Assessment, Development, and Application of Combustor Aerothermal Models

J.D. Holdeman  
*National Aeronautics and Space Administration*  
*Lewis Research Center*  
*Cleveland, Ohio*

H.C. Mongia  
*Allison Gas Turbine Division*  
*General Motors Corporation*  
*Indianapolis, Indiana*

and  

E.J. Mularz  
*Propulsion Directorate*  
*U.S. Army Aviation Research and Technology Activity—AVSCOM*  
*Lewis Research Center*  
*Cleveland, Ohio*

Prepared for the  
33rd International Gas Turbine and Aeroengine Congress and Exposition  
sponsored by the American Society of Mechanical Engineers  
Amsterdam, The Netherlands, June 5-9, 1988
The gas turbine combustion system design and development effort is an engineering exercise to obtain an acceptable solution to the conflicting design trade-offs between: combustion efficiency, gaseous emissions, smoke, ignition, restart, lean blowout, burner exit temperature quality, structural durability, and life cycle cost. For many years, these combustor design trade-offs have been carried out with the help of fundamental reasoning and extensive component and bench testing, backed by empirical and experience correlations.

Recent advances in the capability of computational fluid dynamics (CFD) codes have led to their application to complex three-dimensional flows such as those in the gas turbine combustor. A number of U.S. Government and industry sponsored programs have made significant contributions to the formulation, development, and verification of an analytical combustor design methodology which will better define the aerothermal loads in a combustor, and be a valuable tool for design of future combustion systems. The contributions made by NASA Hot Section Technology (HOST) sponsored Aerothermal Modeling and supporting programs are described in this paper.

**ABSTRACT**

Empirically based procedures have led to successful evolutionary combustor improvements. However, these methods are experience-based, they are not well suited when combustor design requirements are significantly different from that of current technology engines. The rapidly developing CFD (Computational Fluid Dynamics) capability is providing an additional tool in the design process which can have a powerful positive influence on future design capability. In these codes, combustion system subcomponents including fuel injectors, and combustor liners, in addition to the complex internal flow, need to be accurately modeled. To achieve this, physical sub-models and accurate numerical schemes must be developed to describe the various aerothermochemical processes occurring within the combustion chamber.

A number of U.S. Government and company sponsored programs have made significant contributions to the formulation, development, and verification of an analytical combustor design methodology. These have included: U.S. Army Combustor Design Criteria Validation (Bruce et al., 1979; Mongia et al., 1979, Mongia and Reynolds, 1979), NASA Swirling Recirculating Flow (Srinivasan and Mongia, 1980), NASA Soot and NOx Emissions Prediction (Srivatsa, 1980), NASA Primary Zone Study (Sullivan et al., 1983), NASA Mass and Momentum Transfer (Johnson and Bennett, 1981; Roback and Johnson, 1983; Johnson et al., 1984), NASA Lateral Jet Injection (Lilley, 1986; Ferrell and Lilley, 1985; McMurray and Lilley, 1986; Ong and Lilley, 1986), NASA Dilution Jet Mixing (Srinivasan et al., 1982, 1984, 1985; Srinivasan and White, 1986; Holdeman et al., 1984; Holdeman and Srinivasan, 1986; Holdeman et al., 1987a), NASA Transition Mixing Study (Reynolds and White, 1986; Holdeman et al., 1987b), NASA HOST Aerothermal Modeling (Kenworthy et al., 1983; Sturgess, 1983; Srinivasan et al., 1983a, 1983b), NASA Error Reduction (Syed et al., 1985), Industry IR & D programs, and advanced combustor development programs.

The NASA Hot Section Technology (HOST) Combustion Program has supported several of these programs. The overall objective of the HOST Combustion Project is to develop and verify advanced analytical methods to improve the capability to design combustion systems.
for advanced aircraft gas turbine engines. This objective is being approached both computationally and experimentally.

Computationally, HOST first sponsored studies to assess and evaluate the capabilities of existing aerothermal models (circa 1982). Based on the results of these assessments and other studies in the literature, HOST supported several studies to develop new and improved numerical methods for the analysis of turbulent viscous recirculating flows, with emphasis on accuracy and speed of solution.

The objectives of HOST sponsored experimental studies were to improve understanding of the flow physics and chemistry in constituent flows, and to obtain fully-specified, benchmark-quality experimental data suitable for the assessment of the capabilities of advanced computational codes.

