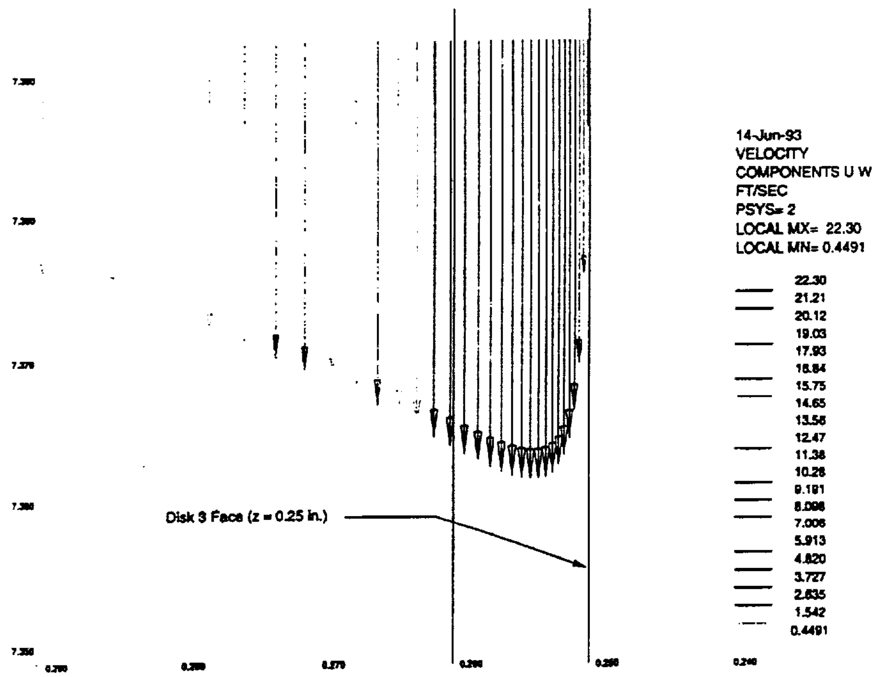
COMPRESSOR DRUM TEST RIG STAR-CD CONJUGATE HEAT TRANSFER MODEL



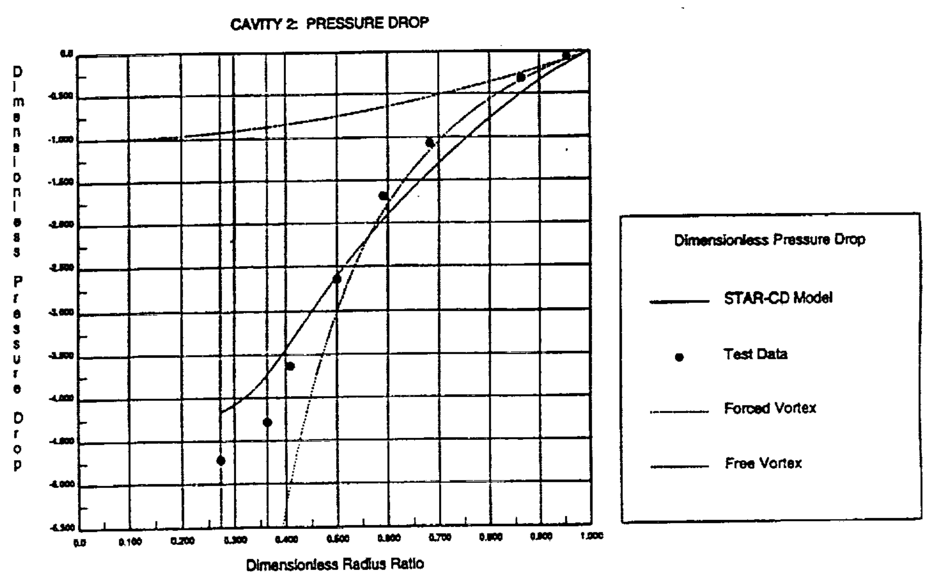


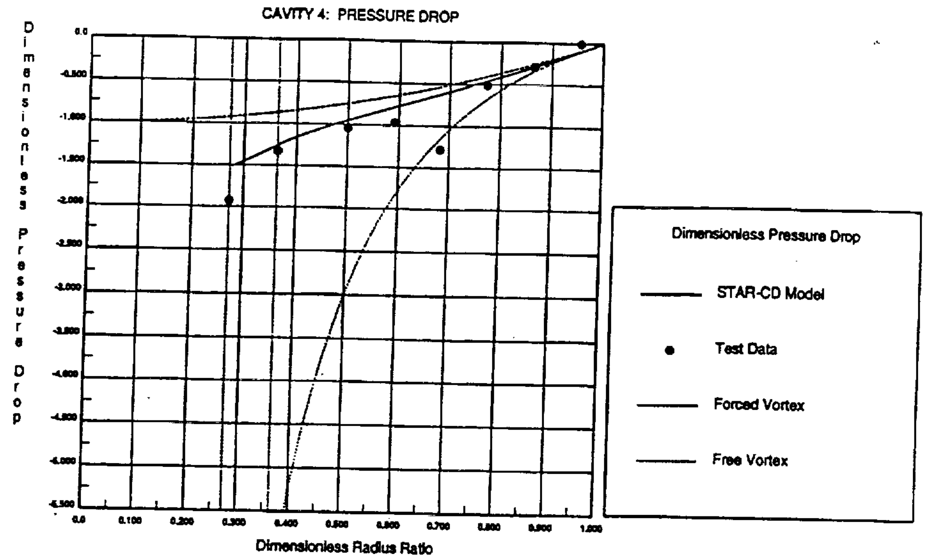


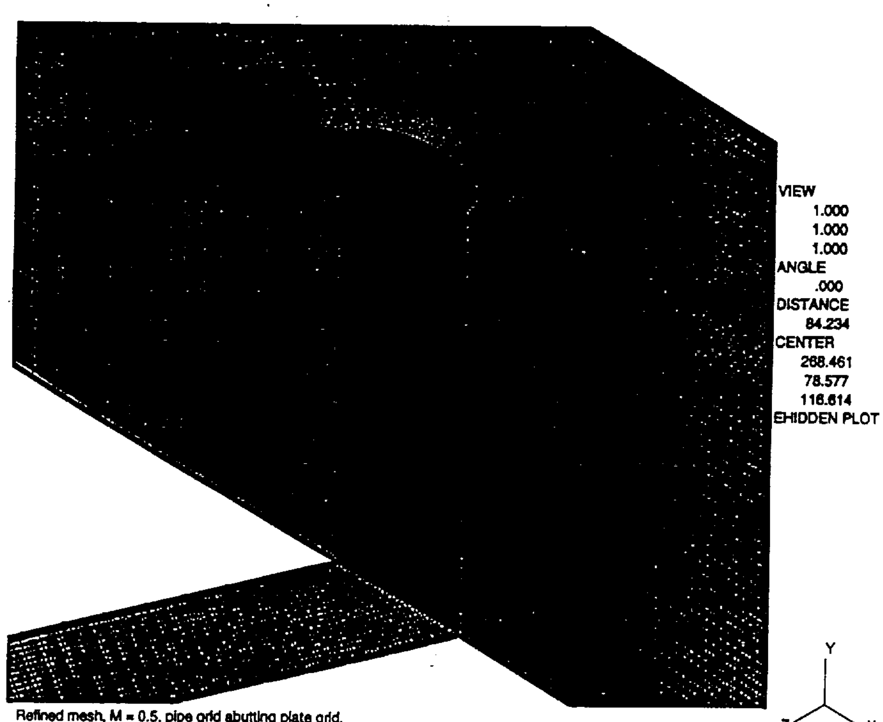
Compressor Drum Test Rig Cold Flow Benchmark Analysis
Secondary Flow



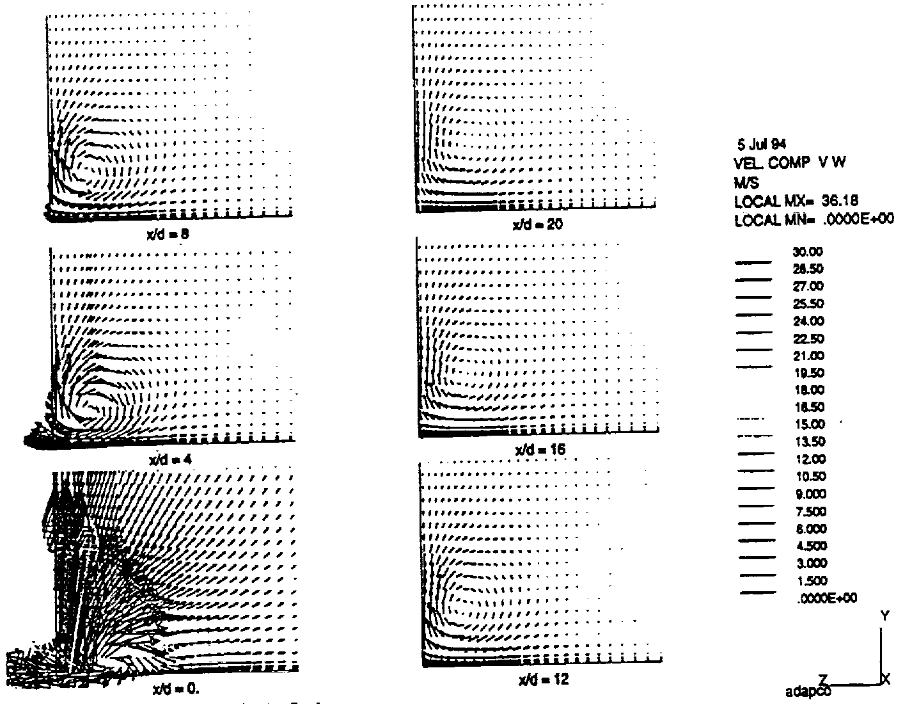
Compressor Drum Test Rig Cold Flow Benchmark Analysis
Secondary Flow in Cavity 2
Velocity Vectors at r = 7.40 inches



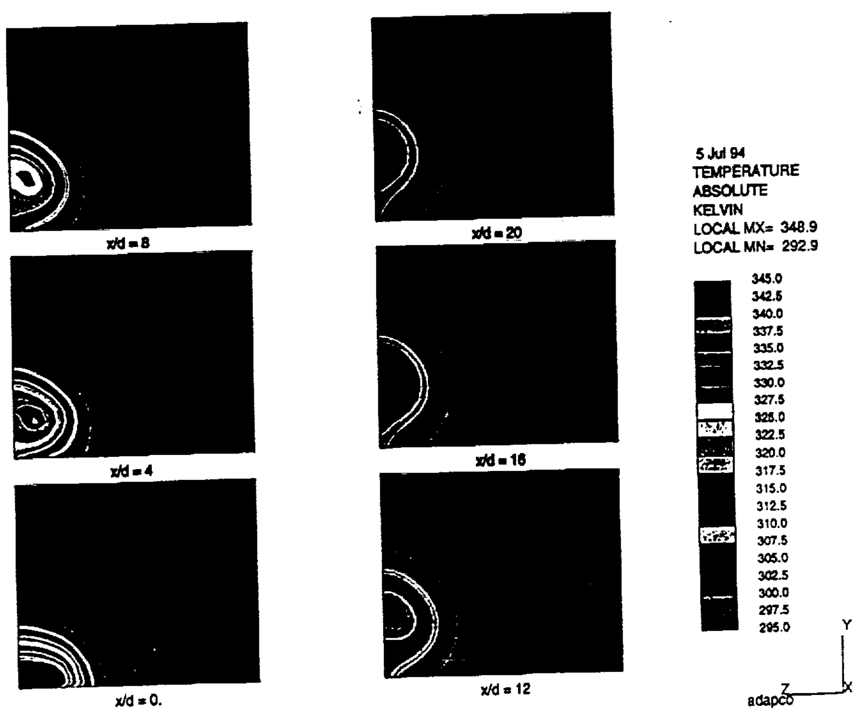




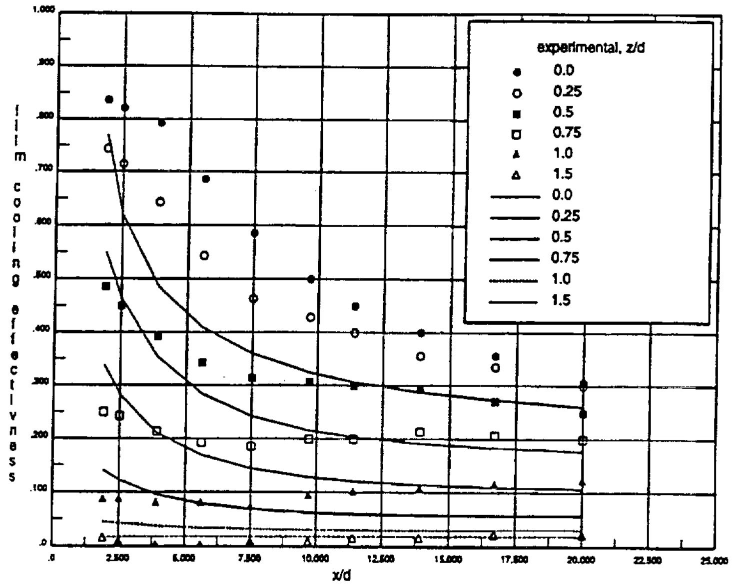
Refined mesh, M = 0.5, pipe grid abutting plate grid.
Mesh = 330000 fluid cells
Two-Layer mesh



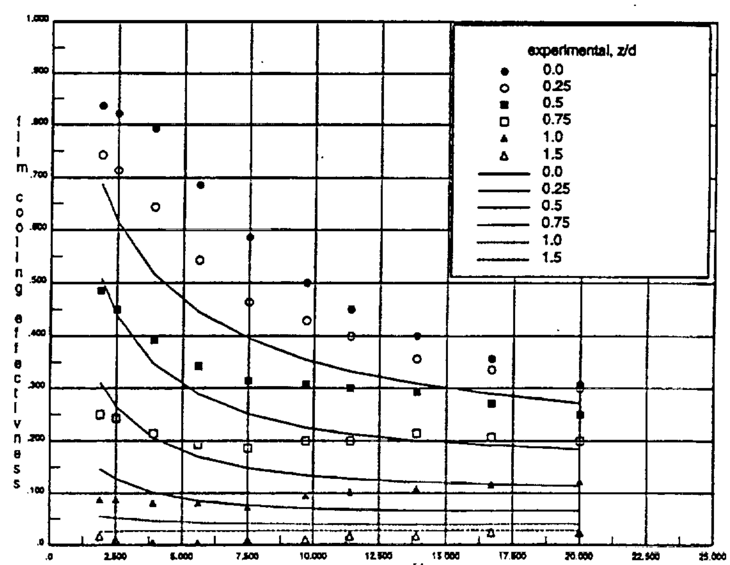
CFD Dicrete Hole Film Cooling Verification Study
Simulation of experiment of Goldstein, et. al. [1968] ; Blowing ratio M =0.5
Velocity vectors on spanwise planes ; 2-layer model.



CFD Dicrete Hole Film Cooling Verification Study
Simulation of experiment of Goldstein, et. al. [1968] ; 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 : Mesh II.

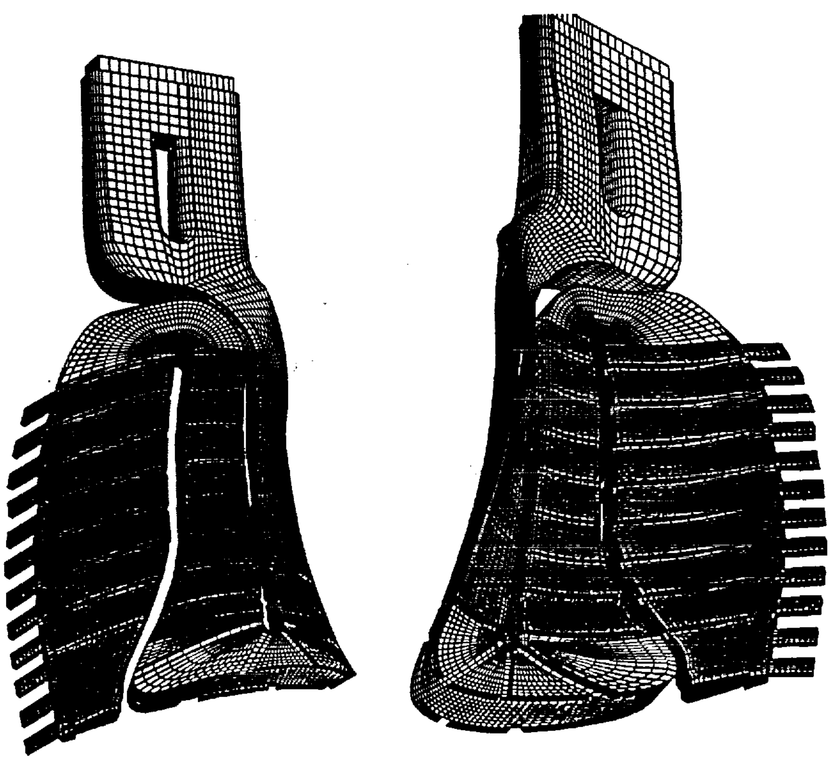


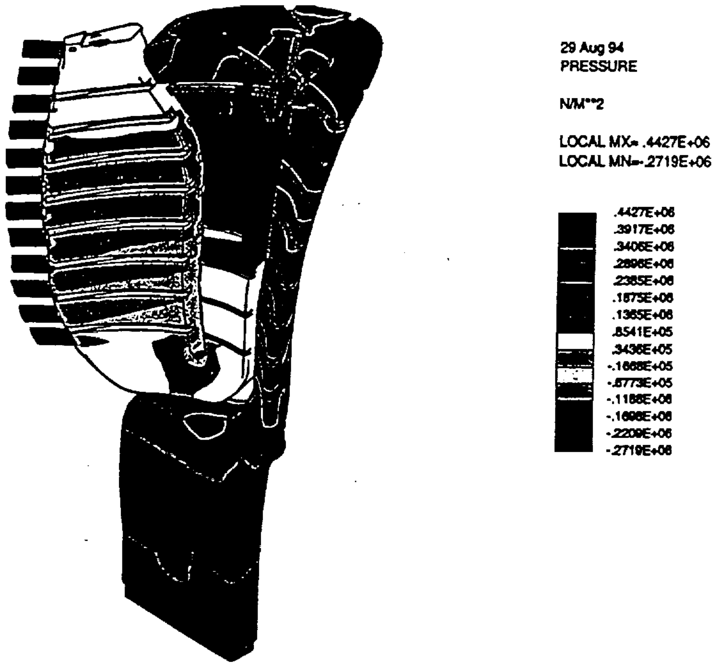
EXPERIMENTS OF GOLDSTEIN ET AL.,1968
COMPARISON OF FILM COOLING EFFECTIVENESS
M = 0.5 : 2 Layer mesh.

APPLICATIONS & EXPERIENCES (cont'd)

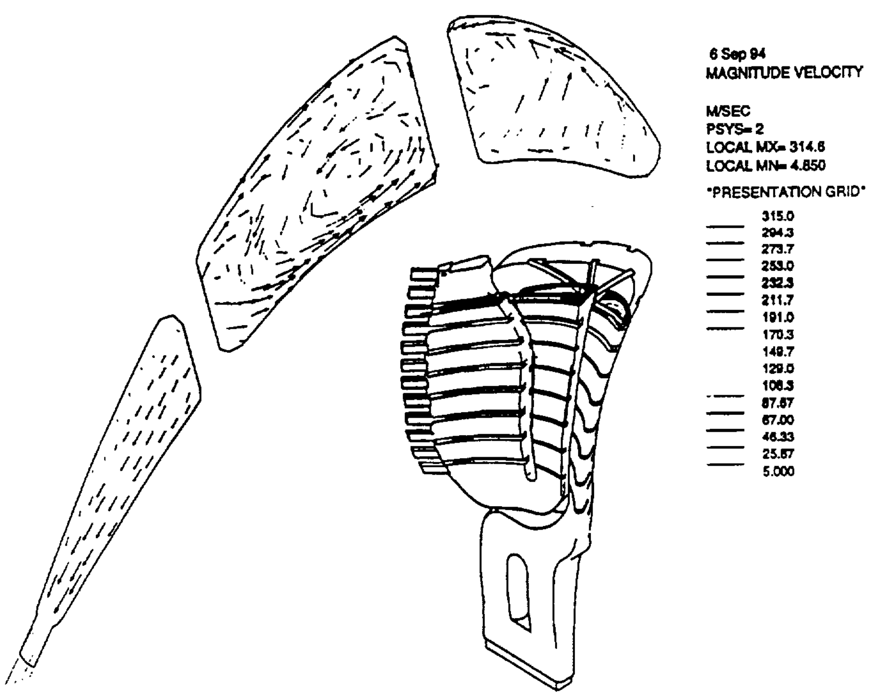
APPLICATION (DATA)	FLOW COMPLEXITY	TURBULENCE MODEL	FINDINGS	T.M. NEEDS
INTERNAL BLADE COOLING[3]	• FORCE FIELD • B.L. DISRUPTION	• k-ε	• DEPENDENCE ON MESH RESOLUTION • GOOD ΔP, h	
EXTERNAL CAR AERO-DYNAMICS[4]	• B.L. STRUCTURE INTERACTION • COMPLEX WAKE	• k-ε • RNG k-ε • 2 LAYER k-ℓ	• DEPENDENCE ON MESH RESOLUTION • GOOD C_D • POOR LIFT	• RSTM • LOW Re RSTM

[3]GE AIRCRAFT ENGINES [ABUAF & KERCHER (1991)]
[4]10 FORD 1/4 SCALE MODELS IN WIND TUNNEL TEST [WILLIAMS et al (1994)]

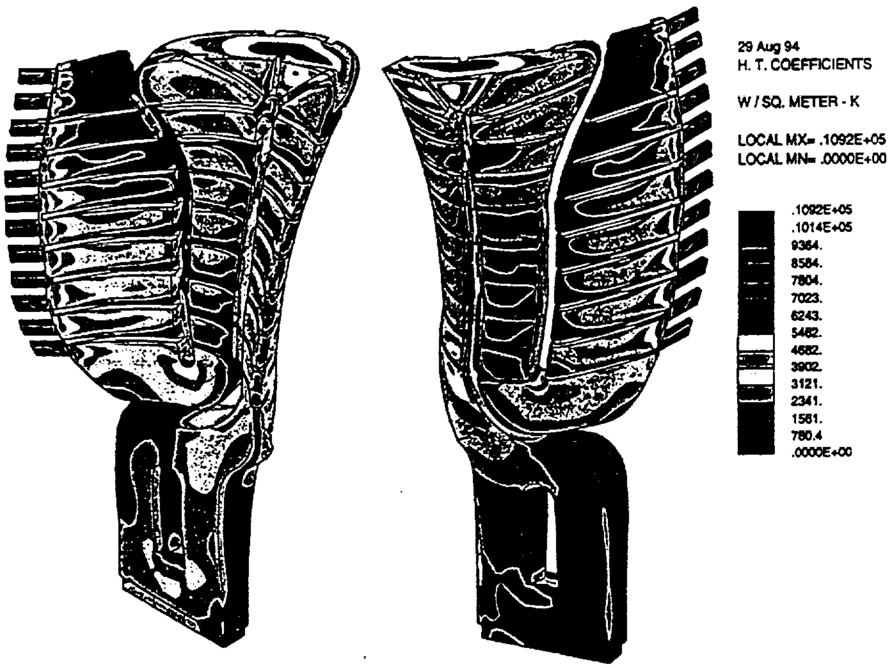


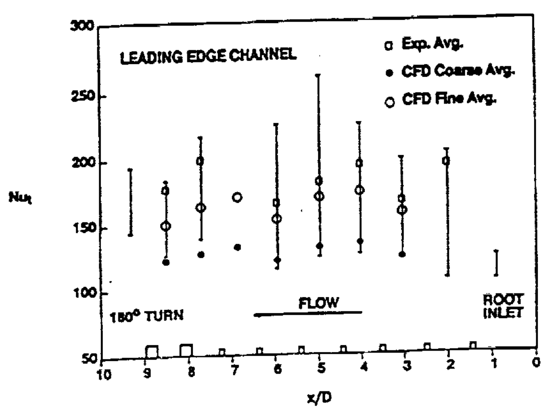


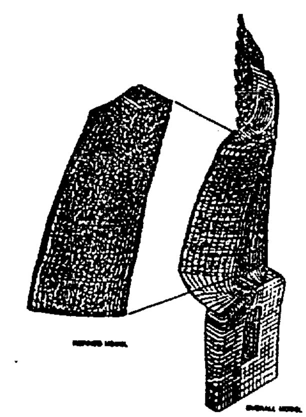
29 Aug 94
PRESSURE
N/M**2
LOCAL MX= .4427E+06
LOCAL MN=-.2719E+06
.4427E+06
.3917E+06
.3406E+06
.2896E+06
.2385E+06
.1875E+06
.1365E+06
.8541E+05
.3436E+05
-.1668E+05
-.6773E+05
-.1188E+06
-.1698E+06
-.2209E+06
-.2719E+06



6 Sep 94
MAGNITUDE VELOCITY
M/SEC
PSYS= 2
LOCAL MX= 314.6
LOCAL MN= 4.850
PRESENTATION GRID
315.0
294.3
273.7
253.0
232.3
211.7
191.0
170.3
149.7
129.0
108.3
87.67
67.00
46.33
25.67
5.000

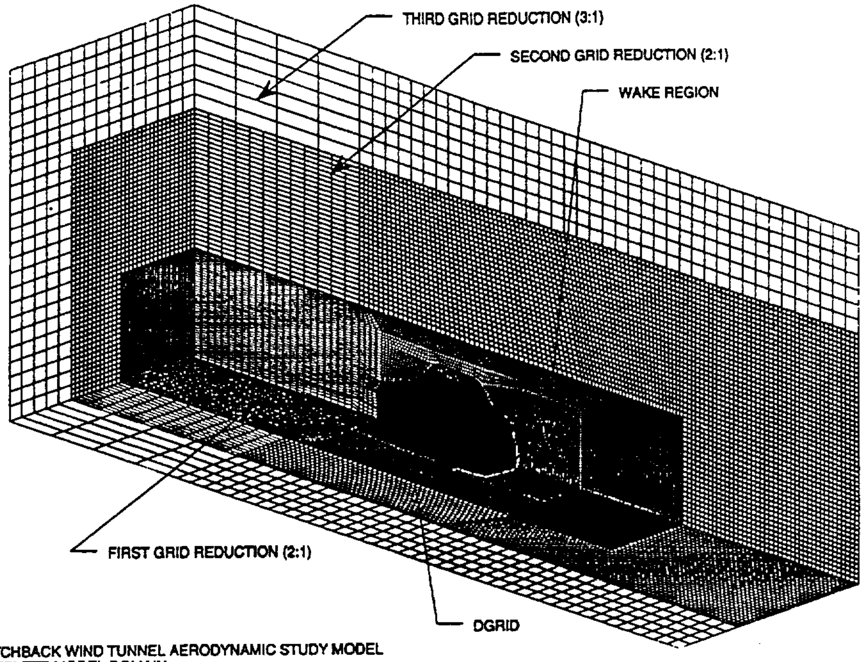




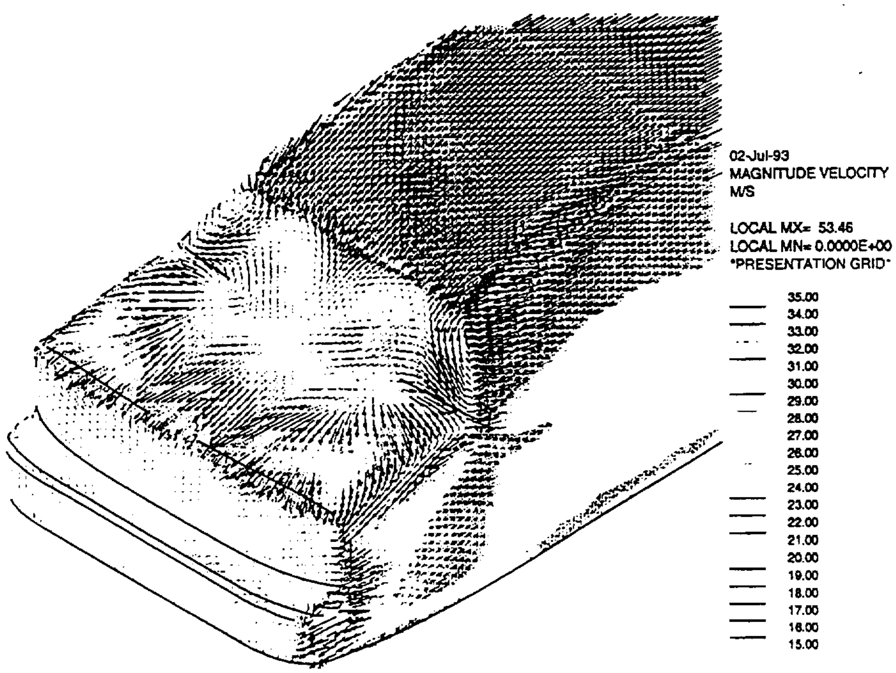


CFD BLADE AND LEADING EDGE MODELS
Marinaccio (1989,1990a,1991)

Figure 4 a Leading edge channel heat transfer distribution with distance from the inlet. Comparison of model turbulated convex surface maximum, minimum and average measurements with blade CFD average predictions.



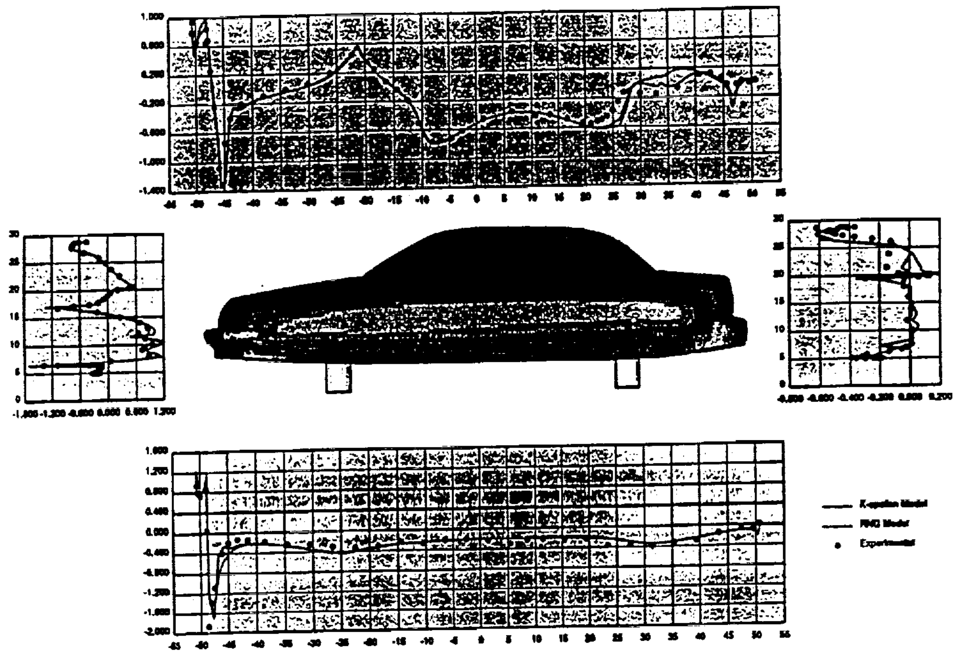
NOTCHBACK WIND TUNNEL AERODYNAMIC STUDY MODEL
COMPLETE MODEL DOMAIN



WIND TUNNEL AERODYNAMICS STUDY OF NOTCHBACK TEST SHAPE
KE RESULTS - KE TURBULENCE MODEL WITH LUD
VELOCITY MAGNITUDE NEAR THE VEHICLE

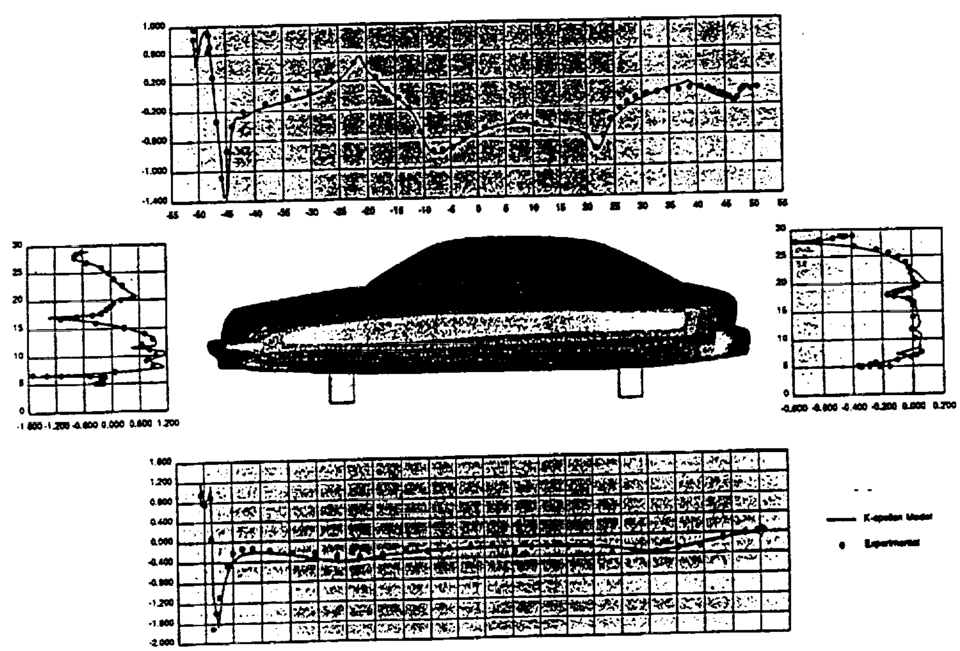
VIEW FROM REAR

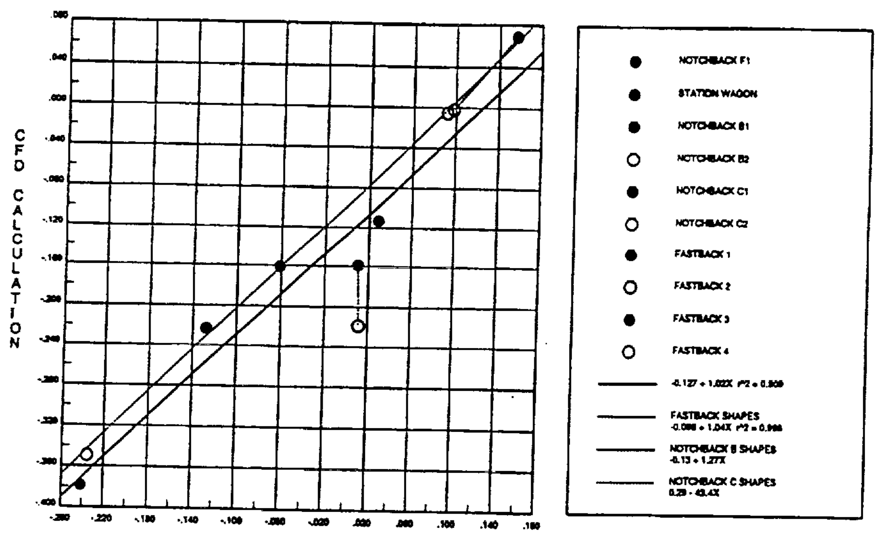
PRESSURE COEFFICIENTS AT THE CENTERLINE OF VEHICLE



NOTCHBACK 1 BODY STYLE

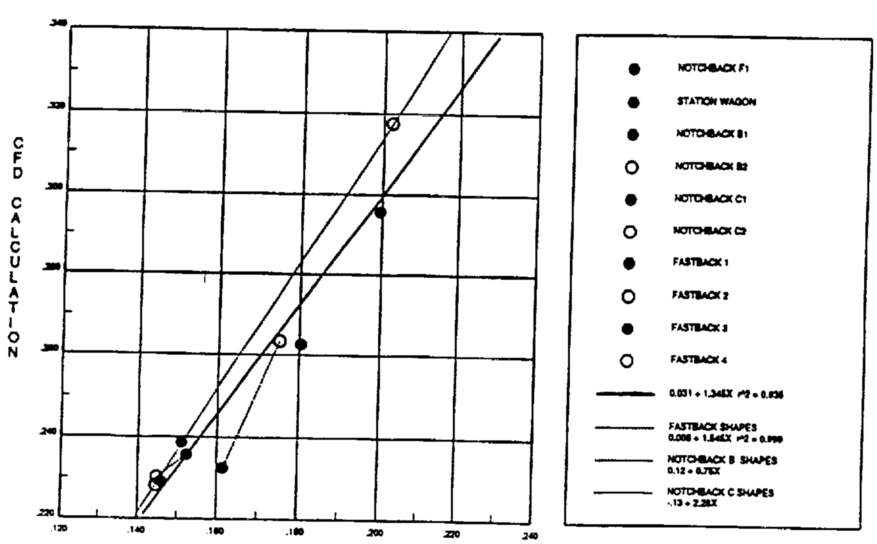
PRESSURE COEFFICIENTS AT THE CENTERLINE OF VEHICLE





EXPERIMENT RESULTS

COMPARISON OF EXPERIMENTAL AND COMPUTIONAL LIFT COEFFICIENTS
K EPSILON TURBULENCE MODEL - *** INITIAL RESULTS ***



EXPERIMENTAL RESULTS

COMPARISON OF EXPERIMENTAL AND COMPUTIONAL 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-ϵ TO INCORPORATE FORCE-FIELD EFFECTS
- BENCHMARKING
  - A RELIABLE DATABASE OF BENCHMARK SET OF REPRESENTATIVE COMPLEX FLOWS
  - BENCHMARK PERFORMANCE CLASSIFICATION OF VARIOUS EVM's (k-ϵ, k-ω, RNG AND NON-LINEAR k-ϵ, 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

TURBULENCE REQUIREMENTS OF A COMMERCIAL CFD CODE

N95- 27897

J.P. Van Doormaal, C.M. Mueller, and M.J. Raw
Advanced Scientific Computing Ltd.
Waterloo, Ontario, Canada

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

Application Profile cont'd

- 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, bouyancy, 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 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