This paper reviews the advances in the state-of-the-art in combustor aerothermal modeling, while highlighting the programs supported by the HOST Project (Turbine Engine Hot Section Technology, 1982, 1984, 1985, 1986, 1987). Due to upper limitations not all programs that received HOST support are included, and, for completeness, some programs that made a significant contribution, but which did not draw their primary support from HOST are discussed.

AEROTHERMAL MODELING ASSESSMENT

Gas turbine combustion models include submodels of turbulence, chemical kinetics, turbulence/chemistry interaction, spray dynamics, evaporation/burnout, radiation, and soot formation and oxidation. A very extensive assessment of numerics, physical submodels, and the suitability of the available data was made by three contractors under Phase 1 of the HOST Aerothermal Modeling program (Kenworthy et al., 1983; Sturgess, 1983; Srinivasan et al., 1983a, 1983b). These investigations surveyed and assessed current models and identified model deficiencies through comparison between calculated and measured quantities. Results of the assessment by Srinivasan et al., (1983a, 1983b) are summarized by Mongia et al. (1986). The constituent flows examined included: (1) simple flows with no streamline curvature, (2) complex flows without swirl, and (3) complex flows with swirl. Geometries for several test cases from each of these categories are shown in Fig. 2.

k-c Turbulence Model

The k-c model is the simplest turbulence model that is suitable for recirculating flow calculations. This model achieves closure by using a gradient transport model for Reynolds stress with an isotropic eddy viscosity. For flows where the isotropic eddy viscosity assumption is not valid, the k-c model may be either modified (e.g., low Reynolds number correction, Richardson number correction) or replaced with an algebraic or differential Reynolds stress model.

Assessment of the k-c model(s) of turbulence showed that these models:

1. Correlation with data for complex swirling flows with recirculation zones
2. Predict trends correctly, for complex three-dimensional flows.

Algebraic Stress Model and Its Modifications

Mean flow predictions with the model agreed with the data as well as the k-c model results, therefore the conclusions above also apply to this model. In addition, the Algebraic Stress Model gives reasonable predictions for the Reynolds stress components, consistent with the strengths and limitations of the k-c models (Mongia et al., 1986).

The results of standard k-c and algebraic and differential Reynolds stress turbulence models, have been compared in several continuing assessment studies. An example comparison (Mongia, 1987) of data and calculations using a hybrid/SIMPLE numerical scheme is shown in Fig. 3. This flow is that of co-annular turbulent jets flowing into an axisymmetric sudden expansion (Roback and Johnson, 1983). In this figure, velocity profiles are shown at downstream, distance from 0.11 to 2.5 pipe diameters from the expansion.

Scalar Transport Model

Mongia et al. (1986) reported that the k-c model with specified Prandtl number predicts scalar fluxes reasonably well for flow where the gradient diffusion approximation is valid. An alternative, the algebraic scalar transport model, has the capability to improve predictions over the k-c approach, but further work is needed to establish its validity for swirling recirculating flows.

Turbulence/Chemistry Interaction Models

It was also concluded by Mongia et al. (1986) that both 2- and 4-step reaction schemes showed promise for application in gas turbine combustors, but need to be further validated against data from simple flames. The modified eddy breakup model predicted trends well, and it was recommended that it should be pursued because this approach could be easily extended to multistep kinetic schemes.

Numerical Accuracy

A significant deficiency identified in the assessments was that for many flows of interest the accuracy of the calculation was limited by the numerical approximations, wherein the false diffusion is of the same order of magnitude as the turbulent diffusion. This masked the differences between turbulence models such that very different models gave essentially the same result, and sometimes resulted in undeservedly good agreement between data and predictions.

False diffusion is present, the numerical solution obtained for any given flow depends on the grid density and distribution. An example of the comparisons made in the assessment program is given by the comparison in Figs. 4 and 5 between measured and calculated temperature distributions downstream from a row of jets entering a confined crossflow. This flow is a constituent flow in most gas turbine combustors, and has been treated extensively in the literature, including the recently completed NASA Dilation Jet Mixing program, from which data were compared with three-dimensional calculations in the Phase I assessment study by Srinivasan et al., (1983).
The calculated and experimental results show that for a single row of jets with an orifice spacing to diameter ratio, S/D = 2 injected into a ducted mainstream with a duct height to orifice diameter ratio H/D = 8. The jet-to-mainstream momentum flux ratio, J, for this test was 25.32. Calculations for this case made with 45x26x17 (1988) nodes, are shown in Fig. 4. The parameters plotted in these figures is the dimensionless mean temperature difference ratio, THETA, where \( \text{THETA} = (T_m - T_j)/(T_m - T_i) \). The predicted jet penetration and mixing are less than that shown by the data.