How Can CMOTT Help? cont'd

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

How Can CMOTT Help? cont'd

Process data

- collect
- distil
- review
- interpret
- describe
- compile

How Can CMOTT Help? cont'd

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

N95- 27898

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

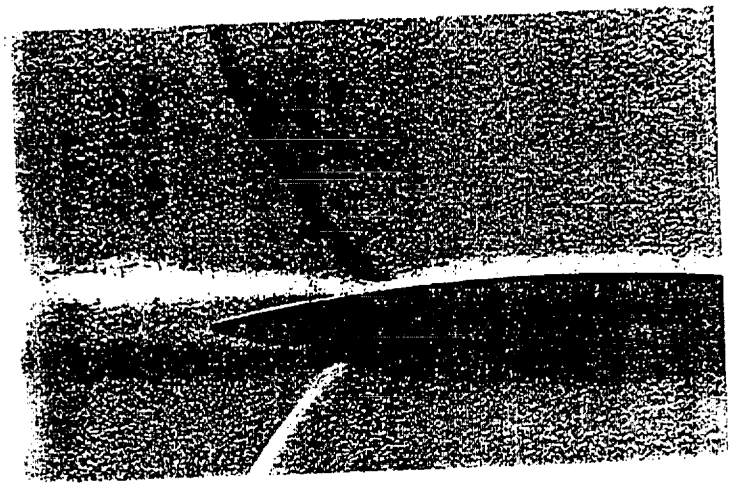
FUTURE WORK

I. PROJECT DESCRIPTION

1. Flows of Interest

2. Motivation

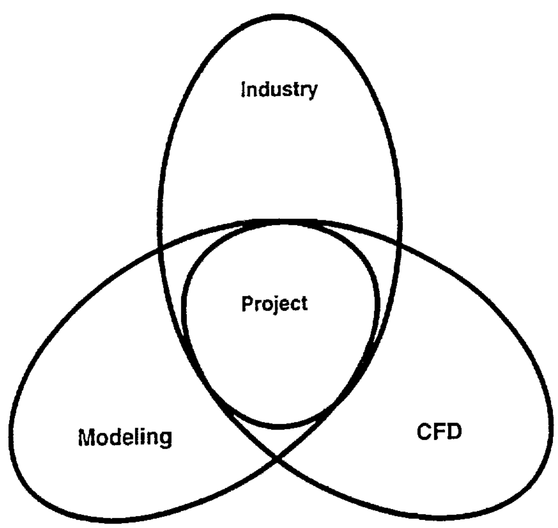
3. Method



Schlieren photograph of a shock-wave turbulent boundary-layer interaction

M=0.90 Re=1,750,000 [Liepmann]

I.2.MOTIVATION

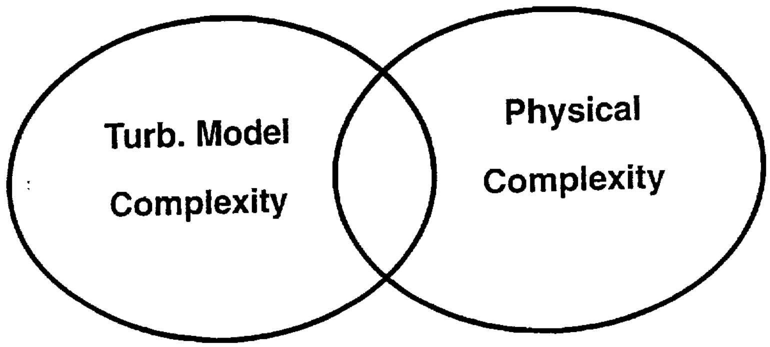


- Physics
  - Boundary Layer Separation & Wall Heat Transfer
  - Spreading rate
- Modeling
  Account for Compressibility Effects on Turbulence
- Numerics
  Compare 1-point Closures on Identical Solver

I.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)

$$-<\tau_{ij}u_{i,j}>=\Pi_d-\varepsilon_d-\varepsilon_s$$

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

- $\Pi d=<pd>$

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

- dilatation dissipation:

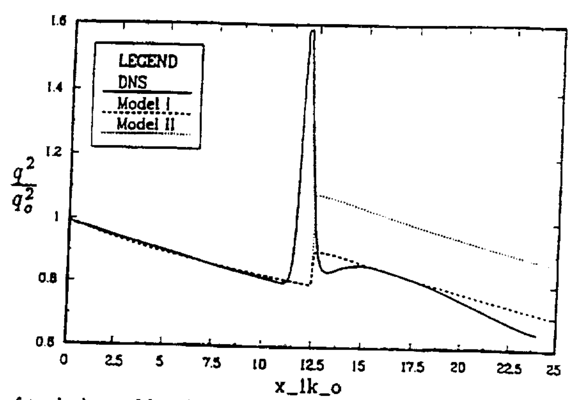
$$\varepsilon_d=(\mu_B+\frac{4}{3}\mu)<d^2>$$

  - Sarkar et al. (asymptotic analysis)
  - Zeman
    (Shocklet model)

- <u>pressure-dilatation correlation:</u>

$$\Pi_d = < pd >$$

  - 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 (Délery 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

<table>
<tr>
<td>

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

</td>
<td>

Current Code: cyste (D.Caughey)

1. Geometry
O- H-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

</td>
<td>

Future

1. Geometry
3D

2. Turbulence Models
SOC

</td>
</tr>
</table>

III.2. Turbulence Models:

Incompressible / Compressible: an additive approach

- Baldwin-Lomax

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

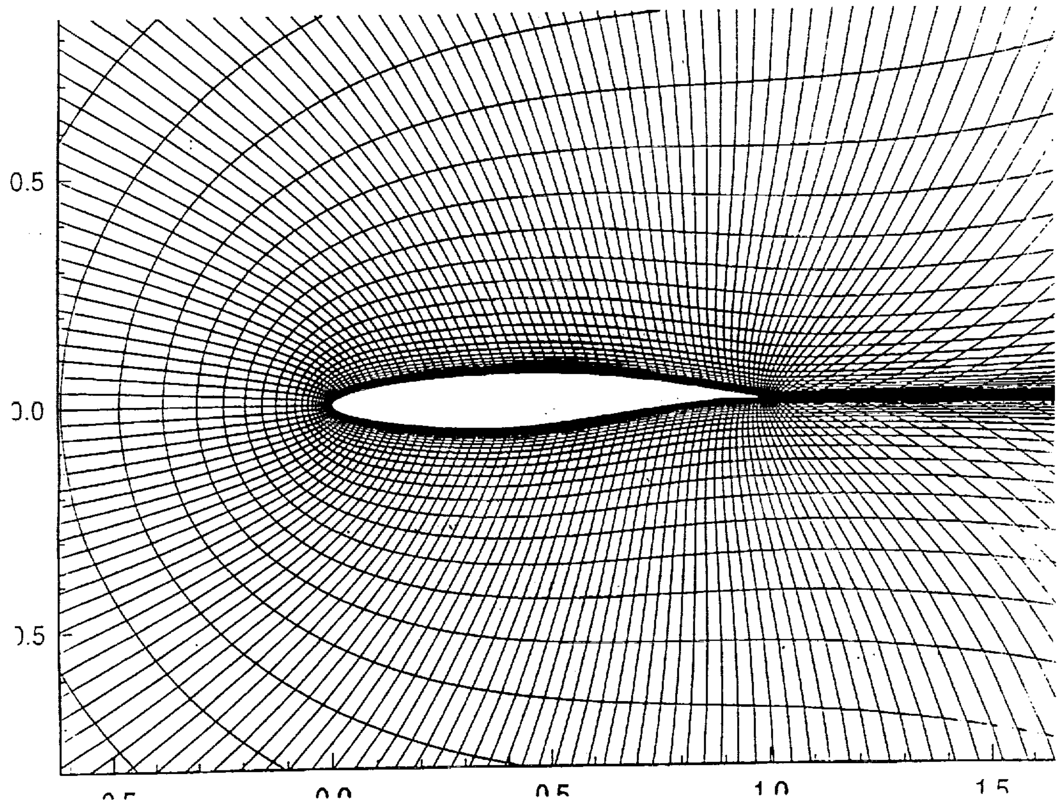
- Second-Order Closures

Boundary Conditions: Wall-Functions

III.3. Validation Procedure / Results

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

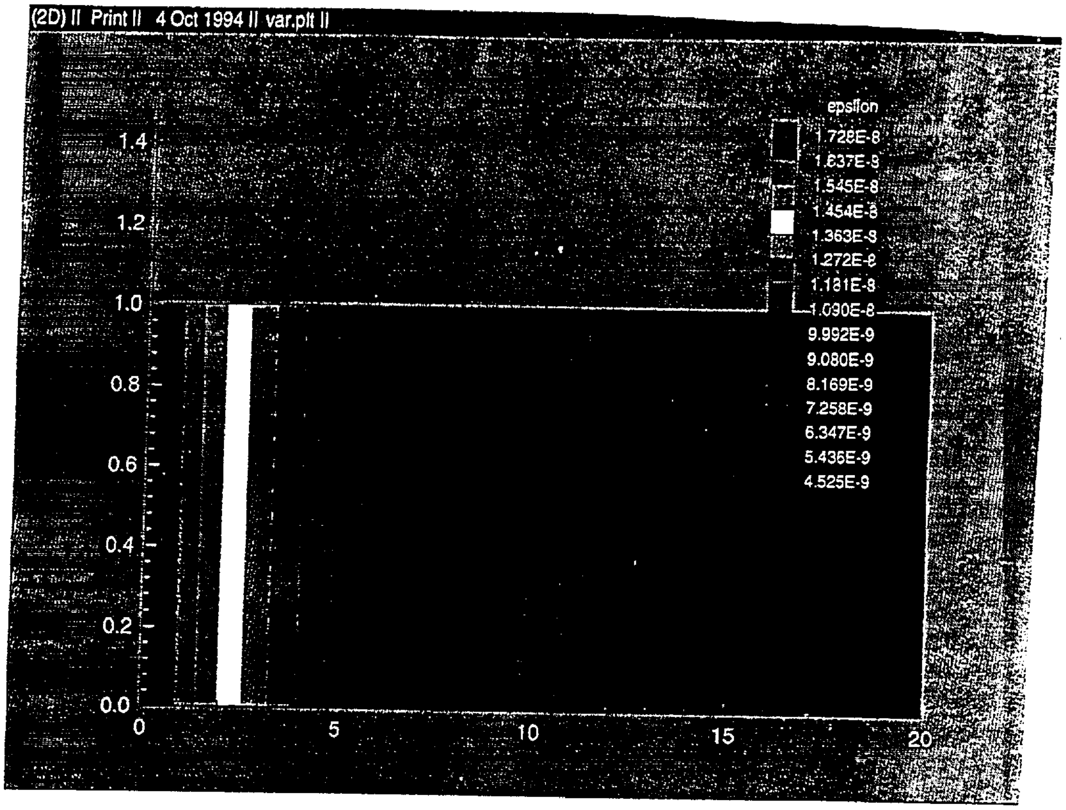
- Results and Comparison of models

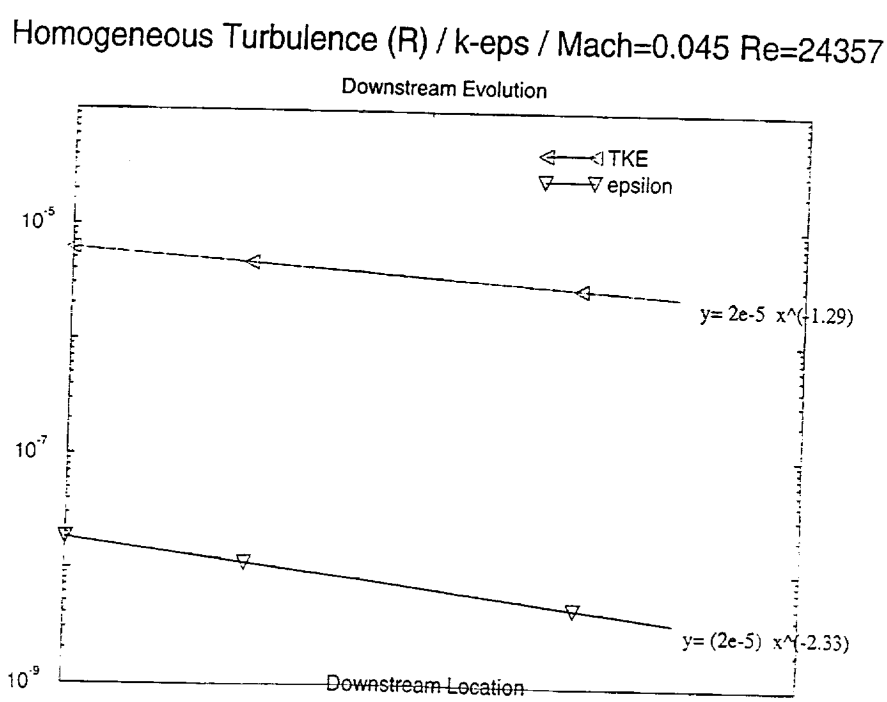


FUTURE WORK

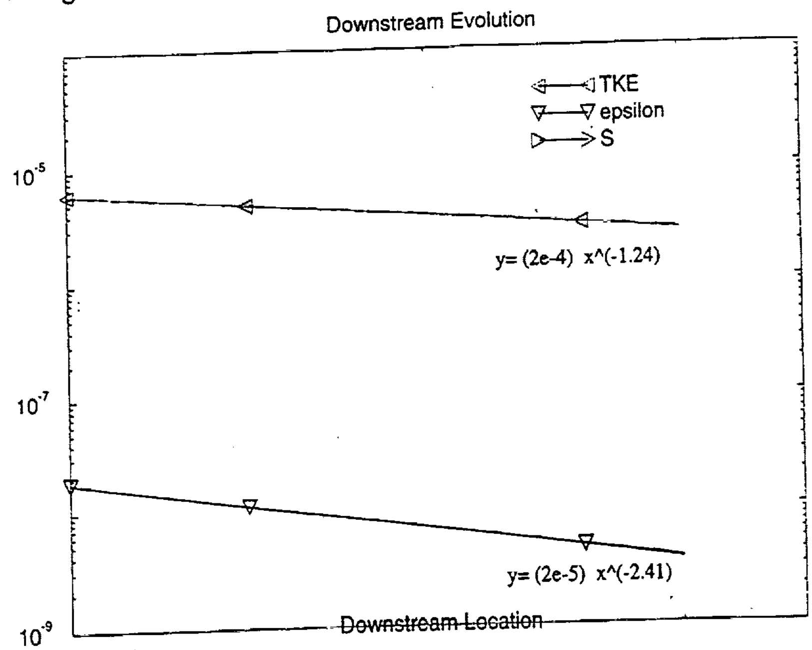
- Numerics
  - 2D $\Rightarrow$ 3D
  - More Complex Wall Functions
  - Realizability Conditions (SOC)
- Modeling
  - Refinement of Existing Models (ε_d , $< pd >$)
  - Shock Oscillation Model

ORIGINAL PAGE IS
OF POOR QUALITY

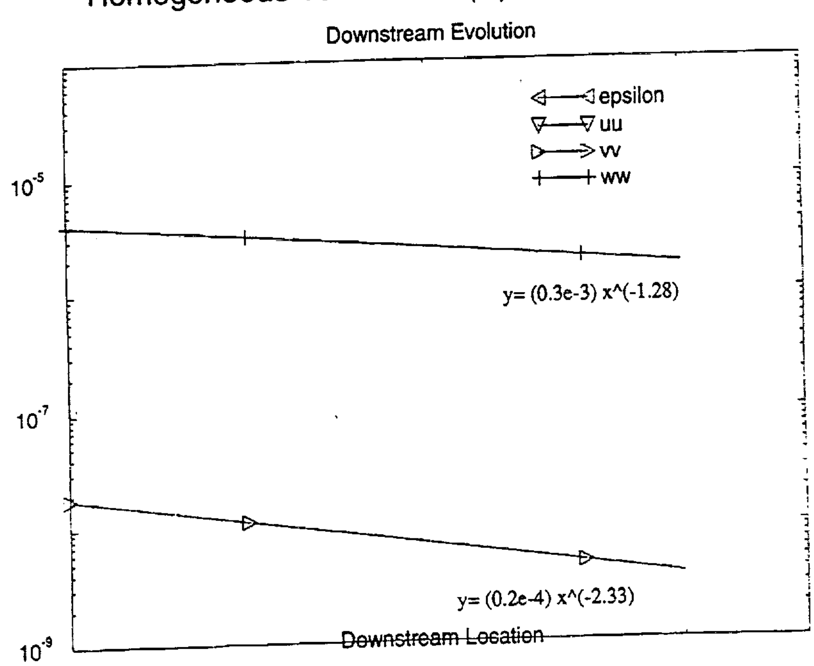




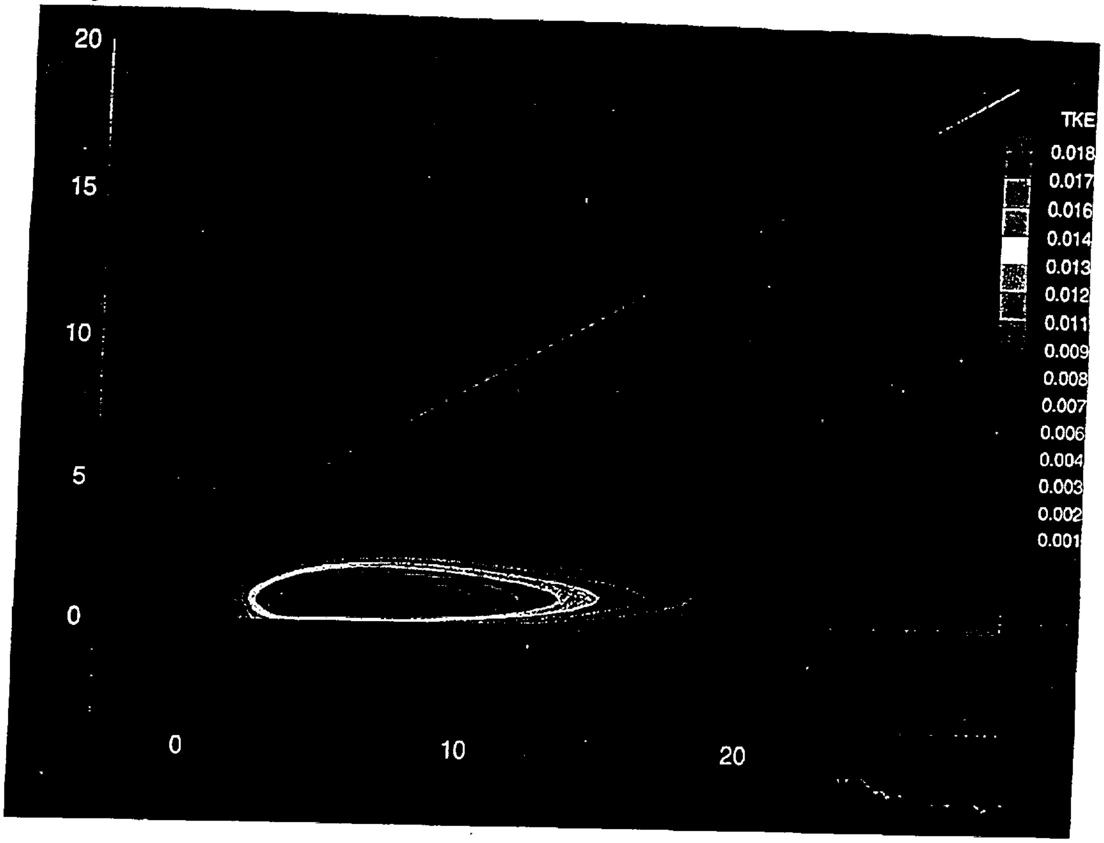
Homogeneous Turbulence (R) / k-eps-S / Mach=0.045 Re=24357

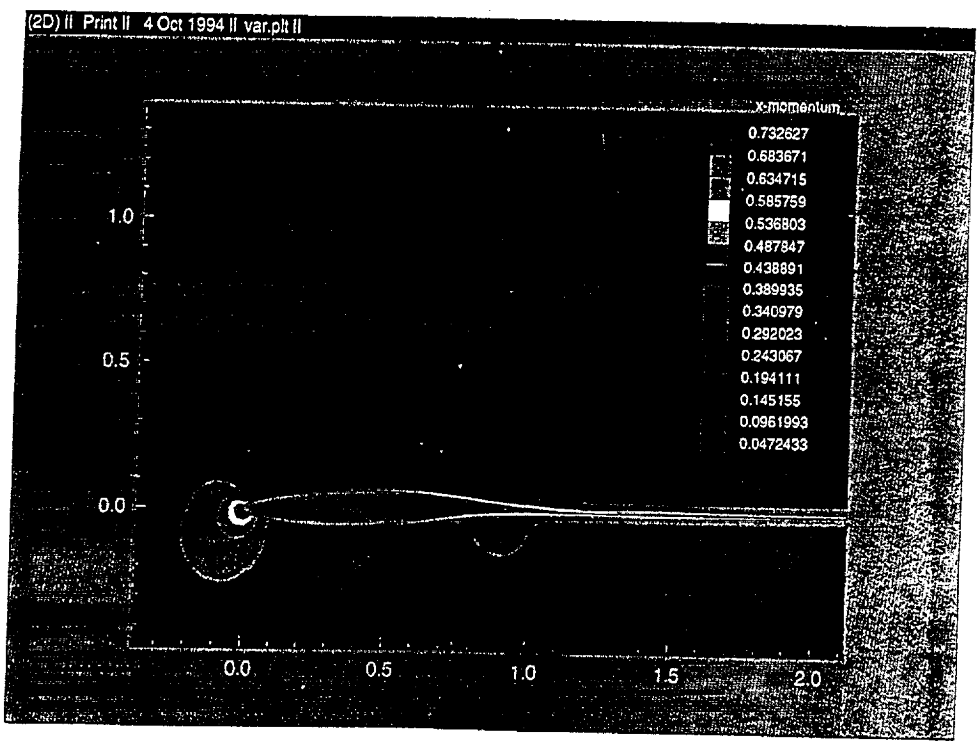


Homogeneous Turbulence (R) / RSC / Mach=0.5



ORIGINAL PAGE IS
OF POOR QUALITY

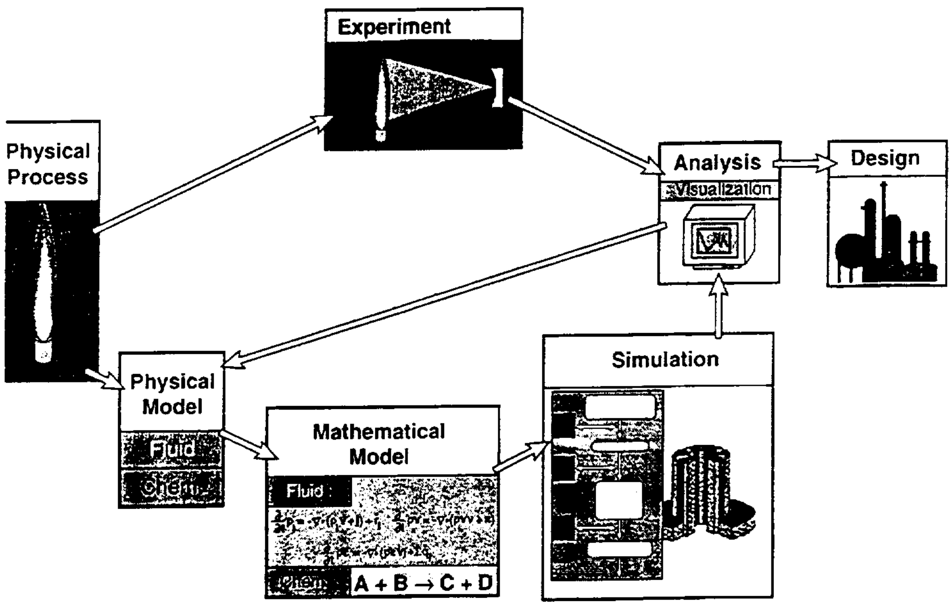




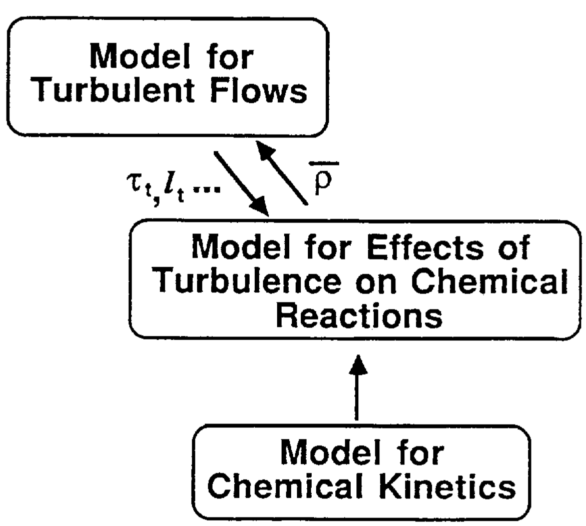
MODELING OF TURBULENT CHEMICAL REACTION

N95- 27899

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

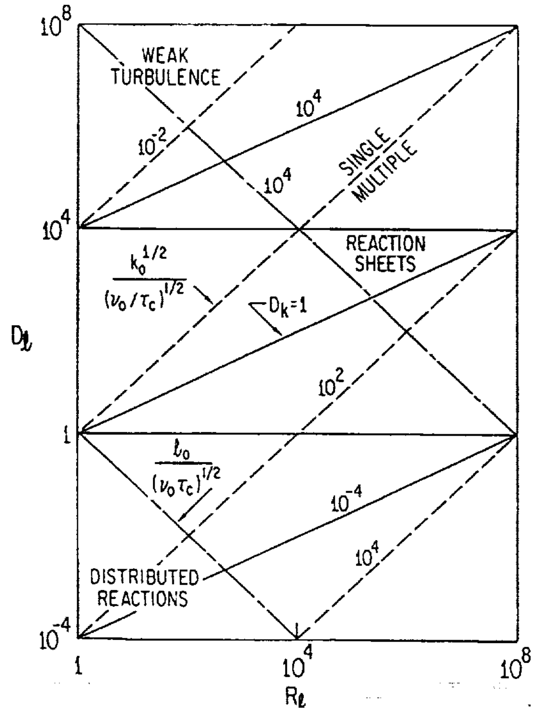


Modeling Turbulent Reacting Flows



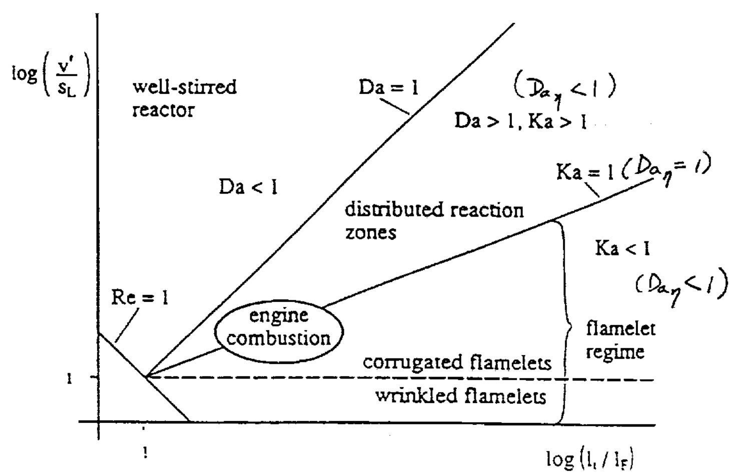
PRECEDING PAGE BLANK NOT FILMED

Regimes of Turbulent Combustion



Turbulent Reactive Flows edited by P.A. Libby and F.A. Williams (1994)

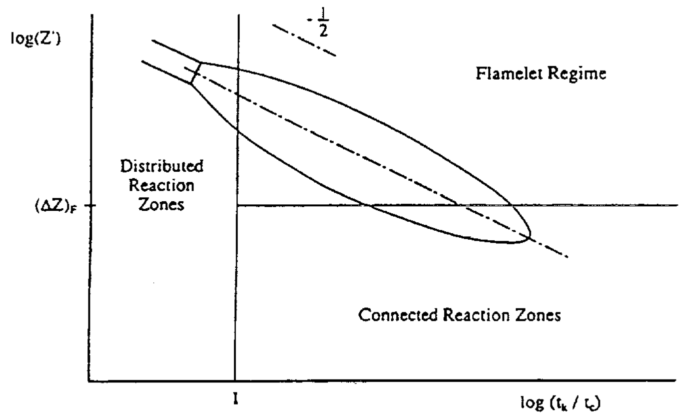
Regimes of Premixed Turbulent Combustion



$Ka = 1/Da_\eta$

Turbulent Reactive Flows edited by P.A. Libby and F.A. Williams (1994)

Regimes of Non-Premixed Turbulent Combustion



Turbulent Reactive Flows edited by P.A. Libby and F.A. Williams (1994)

Chemical Closure Models

(1) Laminar Chemistry

$$<\rho w_i> = \rho w_i(\overline{Y}_i, \overline{T})$$

(2) Fast Chemistry

$$<\rho w_i> \approx -\frac{1}{2}\overline{\rho}\tilde{\chi}_f \frac{\partial^2 Y^e(f)}{\partial^2 f}$$

(3) Flamelet model

$$<\rho w_i> = \iint \rho w_i(\eta, \chi_f) P_{f,\chi_f}(\eta, \varepsilon_f) d\eta d\varepsilon_f$$

(4) Assumed PDF:

$$<\rho w_i> = \int .. \int \rho w_i(\phi_i) \cdot P_\phi d\phi_1 d\phi_2 .. d\phi_n$$

Assumed the shape of P_ϕ.

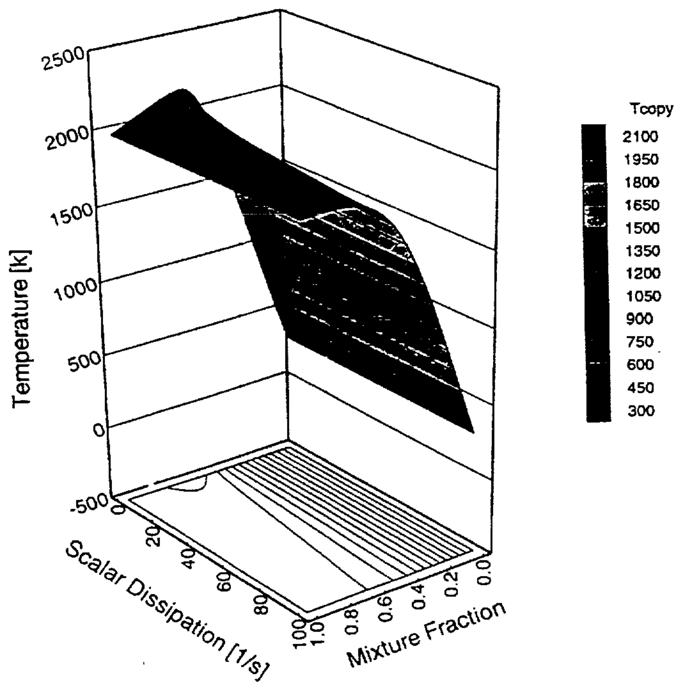
(5) Scalar PDF method:

Solve for P_ϕ directly.

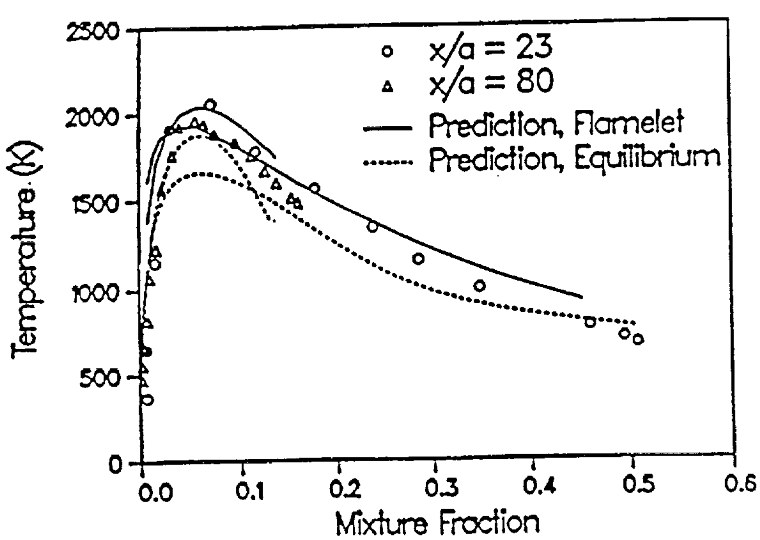
(6) Conditional Moment Closure (CMC)

$$<\rho w_i> = \int <\rho w_i | \eta> \cdot P_f(\eta) d\eta$$

Flamelet library with one side being burned premixed flameϕ=1.4

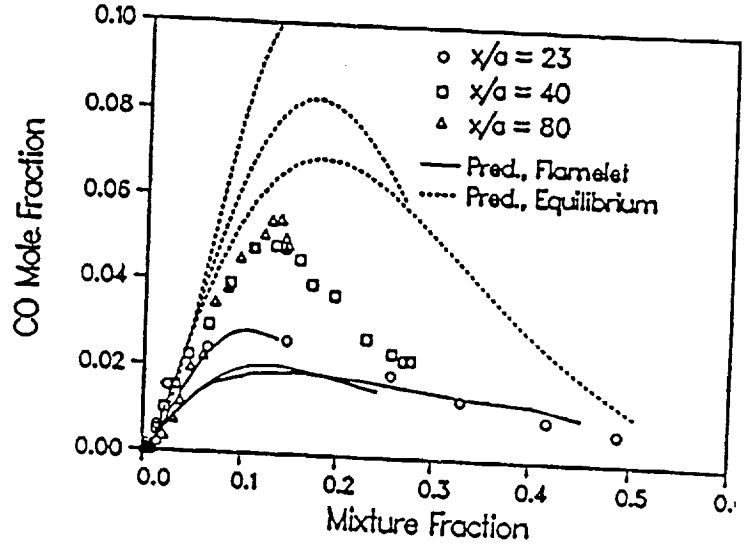


Flamelet Model: 69%H2+31%CH4
Turbulent Jet Flame, Rey.=10,000



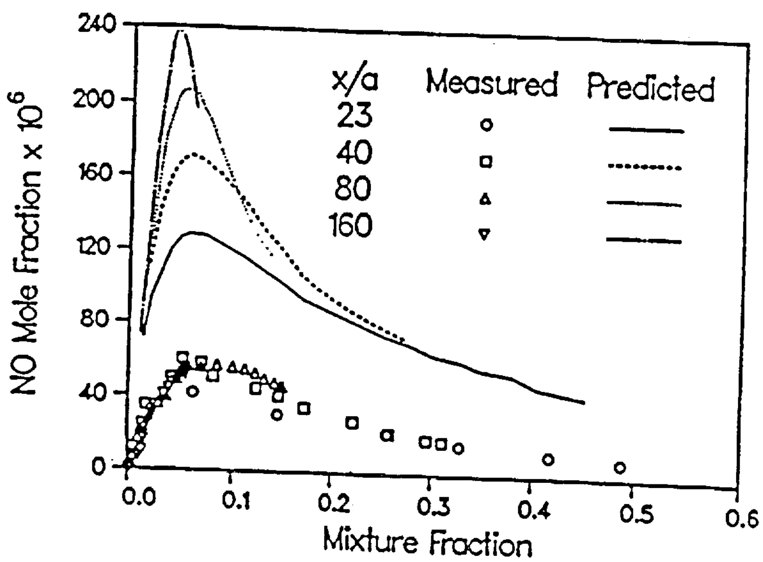
Vranos, et al. "Nitric Oxide Formation and Differential Diffusion in a Turbulent Methane-Hydrogen Diffusion Flame," 24th Symposium(International) on Combustion/The Combustion Institute, 1992/pp. 377-384

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



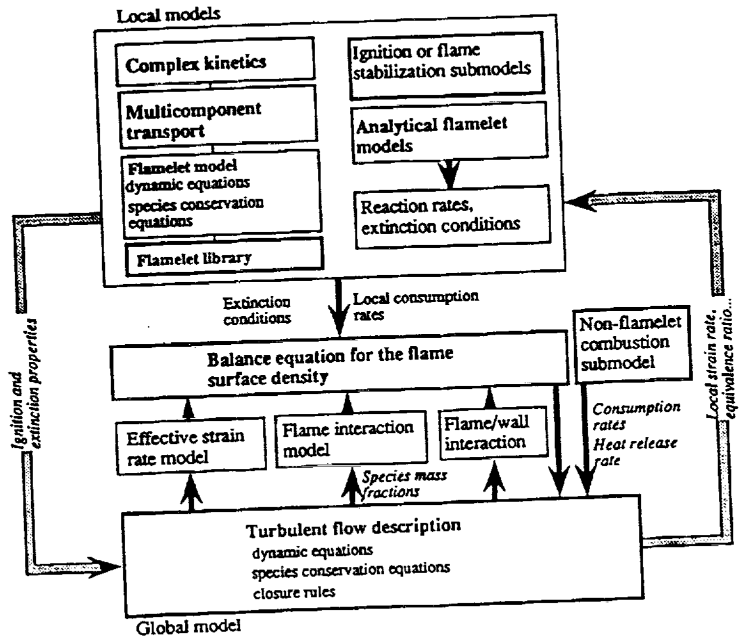
Vranos, et al. "Nitric Oxide Formation and Differential Diffusion in a Turbulent Methane-Hydrogen Diffusion Flame," 24th Symposium(International) on Combustion/The Combustion Institute, 1992/pp. 377-384

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



Vranos, et al. "Nitric Oxide Formation and Differential Diffusion in a Turbulent Methane-Hydrogen Diffusion Flame," 24th Symposium(International) on Combustion/The Combustion Institute, 1992/pp. 377-384

Advanced Flamelet Approach



Conditional Moment Closure (CMC)

Definition:

$$< Y_i | \eta > \equiv < Y \ (\bar{x}, t) | f(\bar{x}, t) = \eta >$$

Equation:

$$< \rho | \eta > \frac{\partial < Y_i | \eta >}{\partial t} + < \rho \tilde{u} | \eta > \cdot \nabla < Y_i | \eta > + \frac{\nabla \cdot \{ < \rho u' y' | \eta > P_f(\eta) \}}{P_f(\eta)}$$

$$= < \rho w_i | \eta > + < \rho D_i \nabla f \cdot \nabla f | \eta > \frac{\partial^2 < Y_i | \eta >}{\partial \eta^2}$$

Modeling:

$$< w_i | \eta > \approx w_i (< T | \eta >, < Y_i | \eta >, ..)$$

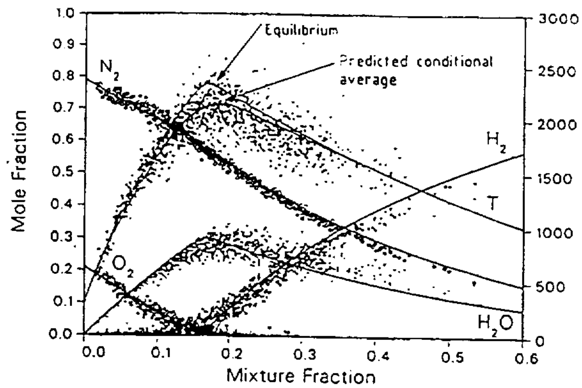
$$< \rho D_i \nabla f \cdot \nabla f | \eta > \approx < \rho D_i \nabla f \cdot \nabla f > \approx \frac{1}{2} \bar{\rho} \chi_f$$