The calculation shown in Fig. 4 used 49 nodes to simulate each jet. It is generally not possible to use this many grid points in such a small region; as few as four may be used in practice for each jet. To simulate the accuracy of this approximation, calculations were performed for the same flow and geometric conditions, but with a 27x26x8 (5615) grid. These coarse-grid calculations (Fig. 5) are in much better agreement with the data than the fine-grid calculations. These and other calculations in Srinivasan et al., (1983b) clearly demonstrated that the three-dimensional calculations were not grid independent.

Conclusions from the Assessments

The major conclusion in the HOST Aerothermal Modeling Phase I assessment studies by Kenworthy et al. (1983), Sturgess (1983), and Srinivasan et al. (1983a, 1983b) was that the available computational fluid dynamics (CFD) codes provided a useful combustor design tool. Although significant advances have been made in the development and validation of multidimensional gas turbine combustion calculation procedures, the codes assessed were only qualitatively accurate, especially for complex three-dimensional flows, and further work was needed. It was concluded that both a significantly improved numerical scheme and fully-specified experimental data (i.e., both mean and turbulence field quantities, with measured boundary conditions) for complex non-reacting and reacting constituent flows were needed before various emerging physical sub-models of turbulence, chemistry, sprays, turbulence/chemistry interactions, soot formation/oxidation, radiation, and heat transfer could be properly assessed.

A SECOND GENERATION MODEL

The first generation combustor design procedure outlined by Mongia and Smith (1978) has been very useful for developing several combustors (Mongia et al., 1986) that exhibited significant technology advances. However, in addition to the model deficiencies identified in the assessments, there were several parameters of importance in gas turbine combustor design that the analytical models could not predict; e.g., gaseous emissions, soot formation, combustor pattern factor, and liner heat transfer. These parameters were, however, successfully predicted by well-established semi-analytical correlations developed by Plee and Mellor (1980), Lebrefevre (1985), and their associates. Therefore, a combustor design procedure that could be applied to current and future gas turbine engines was implemented that made use of empirical design concepts and employs analytical modeling tools to represent various combustion processes (Rizk and Mongia, 1986; Mongia, 1987).

This method makes use of multidimensional models to establish liner flowfield features and combustion characteristics. The analytical results are then integrated with semi-empirical correlations for performance parameters of interest. That is, flow field and geometric parameters that are needed in the empirical equations, such as combustion volume and the fraction of air participating in the primary combustion reaction, are provided by the analytical calculations. Satisfactory agreement with experimental data has been shown (Rizk and Mongia, 1986) for emissions, performance and heat transfer. The combustor for which data were available, and for which calculations were performed, is shown schematically in Fig. 6. A typical comparison between data and predictions for CO, unburned hydrocarbons, NOx, soot emissions, combustion efficiency, pattern factor, and lean blowout are shown in Figs. 7(a) to (g) respectively. The model is in good agreement with the data over the entire sea-level engine operating range. Calculated liner wall temperatures for both the inner and outer walls of this combustor are shown in Fig. 8 for three typical k-planes along k = 5, 14, and 23. Here k denotes nodal planes along the combustor circumferential direction. Although no direct comparison with liner wall temperature data was made, the predictions look reasonable.

AERO THERMAL MODELING PHASE II

Based on the recommendations of the Phase I assessment studies, activities in Phase II of the HOST Aerothermal Modeling program concentrated on developing improved numerical schemes, and collecting completely-specified data for nonreacting single and two-phase swirling and nonswirling flows. The programs initiated were: Improved Numerical Methods; Flow Interaction Experiment; and Fuel Injector/Air Swirl Characterization. The first of these is a prerequisite to further model development, and the data obtained in the latter two studies will be used to validate advanced models being developed independently.