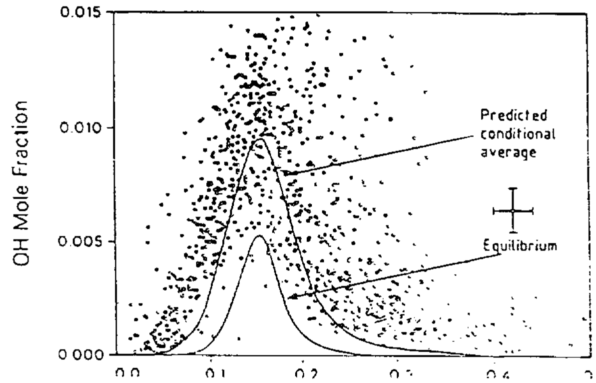
$$< \rho \tilde{u} | \eta > \approx \overline{\rho \tilde{u}}$$

$$< \rho u' y' | \eta > \approx 0$$

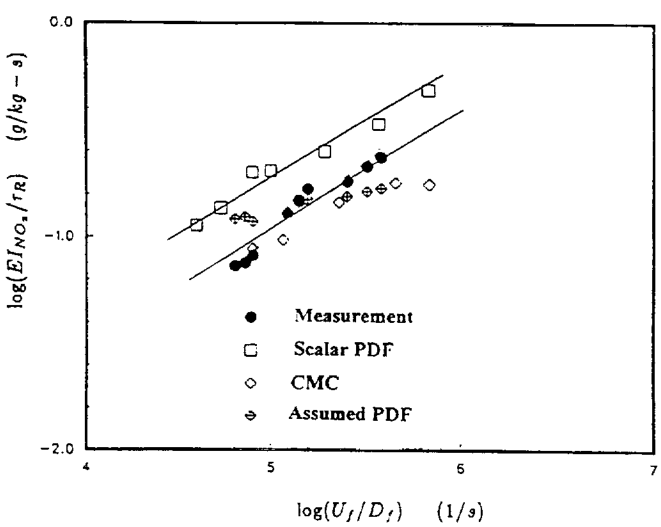
$$< \rho | \eta > \approx \rho (< Y_i | \eta >, < T | \eta >)$$

Conditional Moment Closure (CMC)





NOx Emissions from Turbulent H2 Jet Flames



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:

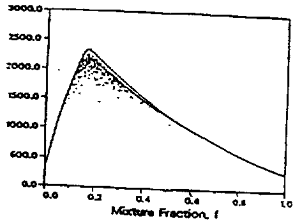
- **NO_X from methane jet flames with reduced chemistry**
- **Sooting flames**
- **2-D flows**

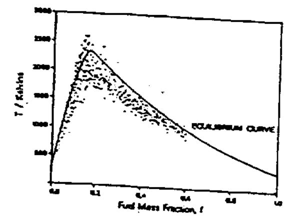
Research Topics:

- **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**

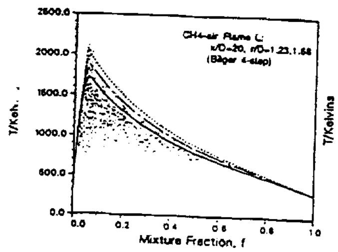
Departures From Chemical Equilibrium

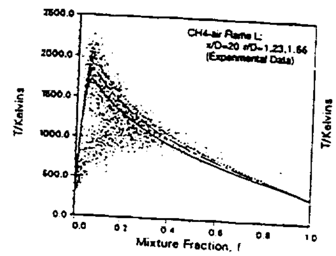
(PDF) Hydrogen (EXP.)



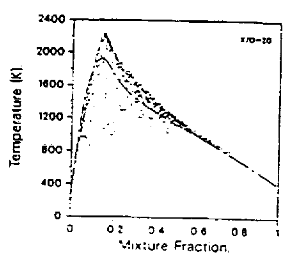


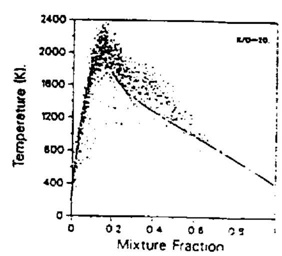
Methane





Methanol





Mixing Models for PDF Methods

- Modified Curl's Model (stochastic)

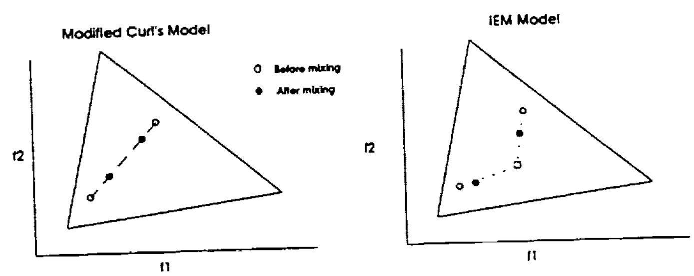
$$-\sum_{\alpha=1,\beta=1}^{k}\frac{\partial^2}{\partial\psi_\alpha\partial\psi_\beta}\left\{\left\langle\varepsilon_{\alpha\beta}|\bar{\phi}=\bar{\psi}\right\rangle\tilde{P}_{\bar{\phi}}(\bar{\psi},t)\right\}=$$

$$\frac{1}{\tau_{mix}}\left\{\iint_{\psi'\psi''}\left[\tilde{P}_{\bar{\phi}}(\psi',t)\tilde{P}_{\bar{\phi}}(\psi'',t)H(\psi',\psi''|\bar{\psi})-\tilde{P}_{\bar{\phi}}(\bar{\psi},t)\right]d\psi'd\psi''\right\}$$

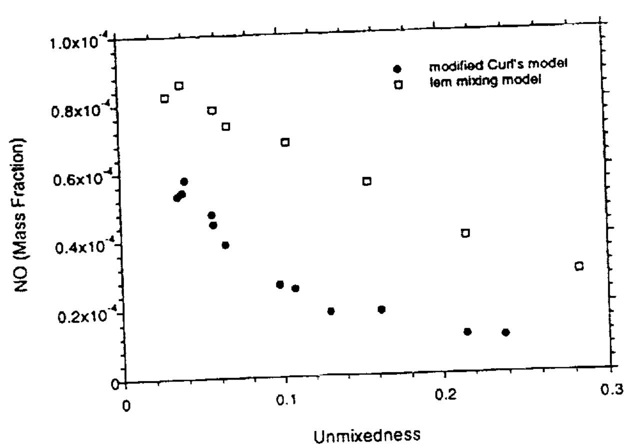
- IEM (Interaction-by-Exchange-with-the-Mean) Model (deterministic)

$$-\sum_{\alpha=1,\beta=1}^{k}\frac{\partial^2}{\partial\psi_\alpha\partial\psi_\beta}\left\{\left\langle\varepsilon_{\alpha\beta}|\bar{\phi}=\bar{\psi}\right\rangle\tilde{P}_{\bar{\phi}}(\bar{\psi},t)\right\}=\frac{C_\phi}{2\tau_{mix}}\frac{\partial}{\partial\psi_\alpha}\left[(\bar{\psi}-\tilde{\phi})\tilde{P}_{\bar{\phi}}(\bar{\psi},t)\right]$$

Mixing Frequency: $\omega_{mix}=1/\tau_{mix}$

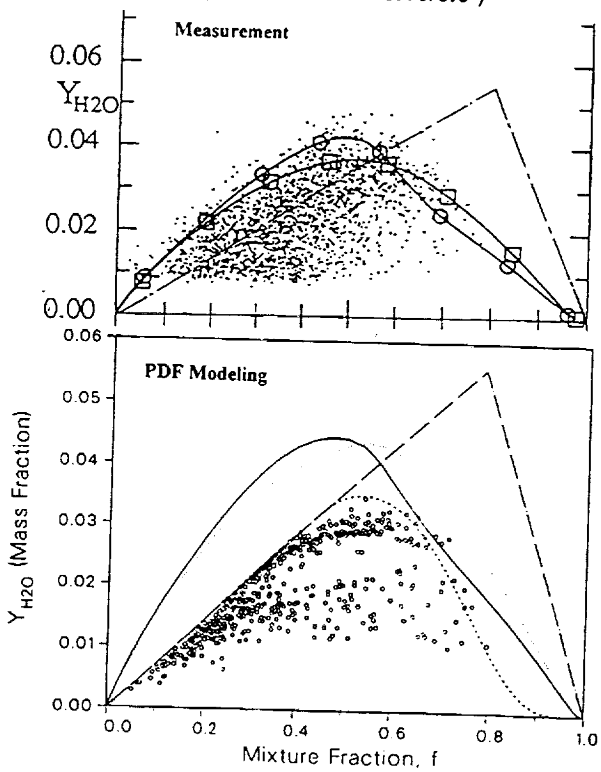


PaSR: H2/NOx Detailed Chemistry ϕ=1 τ=1ms

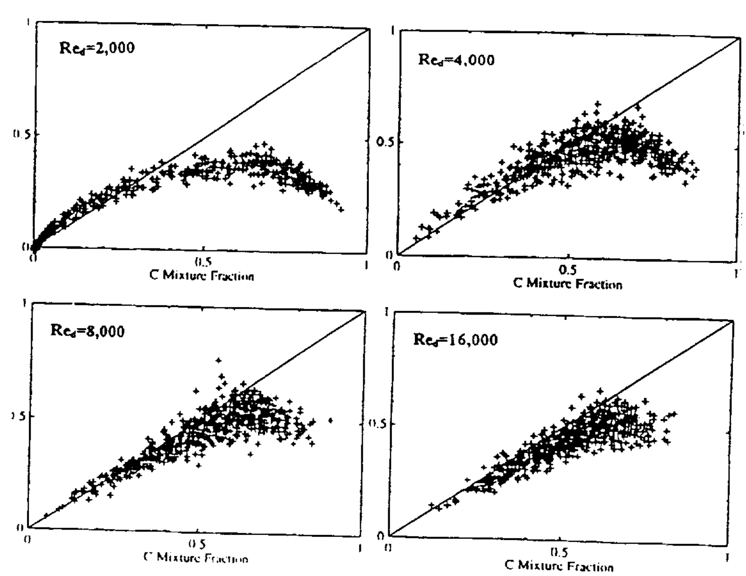


Comparison of Predicted and Measured H2O Mass Fractions
Turbulent Nonpremixed Jet Flames

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

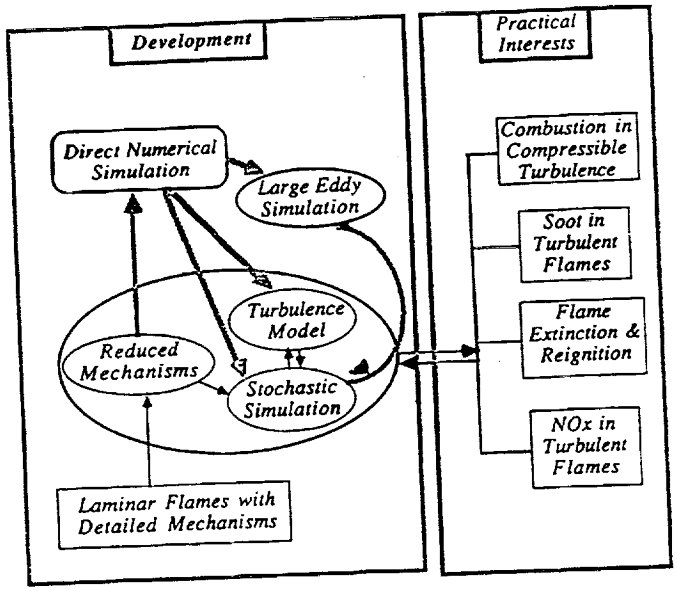


Experimental Evidence of Preferential Diffusion in Turbulent Jet Flames
(Fuel: 36%H_2+64% CO_2)



"Differential Molecular Diffusion in Reacting and Nonreacting Turbulent Jets of H2/CO2 mixing with Air," L.L.Smith Ph D Thesis, University of California at Berkeley (1994)

Computation of Turbulent Reacting Flows



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:

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

- Reynolds stresses will be recasted as:

$$-\rho\overline{u_i u_j} \equiv \mu_T(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3}\frac{\partial U_k}{\partial x_k}\delta_{ij}) + \underbrace{[-\rho\overline{u_i u_j} - \mu_T(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3}\frac{\partial U_k}{\partial x_k}\delta_{ij})]}_{T_{ij}}$$

$$\mu_T \equiv C_\mu \frac{k^2}{\varepsilon}$$

- CFD equations become:

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

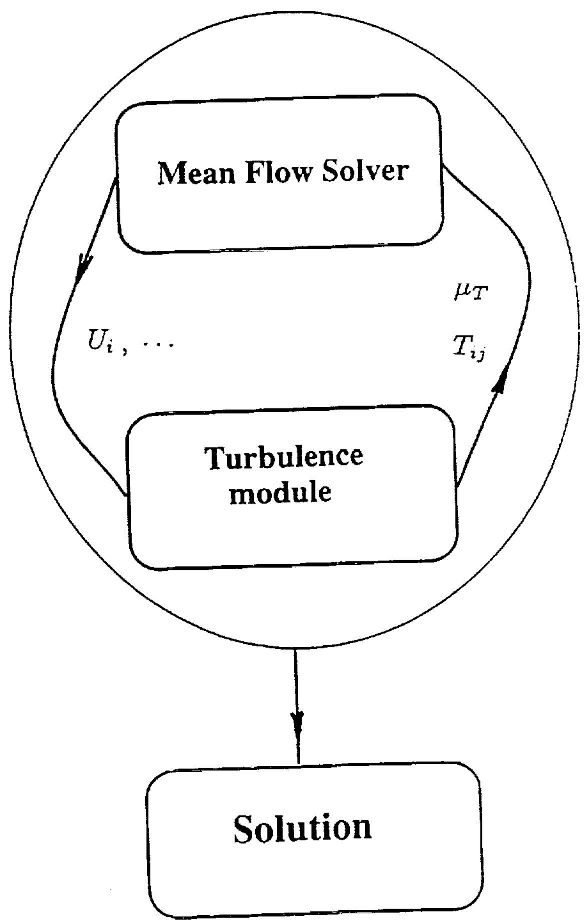
- The task of turbulence module: Provide μ_T and T_{ij}

- Turbulence Module:
  - ◇ Input: U_i, ρ and μ ... from the mean flow solver
  - ◇ Output:

$$\mu_T = C_\mu \frac{k^2}{\varepsilon} \qquad [\frac{Dk}{Dt} = \ldots, \qquad \frac{D\varepsilon}{Dt} = \ldots]$$

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

  - ◇ 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



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 μ_T and T_{ij}
  - ◇ Built-in models without wall function:
    - Launder-Sharma and Chien $k-\varepsilon$ models
    - CMOTT $k-\varepsilon$ model
  - ◇ Built-in models with wall function:
    - $k-\omega$ model, standard $k-\varepsilon$ model
    - CMOTT $k-\varepsilon$ 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 μ_T
  - ◇ Build-in models without wall function:
    - Baldwin-Lomax model and Chien $k-\varepsilon$ model
    - CMOTT $k-\varepsilon$ 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.

DESCRIPTION OF TURBULENCE SUB-PROGRAM

J. Zhu
Institute for Computational Mechanics in Propulsion
NASA Lewis Research Center
Cleveland, Ohio

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

  $S_\phi = S_1 + S_2\phi, \quad S_1 \geq 0$ 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

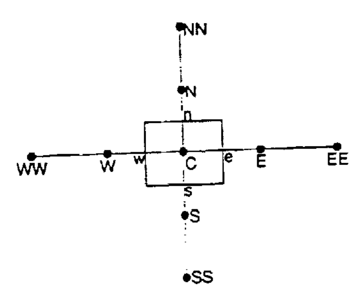
- Convection term: HLPA scheme

  (Hybrid Linear/Parabolic Approximation)

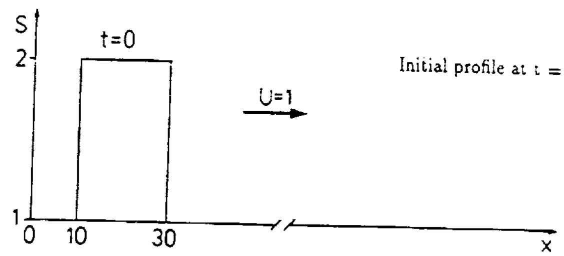
$$\phi_w = \phi_W + \gamma(\phi_C - \phi_W)\hat{\phi}_W, \quad \hat{\phi}_W = \frac{\phi_W - \phi_{WW}}{\phi_C - \phi_{WW}}$$

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

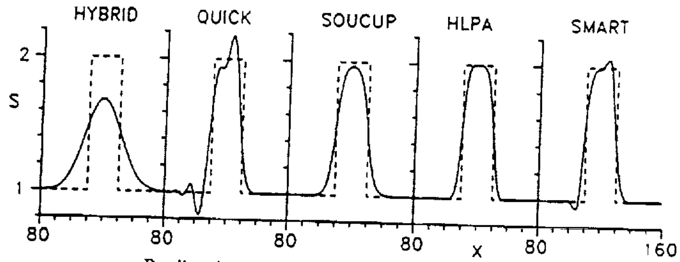
- Second-order accurate
- Bounded (non-oscillatory)
- Diagonally dominant matrix



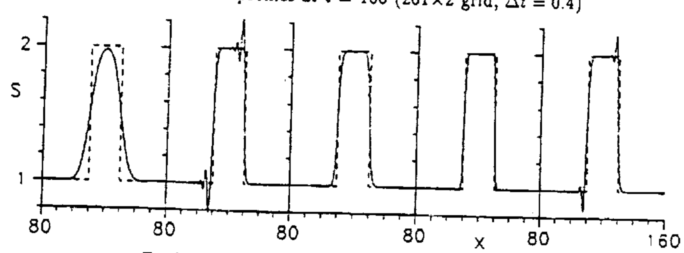
Example 1



Initial profile at t = 0

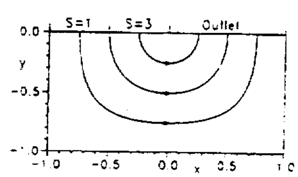


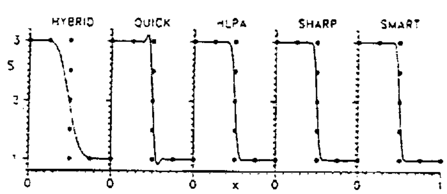
Predicted profiles at t = 100 (201×2 grid, $\Delta t = 0.4$)



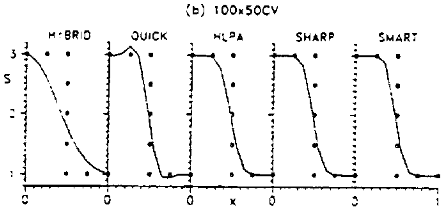
Predicted profiles at t = 100 (1001×2 grid, $\Delta t = 0.1$)

Example 2



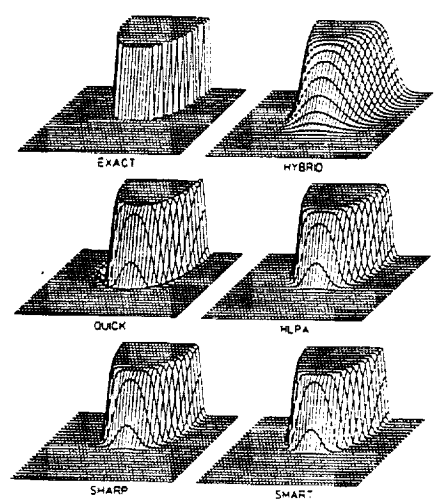


(b) 100×50CV



(a) 20×10CV

S-profiles at outlet (O, 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: ϕ 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