Improved Numerical Methods

The hybrid finite differencing scheme employed in generally available combustor cod gives excessive numerical diffusion errors which preclude accurate quantitative calculations. In response to this deficiency, HOST supported three programs with the primary objective to identify, assess, and implement improved solution algorithms applicable to analysis of turbulent viscous recirculating flows. Both solution accuracy and solution efficiency were addressed (Turbine Engine Hot Section Technology, 1985, 1986, 1987; Turan and VanDoormal, 1987).

For most practical problems, a central differencing scheme would be advantageous if it were unconditionally stable. Central differencing is a simple second-order scheme which easily and straightforward to implement. However, for grid Peclet numbers larger than 2, central differencing can lead to over- and under-shoots and is unstable. The hybrid (central/upwind) scheme is stable for all Peclet numbers, but suffers from excessive false diffusion. An alternative scheme, named CONDIF (Controlled Numerical Diffusion with Internal Fst-hack) (Runchal et al., 1986) has unconditionally positive coefficients and still maintains the essential features of central differencing and its second-order accuracy.

CONDIF uses central differencing with modified central differencing scheme is used, otherwise upwind differencing is used. CONDIF employs just enough numerical diffusion to ensure stability based internally on the field distribution of.
the variable, rather than switching to upwind differ-
encing whenever Pe exceeds 2. Since upwinding is done
at relatively few grid points, CONDIF essentially main-
tains the second-order accuracy of central differen-
ting, and false diffusion is substantially reduced.

Another advanced numerical scheme, called flux-
spline (Patankar et al., 1987), is based on a linear
variation of total flux (convection + diffusion)
between two grid points. This is an improvement over
the assumption of uniform flux used in hybrid schemes,
and leads to reduced numerical diffusion.

Both of these schemes have been extended to solve a
variety of analytical, two-dimensional laminar and tur-
bulent flows (Runchal et al., 1987; Patankar et al.,
1987). As an example, results for a laminar flow
(Re = 400) in a square driven cavity are shown in
Fig. 9. This flow, shown schematically in part a), is
characterized by a strong recirculation zone typical
of many physical situations. The problem was solved
with both CONDIF and flux-spline schemes on a uniform
22x22 grid and compared with the exact analytical solu-
tion and a hybrid solution on an extremely fine 82x82
grid. Velocity profiles at the midsection of the cav-
ity are shown in Fig. 9(b). Both advanced schemes
show improvement over the hybrid calculation.

An attractive feature of both CONDIF and flux-
spline schemes is that their extension to three dimen-
sions is relatively straight-forward. The resulting
linear differential equations involve only seven points
as opposed to 27 points needed in many skewed-upwind
schemes (Syed et al., 1985).

In addition to the need for improved numerical
accuracy, there is a need for improved computational
efficiency for a given level of accuracy. Typically
the continuity and momentum equations are solved sepa-
ately, and then linked through iteration of the
pressure term; e.g. SIMPLE (Semi-Implicit Method for
Pressure Linked Equations). Modifications, such as
SIMPLER and PISO, have been shown to improve computa-
tional efficiency. Other advanced schemes (Turbine
Engine Hot Section Technology, 1985, 1986, 1987; Vanka,
1987), such as block correction techniques and direct
solution of the coupled equations have been proposed.
Calculations with the latter coupled with the flux-
spline technique have shown a speed increase by a fac-
tor of 15 for a calculation of turbulent flow over a
backward-facing step (Monga, 1987).

Gas Phase Experiments
An experimental study of the interactions between
the combustor and diffuser systems (Srinivasan and
Thorp, 1987) is in progress to:

1. Identify the mechanisms and magnitude of
aerodynamic losses in various sections of an
annular combustor-diffuser system
2. Determine the effects of geometric changes in
the prediffuser, dome, and shroud on these
losses
3. Obtain a data base to assess current and
advanced aerodynamic computer models for
predicting these complex flowfields
4. Upgrade the analytical models based on the
experimental data
5. Design and test advanced diffuser systems to
verify the accuracy of the upgraded analytical
model

Another study in progress will obtain comprehen-
sive mean and turbulence measurements of velocity and
species concentration in a three-dimensional flow model
of the primary zone of gas turbine combustor chambers
(Turbine Engine Hot Section Technology, 1985, 1986).
were made with a high-resolution spray patternator, a two-component laser velocimeter, and a single-component Phase/Doppler particle analyzer. The comprehensive experimental data generated in these programs will be used to validate advanced models of turbulence, scalar, and spray transport. Including two-equation turbulence models, algebraic and differential Reynolds stress models, scalar and scalar-velocity transport models, and Eulerian and Lagrangian deterministic and stochastic spray models.