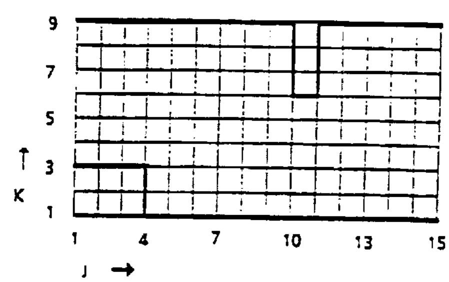
NPARC2D Version

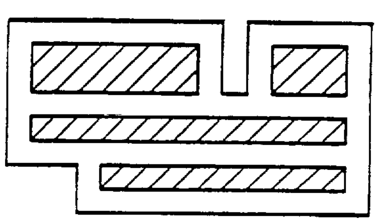
- Grid arrangement

  Control volume centers

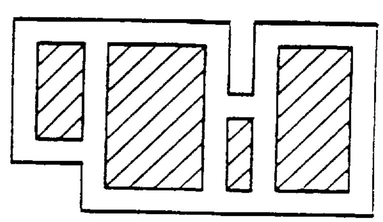
  Boundary nodes

  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: 5×2 parameters
  4. Boundary conditions: 7×2 parameters

- Output
  1. To the main code: μ_t
  2. For post-processing: $K,\ \epsilon,\ y^{+},\ y_n,\ f_\mu$

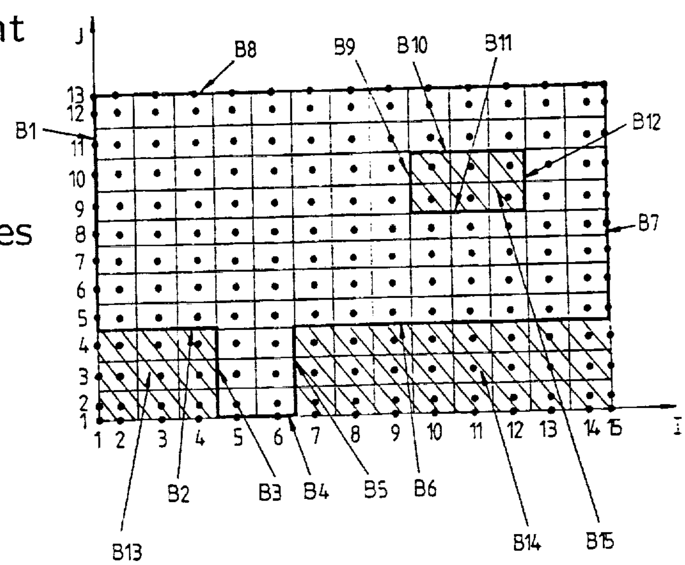
FAST2D Version

- Grid arrangement

  CV centers

  Boundary nodes

  Embedded bodies



- Vectorization

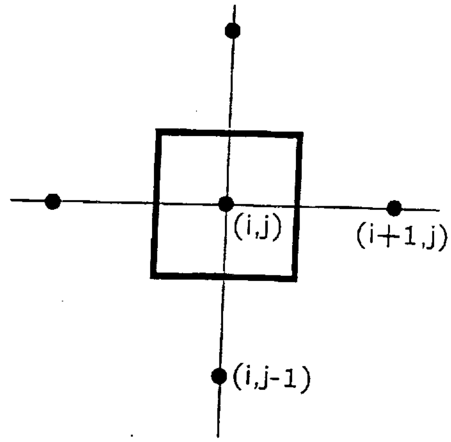
  Single-index:

  ii=i+(j-1)ni

  ϕ(i,j)=ϕ(ii)

  ϕ(i+1,j)=ϕ(ii+1)

  ϕ(i,j-1)=ϕ(ii-ni)

  Control parameter:

  $$\text{KBLK} = \begin{cases} 1 & \text{for computational nodes} \\ 0 & \text{otherwise} \end{cases}$$

  ϕ=KBLK$\cdot\phi_c$+(1-KBLK)ϕ_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

N95- 27900

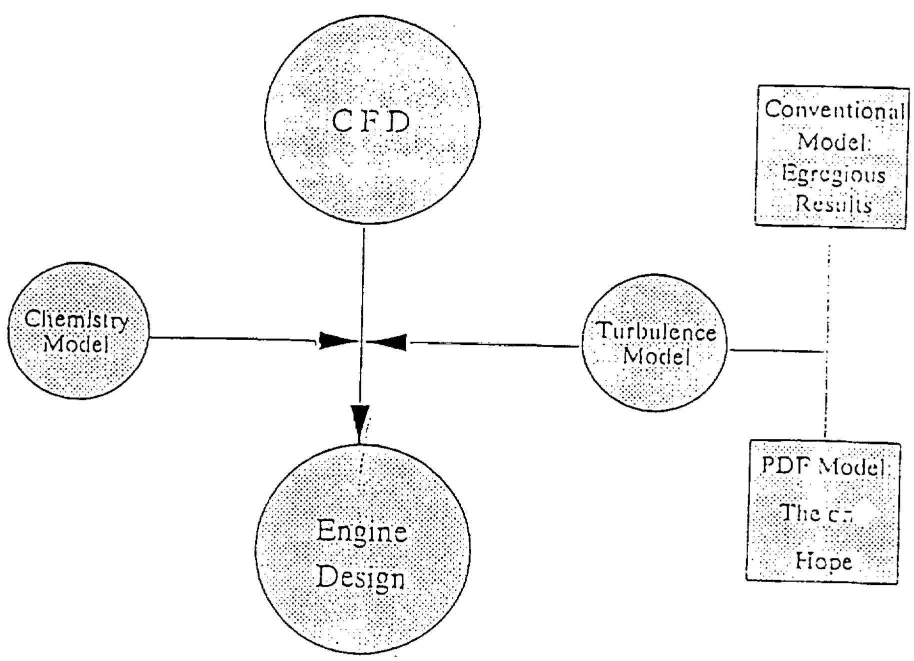
PDF METHODS FOR TURBULENT REACTIVE FLOWS

Andrew T. Hsu
NYMA, Inc.
NASA Lewis Research Center
Brook Park, Ohio

OUTLINE

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

COMPUTATION OF TURBULENT COMBUSTION



GOVERNING EQUATIONS

$$
\begin{aligned}
&\rho_{,t} + (\rho u_i)_{,i} = 0 \\
&(\rho u_i)_{,t} + (\rho u_j u_i)_{,j} = -p_{,i} + \tau_{ij,j} \\
&(\rho E)_{,t} + (\rho u_j E)_{,j} = -q_{i,i} + \Phi \\
&(\rho Y_k)_{,t} + (\rho u_j Y_k)_{,j} = (\rho D Y_{k,j})_{,j} + \omega_k
\end{aligned}
$$

$$A_{,t} \equiv \frac{\partial A}{\partial t}$$

$$A_{,j} \equiv \frac{\partial A}{\partial x_j}$$

CLOSURE PROBLEM:

$$
\begin{aligned}
u_i &= \overline{u_i} + u'_i, \\
Y_i &= \overline{Y_i} + Y'_i,
\end{aligned}
$$

$\overline{u''_i u''_j}$ — Turbulence Modeling

$\overline{Y''_i u''_j}$ — Analogy of shear stress: diffusion model.

$\overline{\rho w_i}$ — ???

$$\rho w_i = \rho w_i(Y_1, ..., Y_n, T \quad)$$

But in general:

$$\overline{\rho w_i} \neq \bar{\rho} w_i(\overline{Y_1}, ..., \overline{Y_n}, \overline{T} \quad)$$

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

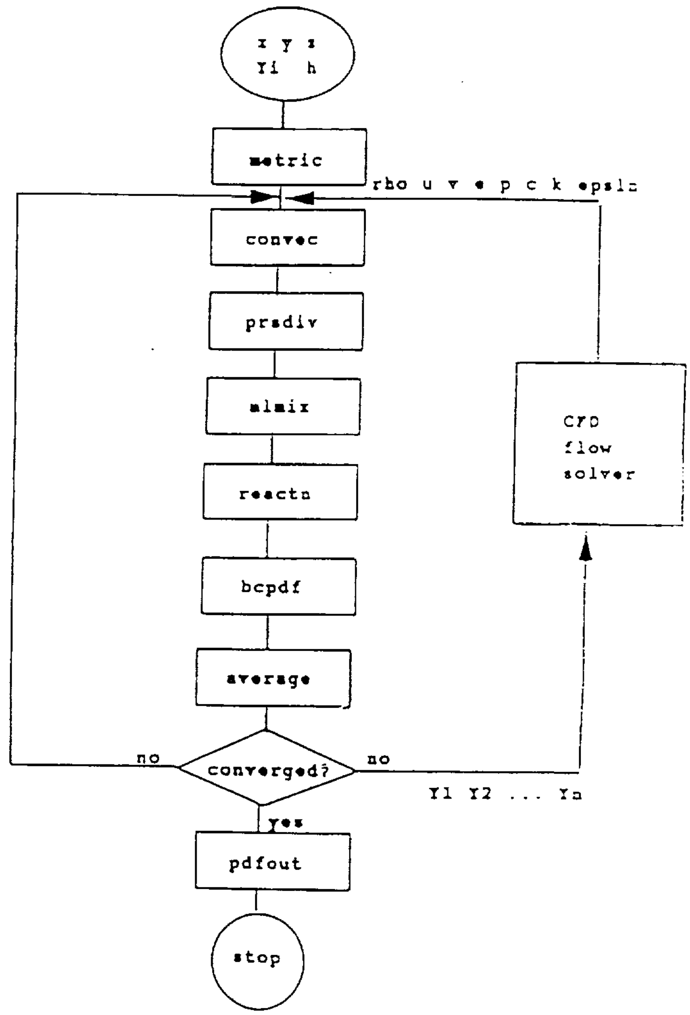
- Approach:
  - ◇ Joint pdf method for scalar compositions.
  - ◇ Moment closure schemes for velocity field.
  - ◇ Develop hybrid solver consisting of Monte Carlo method and finite-difference/finite-volume method.

PDF Modeling of Turbulent Reactive Flows

- Current status (Lewis)

$$
\begin{aligned}
&(\rho P)_{,t} + (\rho < u_j | Y_i, h > P)_{,j} + (\rho w_j P)_{,Y_j} \\
=\ &(D_t P_{,j})_{,j} + M(P) - (S_p P)_{,h}.
\end{aligned}
$$

  - ◇ 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

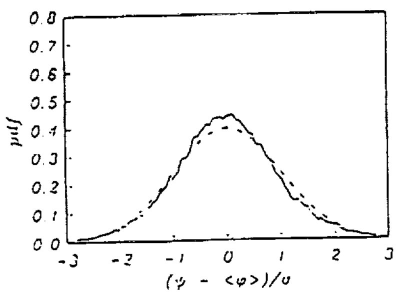


Figure 2. Asymptotic pdf distribution for a scalar in homogeneous turbulence. — present model; · · · Gaussian.

Modified curl's model

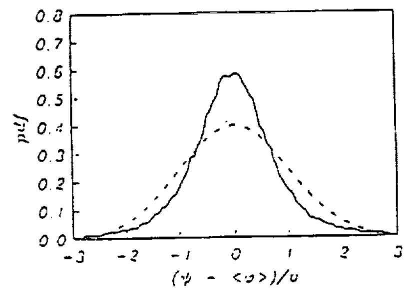
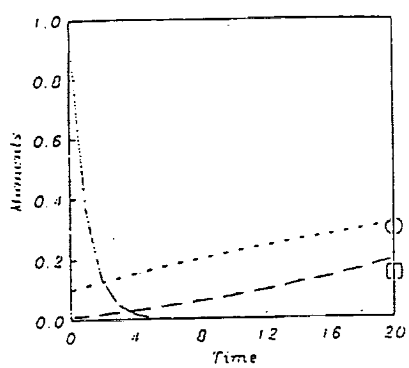


Figure 1. Asymptotic pdf distribution for a scalar in homogeneous turbulence. — modified Curl model; · · · Gaussian.

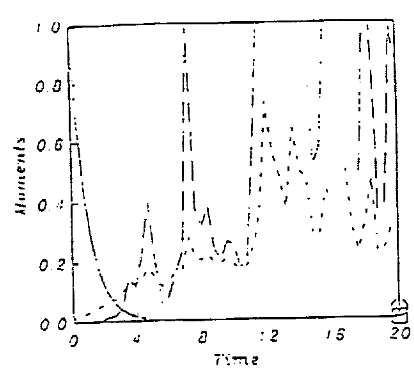
Scalar field in homogenous turbulence
3rd and 4th moments compared with Gaussian

Current model



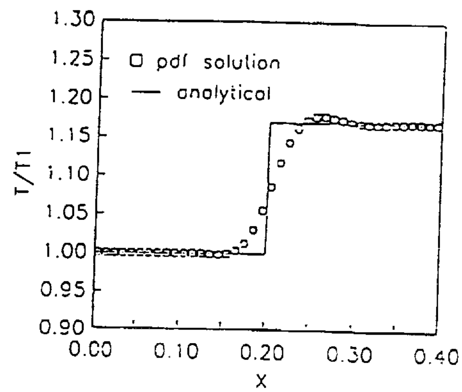
Evolution of moments from the present model. — standard deviation, · · · 0.1 × fourth central moment, – – 0.01 × sixth central moment, o 0.1 × fourth moment for Gaussian distribution, □ 0.01 × sixth moment for Gaussian distribution.

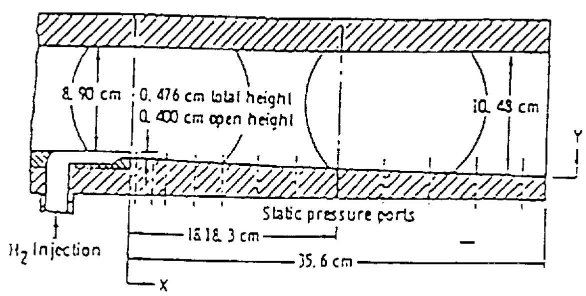
Modified curl's model



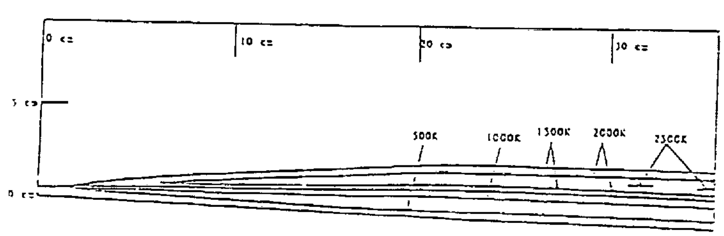
Evolution of moments from the modified Curl model. — standard deviation, · · · 0.01 × fourth central moment, – – 0.0001 × sixth central moment, o 0.01 × fourth moment for Gaussian distribution, □ 0.0001 × sixth moment for Gaussian distribution.

Temperature across an oblique shock: pdf solution compared with analytical prediction.



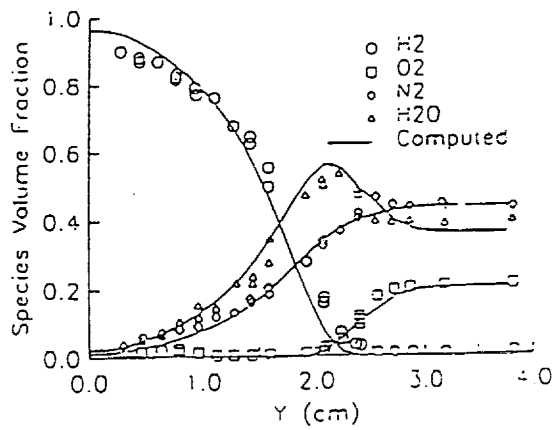


Supersonic hydrogen combustor
(Exp. Burrows & Kurkov, 1973)



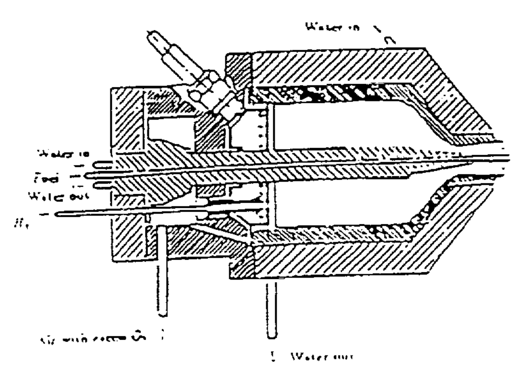
Temperature Contour (pdf solution)

Supersonic hydrogen combustor
Mole fraction:
pdf solution compared with exp. data

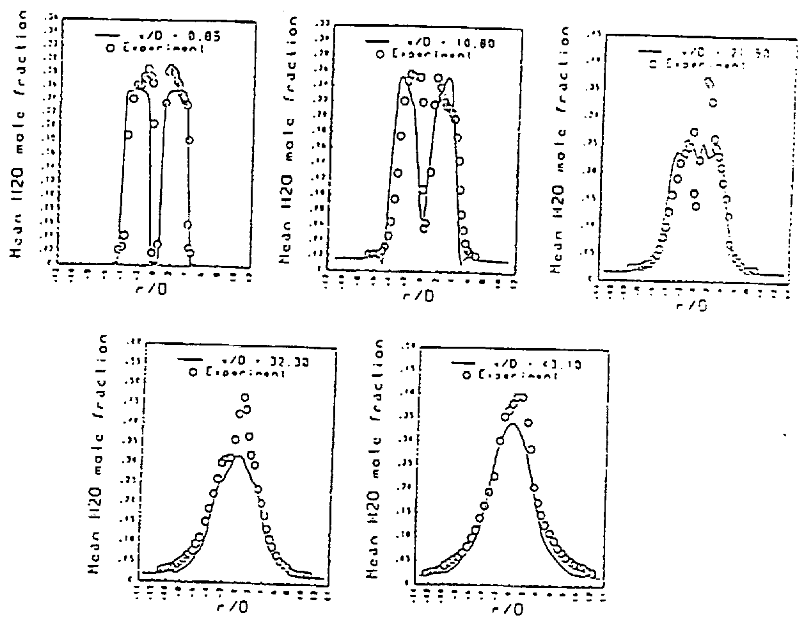


Coaxial burner: geometry and test condition
(Exp. Cheng, et al. 1991)

Exit Conditions	Hydrogen Jet	Outer Jet	Ambient Air
Mach Number	1	2	0
Temperature, K	545	1250	300
Velocity, m/s	1780	1417	0
Pressure, MPa	.112	.107	.101
Mass Fraction			
Y_{H_2}	1.	0.	0.
Y_{O_2}	0.	.245	.233
Y_{N_2}	0.	.55	.757
Y_{H_2O}	0.	.173	.01

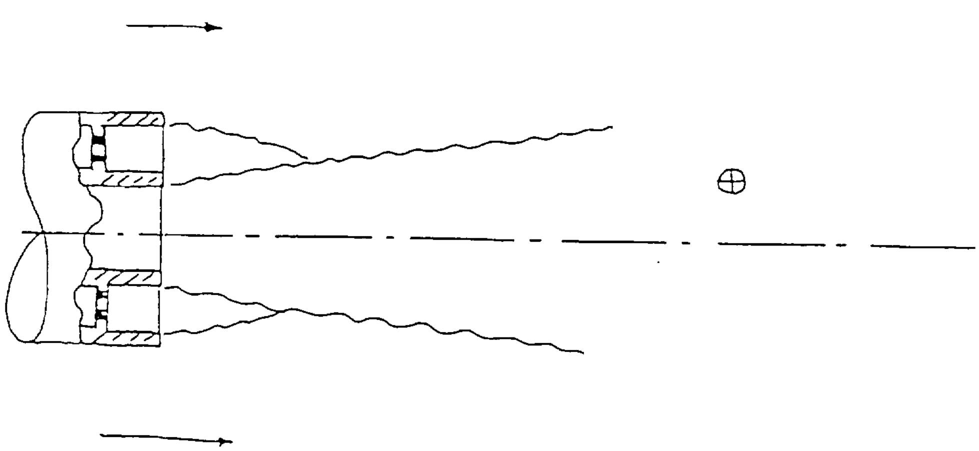


Mean H2O mole fraction
Coaxial burner
pdf solution compared with exp. data

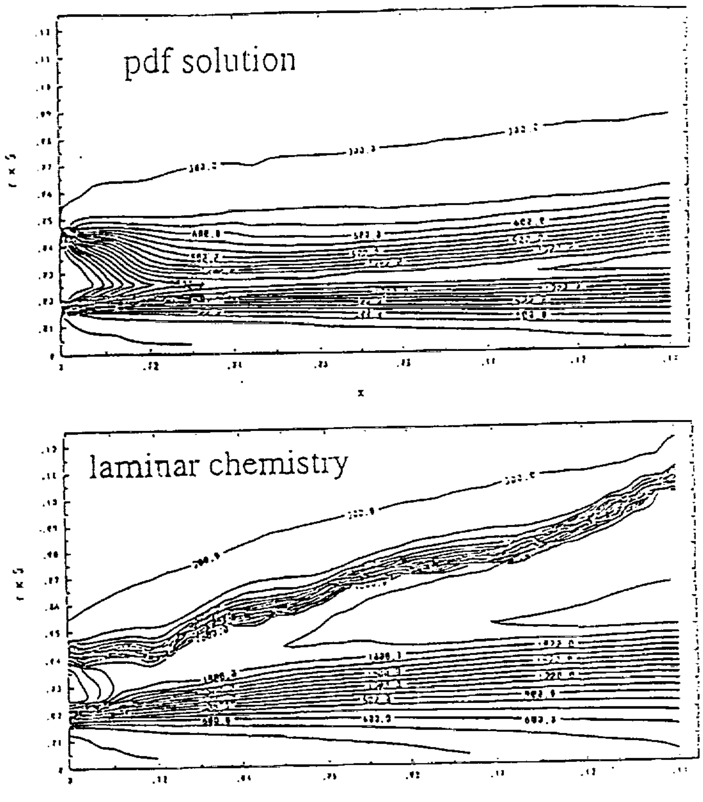


Piloted flame (Masri et al., 1994)

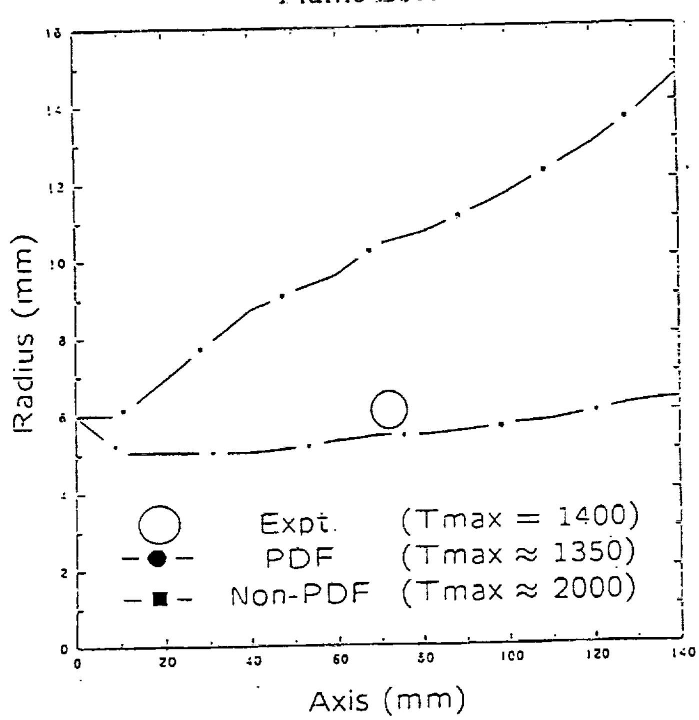
Fuel: 45% CO, 15% H2, and 40% N2
Flame close to extinction



Piloted Flame
Mean Temperature



Piloted Flame
Flame Location



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 requiement.
  - ◇ General chemistry package.
  - ◇ Parallelized workstation version.
- Technology transfer: workshops
  - ◇ July, 1993; code released to 15 US institutions.
  - ◇ October, 1994.

A COMPOSITION JOINT PDF METHOD FOR THE MODELING OF SPRAY FLAMES

N95- 27901

M.S. Raju
Nyma, Inc.
NASA Lewis Research Center
Cleveland, Ohio

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

$$\bar{\rho}\tilde{p}_{,t} + \bar{\rho}\tilde{u}_i\tilde{p}_{,x_i} + [\bar{\rho}w_\alpha(\underline{\psi})\tilde{p}]_{,\psi_\alpha} =$$

$$\{Mean\ convection\} \quad \{Chemical\ reactions\}$$

$$-[\bar{\rho} < u_i'' \mid \underline{\psi} > \tilde{p}]_{,x_i} - [\bar{\rho} < \frac{1}{\rho}J^{\alpha}_{i,x_i} \mid \underline{\psi} > \tilde{p}]_{,\psi_\alpha}$$

$$\{Turbulent\ convection\} \quad \{Molecular\ mixing\}$$

$$-[\bar{\rho} < \frac{1}{\rho}s_\alpha \mid \underline{\psi} > \tilde{p}]_{,\psi_\alpha}$$

$$\{Liquid - phase\ exchange\}$$

- $\tilde{p}$ = Density-weighted joint pdf.
- w_α = chemical source term for the α-th composition variable.
- $< u_i'' \mid \underline{\psi} >$ = conditional average of Favre velocity fluctuations.
- $< \frac{1}{\rho}J^{\alpha}_{i,x_i} \mid \underline{\psi} >$ = conditional average of scalar dissipation.
- $< \frac{1}{\rho}s_\alpha \mid \underline{\psi} >$ = conditional average of liquid-phase source term for the α-th composition variable.

Modeling Aspects of the Pdf Transport Equation

- $< u_i'' \mid \underline{\psi} >$ is modeled using a gradient-diffusion model.
- $< \frac{1}{\rho} J^{\alpha}_{i,x_i} \mid \underline{\psi} >$ is modeled using a variant of Curl's model.
- The new term $< \frac{1}{\rho} s_\alpha \mid \underline{\psi} >$ involving the conditional average of liquid-phase source term is modeled based on the average values of species and enthalpy:

$$< \frac{1}{\rho} s_\alpha \mid \underline{\psi} >= \frac{1}{\bar{\rho}\Delta v} \sum n_k m_k (\epsilon_\alpha - \phi_\alpha)$$

$$for \;\; \phi_\alpha = Y_\alpha, \alpha = 1, 2, ..., s = \sigma - 1$$

$$< \frac{1}{\rho} s_\alpha \mid \underline{\psi} >= \frac{1}{\bar{\rho}\Delta v} \sum n_k m_k (-l_{k,eff} + h_{ks} - \phi_\alpha)$$

$$for \;\; \phi_\sigma = h.$$

MODELING ASPECTS

- THE MODELED PDF TRANSPORT EQUATION PROVIDES THE SOLUTION FOR THE SPECIES AND TEMPERATURE FIELDS WITH THE MEAN VELOCITY AND THE TURBULENT DIFFUSIVITY AND FREQUENCY PROVIDED AS INPUTS FROM THE CFD SOLVER AND THE SPRAY SOURCE TERMS FROM THE LIQUID-PHASE SOLVER.
- 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–ε 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.

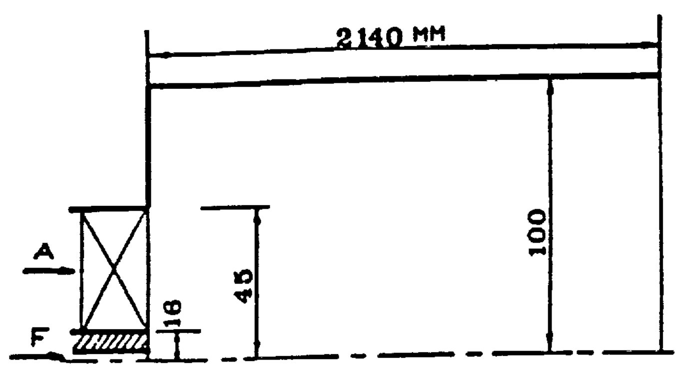
NUMERICAL METHOD

- 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:

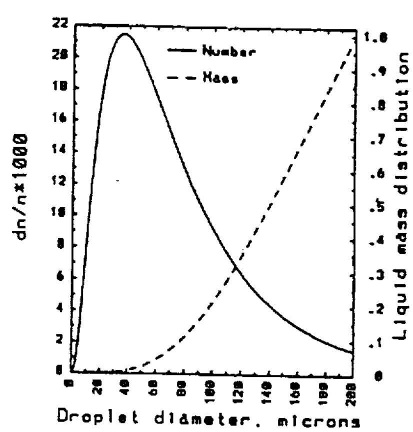
  inflow temperature = 310 K,
  air mass flow rate = 355 kg/h,
  air/fuel ratio =20.17,
  swirl vane angle = 45 deg,
  swirl number =0.721.

- 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:

$$\frac{dn}{n} = 4.21\ 10^6 \left[\frac{D}{D_{32}}\right]^{3.5} e^{-16.98\left(\frac{D}{D_{32}}\right)^{0.4}} \frac{dD}{D_{32}}$$

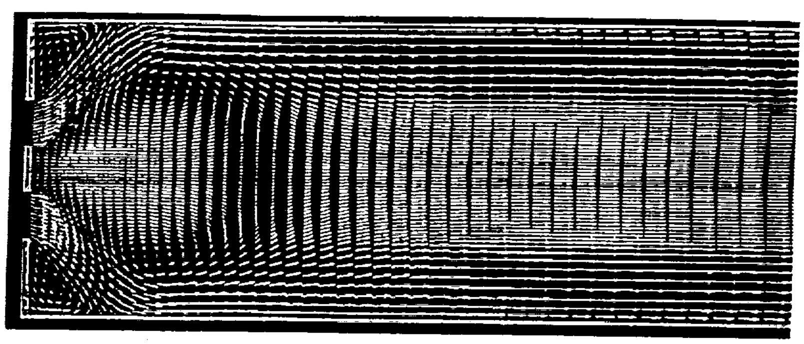


Droplet size Distribution

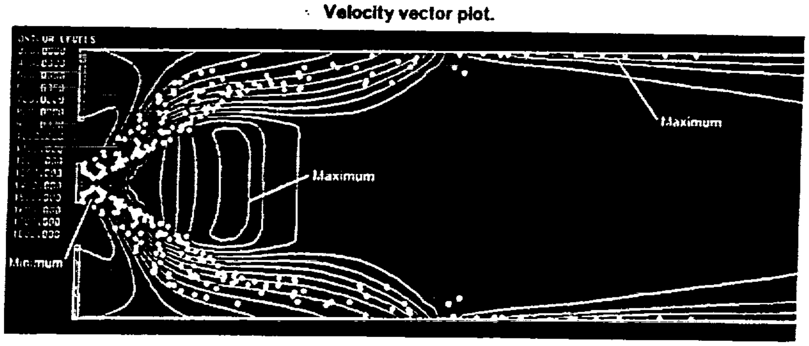
- The initial droplet injection velocity corresponds to: $u_k = 11.0$ m/s, $w_k = 6.1$, and $v_k = 0.5 - 2.5$.

PARAMETER SELECTIONS

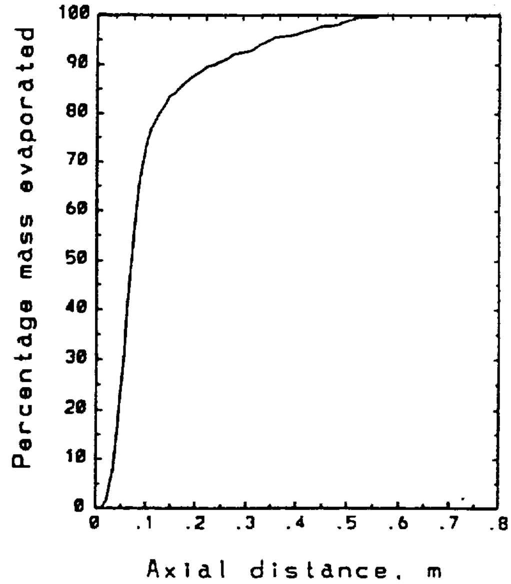
- The computations were performed on a grid with a mesh size of 60x60.
- The PDF solution is obtained by making use of 250 particles per cell.
- $Dt_g = Dt_{injection} = 1.5$ ms, $Dt_k = 0.0375$ ms, and $Dt_{Monte-Carlo} = 0.015$ ms.
- Two CPU seconds on a CRAY Y-MP per one Dt_g and about 2 to 3 CPU hours – 4000 time steps – for the solution to reach steady state.



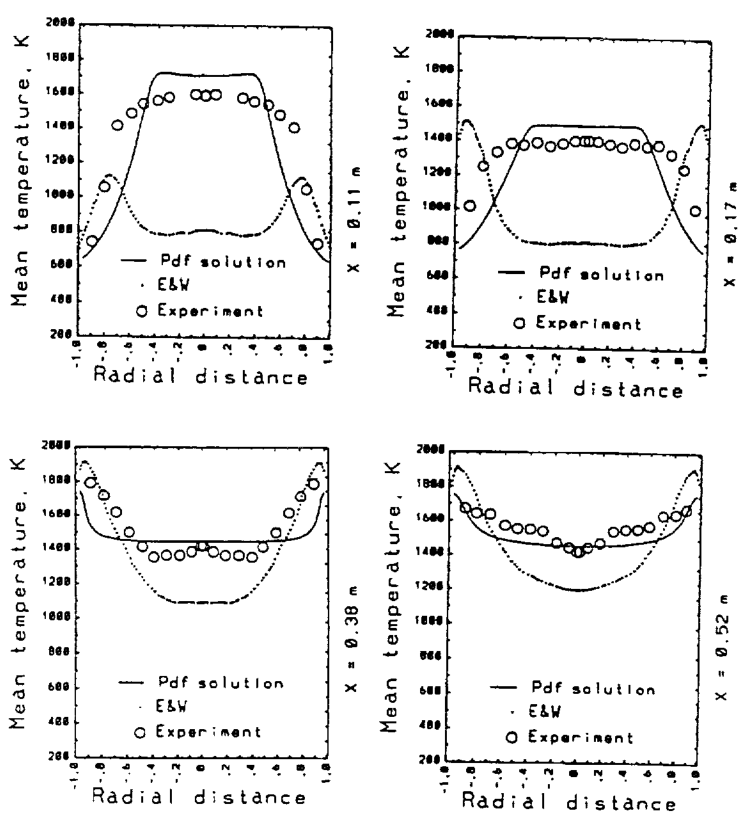
Velocity vector plot.



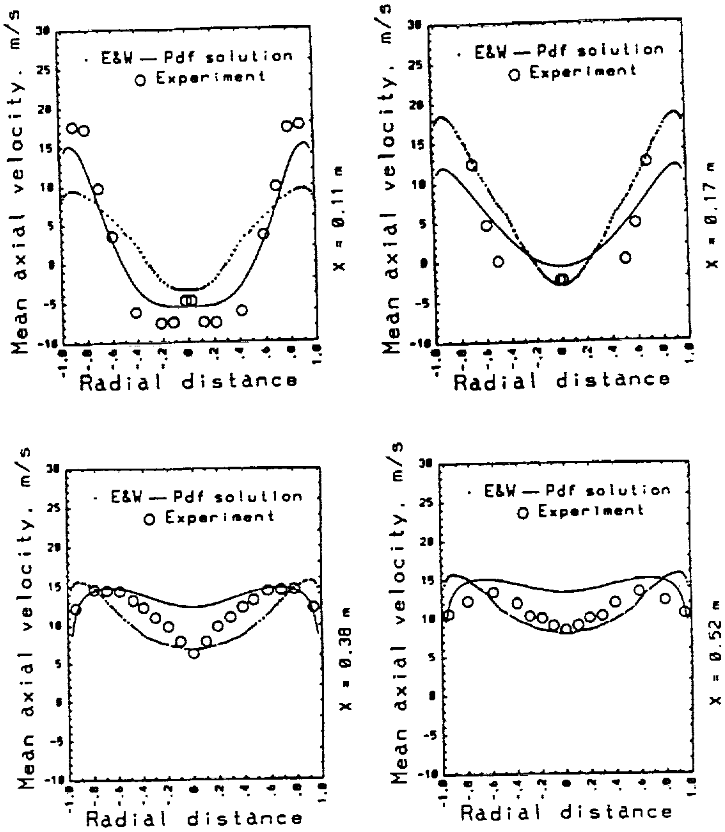
Temperature contours and droplet locations.



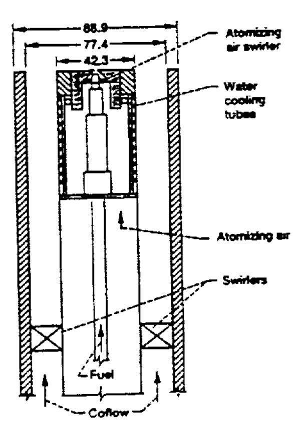
Spray evaporation rate.



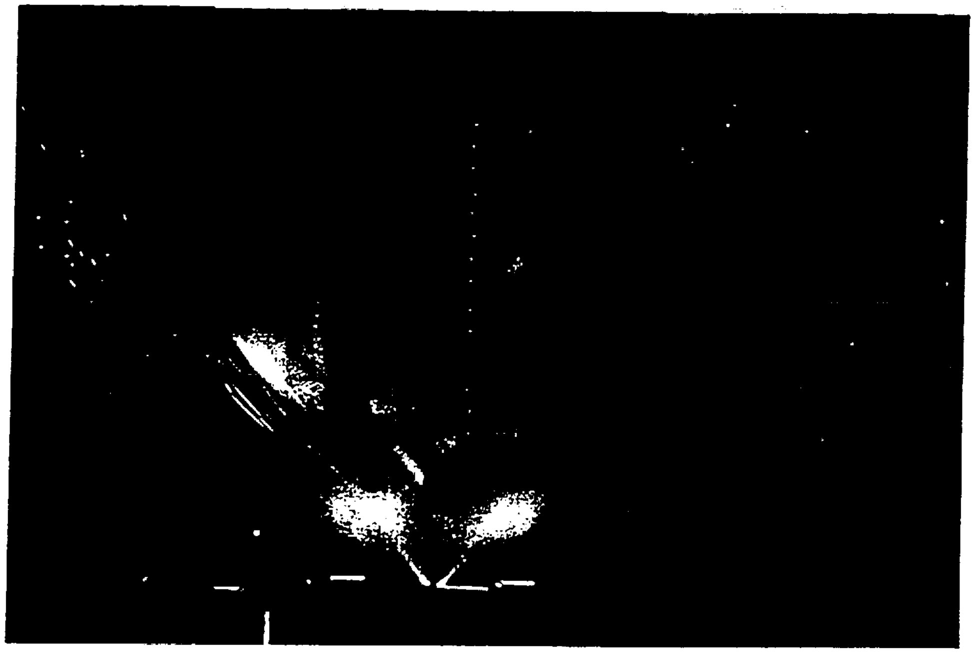
Near wake radial profiles of temperature.



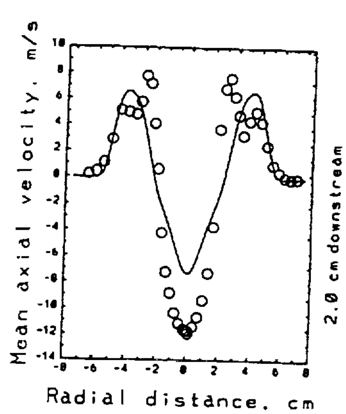
Near wake radial profiles of velocity.

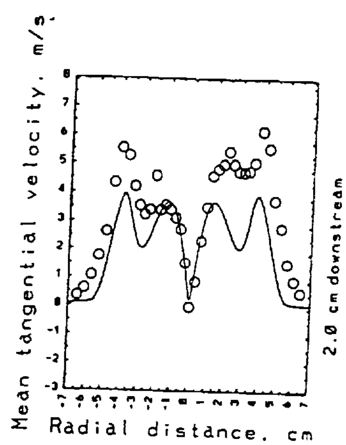


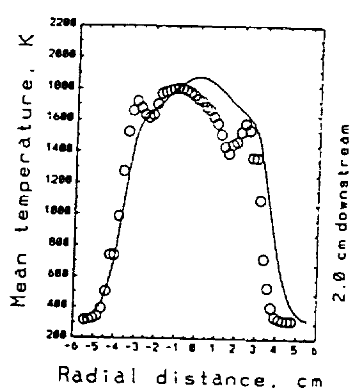
Schematic of an open spray burner.
(Dan Bulzan of IFMD at LeRC)

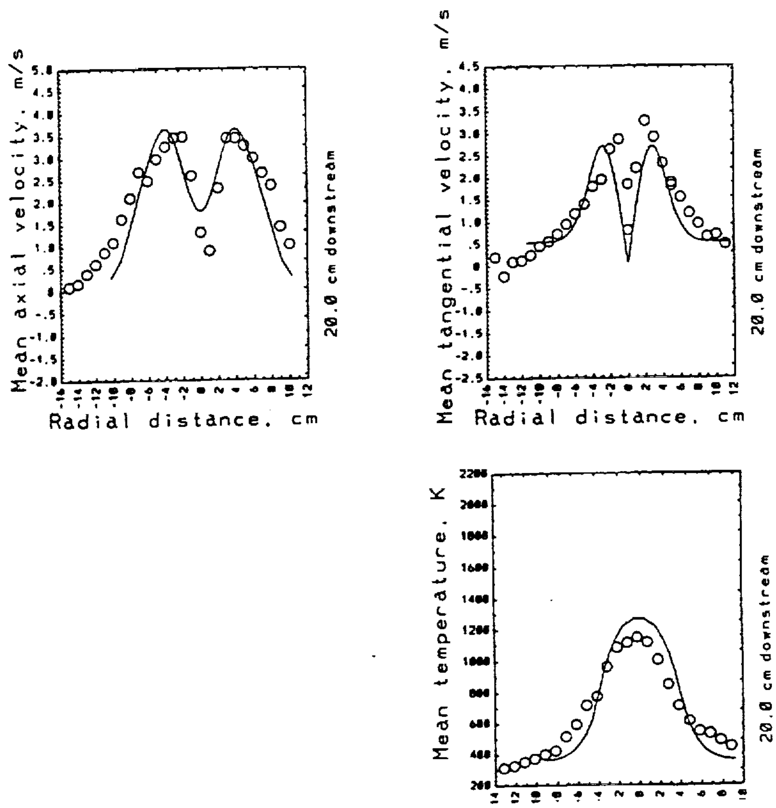


Photograph of swirl-stabilized, spray flame.









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-ε 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.

IMPROVEMENTS AND NEW FEATURES IN THE PDF MODULE

N95- 27902

A.T. Norris
Institute for Computational Mechanics in Propulsion
NASA Lewis Research Center
Cleveland, Ohio

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:

$$
\begin{aligned}
\frac{\partial}{\partial t}(\bar{\rho}P) \; &+ \; \frac{\partial}{\partial x_i}(\bar{\rho}\bar{U}_i P) + \frac{\partial}{\partial \phi_\alpha}(\bar{\rho}S_\alpha(\hat{\underline{\psi}}, p, \eta)P) \\
&= \; \frac{\partial}{\partial x_i}(\langle \bar{\rho}\tilde{u}_i | \underline{\psi}, \eta\rangle P) + \frac{\partial}{\partial \phi_\alpha}(\langle \frac{\partial J_i^\alpha}{\partial x_i} | \underline{\psi}, \eta\rangle P) \\
&+ \; \frac{\partial}{\partial \eta}(\langle \frac{\partial q_i^\alpha}{\partial x_i} | \underline{\psi}, \eta\rangle P) + \frac{\partial}{\partial \eta}(\langle \frac{Dp}{Dt} | \underline{\psi}, \eta\rangle P)
\end{aligned}
\tag{1}
$$

- 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.

$$
\langle \bar{\rho}\tilde{u}_i | \underline{\psi}, \eta\rangle P \approx D_t \frac{\partial P}{\partial \psi_\alpha} \tag{2}
$$

- 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 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 (eg. Sarkar).

$$\begin{aligned}\langle \frac{Dp}{Dt} | \underline{\psi}, \eta \rangle \approx \frac{\partial \langle p \rangle}{\partial t} &+ \langle U_i \rangle \frac{\partial p}{\partial x_i} + 0.8 \rho \langle k \rangle \frac{\partial \langle U_i \rangle}{\partial x_i} \\ &+ 0.15 \rho P_r M_t - 0.2 \rho \epsilon M_t^2\end{aligned} \tag{3}$$

• Advantages are that model is tried and tested in finite volume codes. Disadvantage is that only the mean pressure can be used for model. Idealy 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.

$$\text{if 6.3 particles then} \begin{cases} 6.0 & 70\% \text{ of the time} \\ 7.0 & 30\% \text{ of the time} \end{cases} \tag{4}$$

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 seperate 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 incriments. Users create their own table of composition incriments 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 recomended 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 recient values and less to those in the far past.

$$\langle\langle\phi\rangle\rangle_t = \frac{1}{w_t + 1}(\langle\phi\rangle_t + w_t\langle\langle\phi\rangle\rangle_{n-1}) \tag{5}$$

$$w_t = c_t(w_{t-1} + 1) \tag{6}$$

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. ie. no negative numbers of particles.

- Rplus/PDF release ported to workstation enviroment. K-epsilon now standard turbulence model.

Future Work.

- Release of 3D version with new improvements.

- Implimentation of parallel processing for distributed cluster environment. (PVM based)

- Include model for another state variable to close PDF modeling.

Attendees List

Name	Address	Phone/Fax	E-mail Address
Abrahim, Mounir	Cleveland State University Dept. of Mechanical Engineering Cleveland, Ohio	(216) 687-2580	
Ajmani, Kumud	ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142	(216) 433-3166 (216) 433-3200 (F)	fskumud@icomp01.lerc.nasa.gov
Anand, M.S.	Allison Engine Company P.O. Box 420, Speed Code T-14 Indianapolis, IN 46206	(317) 230-2828 (317) 230-3691 (F)	iemsa@agt.gmeds.com
Ashpis, David	NASA Lewis Research Center 21000 Brookpark Road Mail Stop 5-11 Cleveland, OH 44135	(216) 433-8317 (216) 433-5802 (F)	ashpis@lerc.nasa.gov
Befrui, Bizhan	ADAPCO 60 Broadhollow Road Melville, NY 11747	(516) 549-2300 (516) 549-2654 (F)	bizhan@adapco.com
Brankovic, Andreja	Pratt & Whitney P.O. Box 109600 Mail Stop 715-89 West Palm Beach, FL 33410	(407) 796-8811 (407) 796-5825 (F)	brankov@pwfl.com
Brewster, B. Scott	Brigham Young University 75 D CTB; P.O. Box 24212 Provo, Utah 84602-4212	(801) 378-6240 (801) 378-3831 (F)	brewster@homer.et.byu.edu
Burrus, David L.	GE Aircraft Engines 1 Neumann Way, MD A-404 Cincinnati, OH 45215	(513) 243-2611 (513) 243-2541 (F)	burrus@co348.ae.ge.com
Chen, J. Y.	University of California at Berkeley 6163 Etchererry Hall Berkeley, CA 94720	(510) 642-3286 (510) 642-6163 (F)	jychen@euler.berkeley.edu
Cheng, Chi-Yang	Dept. of Mechanical Engineering Grove City College Grove City, PA 16127	(412) 458-3367	
Chitsomboon, Tawit	ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142	(216) 962-3106 (216) 433-3200 (F)	
Choudhury, D.	Fluent Inc. 10 Cavendish Court, Lebanon, NH 03755	(603) 643-2600	
Choi, Dochul	UTRC East Hartford, CT	(203) 727-7791 (203) 727-7656 (F)	
Coirier, William	NASA Lewis Research Center 21000 Brookpark Road Mail Stop 5-11 Cleveland, OH 44135	(216) 433-5764 (216) 433-5802 (F)	coirier@lerc.nasa.gov
Dalton, Charles	University of Houston Deans Office, College of Engineering Houston, TX 77204-4814	(713) 743-4205 (713) 743-4214 (F)	dalton@uh.edu
Ganjoo, Deepak	Swanson Analysis Systems, Inc. P.O. Box 65, Johnson Road Houston, PA 15342-0065	(412) 873-3055 (412) 746-9494 (F)	dganjoo@swanson. com

Georgiadis, Nick	NASA Lewis Research Center 21000 Brookpark Road, M. S. 86-7 Cleveland, OH 44135	(216) 433-3958 (216) 433-2182 (F)	tonjg@scivax.lerc.nasa.gov
Giel, Paul W.	Nyma, Inc. 2001 Aerospace Parkway Brook Park, OH 44142	(216) 977-1340 (216) 977-1269 (F)	pwgiel@lerc.nasa.gov
Goldman, Louis J.	NASA Lewis Research Center 21000 Brookpark Road, M. S. 5-11 Cleveland, OH 44135	(216) 433-5845	
Hadid, A.	Rocketdyne, MS IB 39 6633 Canoga Avenue Canoga Park, CA 91303	(818) 718-3405	
Harloff, Gary	NYMA, Inc. 2001 Aerospace Parkway Brook Park, OH 44142	(216) 977-1337 (216) 977-1269 (F)	
Haroutunian, Vahé	Fluid Dynamics International 500 Davis Street, #600 Evanston, IL 60201	(708) 491-0200 (708) 869-6495 (F)	vahe@fdi.com
Hayder, Ehtesham	ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142	(216) 962-3146 (216) 962-3200 (F)	fshyder@icomp01.lerc.nasa.gov
Hunter, Scott	GEAE 1 Neumann Way, M/D A405 Cincinnati, OH 45215	(513) 243-0589 (513) 243-1343 (F)	hunter@c0164.ae.ge.com
Hsu, Andrew	NYMA, Inc. 2001 Aerospace Parkway Brook Park, OH 44142	(216) 977-1302	
Jorgenson, Philip	NASA Lewis Research Center 21000 Brookpark Road Mail Stop 5-11 Cleveland, OH 44135	(216) 433-5386 (216) 433-3000 (F)	aejorgen@zaphod.lerc.nasa.gov
Kral, Linda D.	McDonnell Douglas Aerospace P.O. Box 516, MC 106-4126 St. Louis, MO 63166	(314) 233-3591 (314) 777-1328 (F)	kral@mdcgwy.mdc.com
Lai, Ming-Chia	Wayne State University Mechanical Engineering Department Detroit, MI 48202	(313) 577-3893	lai@moon1.eng.wayne.edu
Leonard, Andy	CFD Research Corporation 3325 Triana Blvd. Huntsville, AL 35805	(205) 536-6576 (205) 536-6590 (F)	adl@cfd1.cfdrc.com
Liou, William	ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142	(216) 962-3152 (216) 962-3200 (F)	fswwl@icomp01.lerc.nasa.gov
Loh, C.Y.	University of Cincinnati M.L. #70 Cincinnati, OH 45221	(513) 556-1923 (513) 556-3231 (F)	cyloh@afiris.ase.uc.edu
Lumley, Prof J.	Sibley School of Mechanical and Aerospace Engineering, Cornell University Ithaca NY 14853	(607) 255-4050	
Mishra, Vinod	Department of Physics Case Western Reserve University Cleveland, OH 44106	(216) 368-4127	
Mularz, Edward	NASA Lewis Research Center 21000 Brookpark Road, M. S. 5-11 Cleveland, OH 44135	(216) 433-5850 (216) 433-3000 (F)	Edward.Mularz@lims.lerc.nasa.
Mueller, C. M.	Advanced Scientific Computing Ltd. 554 Parkside ve, Unit 4 Waterloo, Ontario Canada N2L 5Z4	(519) 886-8435	

Norris, Andrew	ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142	(216) 962-3071 (216) 962-3200 (F)	fsandy@icomp01.lerc.nasa.gov
Pal, Shankha	University of Cincinnati 2930 Scioto St., #802 Cincinnati, OH 45219	(513) 556-1923 (513) 556-3231	spal@uceng.uc.edu
Papp, John L.	University of Cincinnati 727 Martin Luther King Drive, 1012W Cincinnati, OH 45220	(513) 861-9520 (513) 556-3231	jpapp@uceng.uc.edu
Prakash, Chander	GE Aircraft Engines One Neumann Way Cincinnati, OH 45215	(513) 243-0788 (513) 243-1343 (F)	prakash@hr004.ae.ge.com
Raju, Manthena	NYMA, Inc. 2001 Aerospace Parkway Brook Park, OH 44142	(216) 977-1366	
Ramachandra, S.	Ram & Ram Research Associates 27743 Edgepark Boulevard North Olmsted, OH 44070	(216) 779-5886 (216) 979-9160 (F)	co395@cleveland.freenef.edu
Razdan, Mohan	Allison Engine Company P.O. Box 420, Speed Code T-14 Indianapolis, IN 46206	(317) 230-6404 (317) 230-3691 (F)	iemkr@agt.gmeds.com
Reddy, D.R.	NASA Lewis Research Center 21000 Brookpark Road Mail Stop 5-9 Cleveland, OH 44135	(216) 433-8133	
Richardson, Pamela	NASA Headquarters Washington DC 20546	(202) 358-4631	pf_richardson@aeromail.hq.nasa.gov
Schwab, John	NASA Lewis Research Center 21000 Brookpark Road, M. S. 5-11 Cleveland, OH 44135	(216) 433-8446 (216) 433-5802 (F)	jrschwab@lerc.nasa.gov
Shabbir, Aamir	ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142	(216) 962-3149 (216) 962-3200 (F)	fsaamir@icomp01.lerc.nasa.gov
Shieh, Geoffrey	Department of Mech & Aero Engineering SUNY at Buffalo Buffalo, NY 14260		
Shih, T.-H.	ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142	(216) 962-3161 (216) 962-3200 (F)	fsshih@icomp01.lerc.nasa.gov
Sindir, Munir	Rocketdyne, MS IB 39 6633 Canoga Avenue Canoga Park, CA 91303	(818) 586-1627 (818) 586-0588 (F)	
Singhal, Ashok	CFD Research Corporation 3325 Triana Blvd. Huntsville, AL 35805	(205) 536-6576 (205) 536-6590	aks@cfdrc.com
Smith, Brian	Lockheed - Fort Worth Company P.O. Box 748 Fort Worth, TX 76101	(817) 763-2836	
Sonnemeier, James	University at Buffalo 309 Jarvis Hall Amherst, NY 14260	(716) 645-2593 x2303 (716) 645-3875 (F)	jrs@santa.eng.buffalo.edu
Spalding, D. B.	CHAM UK Bakery House, 40 High Street, Wimbledon Village, London SW195AU UNITED KINGDOM	44-81-947-7651	
Steffen, Chris	NASA Lewis Research Center 21000 Brookpark Road Mail Stop 5-11 Cleveland, OH 44135	(216) 433-8508 (216) 433-5802 (F)	fssteff@faust.lerc.nasa.gov

Steinthorsson, Erlendur	ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142	(216) 962-3162 (216) 962-3200 (F)	fsstein@icomp01.lerc.nasa.gov
Sung, Chao-Ho	David Taylor Model Basin Bethesda, MD 20084-5000	(301) 227-1865 (301) 227-4589 (F)	
Syed, Saadat	Pratt & Whitney M.S. 715-89 P.O. Box 109600 West Palm Beach FL 33410-9600	(407) 796-3560 (407) 796-5825 (F)	
Taulbee, Dale B.	State University N.Y. at Buffalo Dept. Mech & Aerospace Engineering Buffalo, NY 14260	(716) 645-2593 x2307 (716) 645-3875 (F)	trldale@euler.eng.buffalo.edu
Tekriwal, P.	GE-CRD P.O. Box 8, Bldg. K-1, ES108 Schenectady, NY 12301	(518) 387-6732	
Tew, Roy	NASA Lewis Research Center 21000 Brookpark Road Mail Stop 301-2 Cleveland, OH 44135	(216) 433-8471 (216) 433-6133 (F)	tew@lerc.nasa.gov
Tselepidakis, Dimitri	Fluent Inc. 10 Cavendish Court Lebanon, NH 03766	(603) 643-2600 (603) 643-3967 (F)	dpt@fluent.com
To, Wai-Ming	NYMA 2001 Aerospace Parkway Mail Stop 5-9 Brook Park, OH 44142	(216) 433-5937 (216) 433-8864 (F)	fswmto@perch.lerc.nasa.gov
Tolpadi, Anil K.	General Electric R&D Center P.O. Box 8, M.S. K1-ES206 Schenectady, NY 12301	(518) 387-5787 (518) 387-7104 (F)	tolpadi@crd.ge.com
Tran, Le	NYMA, Inc. 2001 Aerospace Parkway Brookpark, OH 44142	(216) 977-1357 (216) 977-1269 (F)	totran@lerc.nasa.gov
Van Doormaal, J. P.	Advanced Scientific Computing Ltd. 554 Parkside ve, Unit 4 Waterloo, Ontario Canada N2L 5Z4	(519) 886-8435	
Wu, Jie	ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142	(216) 962-3096 (216) 962-3200 (F)	fsjiewu@icomp01.lerc.nasa.gov
Yang, Zhigang	ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142	(216) 962-3093 (216) 962-3200 (F)	fsyang@icomp01.lerc.nasa.gov
Yungster, Shaye	ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142	(216) 962-3162 (216) 962-3200 (F)	
Zerkle, Ronald	GE Aircraft Engines One Neuman Way Mail Drop A405 Cincinnati, OH 45215	(513) 243-2470 (513) 243-1343 (F)	
Zhu, Jiang	ICOMP/OAI 22800 Cedar Point Road Brook Park, OH 44142	(216) 962-3095 (216) 962-3200 (F)	fszhu@icomp01.lerc.nasa.gov

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY (*Leave blank*)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	March 1995	Conference Publication

4. TITLE AND SUBTITLE

Industry-Wide Workshop on Computational Turbulence Modeling

5. FUNDING NUMBERS

WU-505-90-5K

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

Compiler, A. Shabbir, Institute for Computational Mechanics in Propulsion, Lewis Research Center. Proceedings of a workshop sponsored by the Institute for Computational Mechanics in Propulsion and Center for Modeling of Turbulence and Transition (work funded by NASA Cooperative Agreement NCC3-233). ICOMP Program Director, Louis A. Povinelli, organization code 2600, (216) 433-5818.

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

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	