**SUMMARY**

Although significant progress has been made in the development of three-dimensional analytical CFD codes and their application in future gas turbine combustor design, these codes are neither sufficiently comprehensive nor quantitatively accurate enough to permit a complete design alone. They are, however, a valuable component in an evolving combustor design methodology in which their capability is integrated with the substantial base of empirical experience and one-dimensional flow modeling.

**CONCLUDING REMARKS**

The NASA HOST sponsored Aerothermal Modeling Phase II programs will lead to significant improvements in our technical ability to predict nonreacting gas turbine combustor flow fields with and without spray injection. Significantly enhanced capabilities for accurately predicting combustor aerothermal performance and wall temperature levels and gradients will require further improvements in numerical schemes and physical submodels. It is equally important to collect fully-specified reacting flow data, similar to what is being done for nonreacting flows under HOST Phase II, for both complex constituent flows, and generic gas turbine combustors.

In parallel, work should continue in the formulation and systematic validation of turbulent combustion models for reacting sprays and multidimensional heat transfer models. These capabilities will provide the tools needed to analytically conduct the combustion trade-off studies so that optimum future combustion systems can be designed, fabricated, and developed within acceptable cost and schedule constraints.

**REFERENCES**


Turbine Engine Hot Section Technology (HOST) 1982 NASA TM-83022.


Turbine Engine Hot Section Technology 1984, NASA CP-2339.

Turbine Engine Hot Section Technology 1985, NASA CP-2405.

Turbine Engine Hot Section Technology 1986, NASA CP-2444.

Turbine Engine Hot Section Technology 1987, NASA CP-2493.

COMPRESSOR EXITS AND DIFFUSER

HIGH SWIRL AIR

DILUTION AIR JET

COMBUSTOR EXIT

AIR/REACTANT RECIRCULATION

FILM COOLING AIR

- FULLY 3-DIMENSIONAL FLOW
- CHEMICAL REACTION/HEAT RELEASE
- HIGH TURBULENCE LEVELS
- 2 PHASE WITH VAPORIZATION

FIGURE 1. - COMBUSTOR FLOW PHENOMENA.

CD-81-12820
Figure 2. Flows for which analytical model calculations were performed in Srinivasan et al. (1985).
FLOW IN A CURVED CHANNEL
SHIVA PRASAD AND RAMA PRIYAN (1978)

FLOW OVER A BACKWARD FACING PLANE STEP

FLOW OVER A RING IN A PIPE
PHATARAPHRUK AND LOGAN (1979)

SUDDEN PIPE-EXPANSION
MOON AND RUDINGER (1977)

FLOW THROUGH A SUDDEN EXPANSION IN A PIPE
JOHNSON AND BENNETT (1981)

OPPOSED JET COMBUSTOR
SCHIFFER AND SAWYER (1976)

AXISYMMETRIC COMBUSTOR WITH COALIRED FUEL AND AIR JETS
LEWIS AND SMOOT (1973)

FLOW BEHIND A BACKWARD FACING STEP
PITZ AND DAILY (1981)

(B) COMPLEX NONSWIRLING FLOWS.

FIGURE 2. - CONTINUED.
TWO COAXIAL JETS IN STAGNANT AIR
MORSE (1980)

\[ \frac{U_2}{U_1} = \frac{f_2}{f_1} = \frac{D_2}{D_1} \]

CONFINED SWIRL-DRIVEN FLOW
ALTGELD, ET AL (1983)

\[ U_2 = 41.2 \text{ m/s} \]
\[ U_1 = 34.3 \text{ m/s} \]
\[ f_2 = 0.42 \]
\[ f_1 = 0.7 \]
\[ D = 0.17 \text{ m} \]

AIR JET
\[ \text{SWIRL} \]
\[ \text{CORE} \]
\[ \text{GAP} \]

SWIRLING FLOW IN A PIPE EXPANSION
JANJUA ET AL (1982)

SWIRL COMBUSTOR WITH COOLING AIR
BRUM AND SAMUELSN (1982)

\[ U_2 = 0.089 \text{ m/s} \]
\[ \text{SWIRL} \]
\[ \text{AIR} \]
\[ \text{CO}_2 \]
\[ \text{INJECTION} \]
\[ \text{GAP} \]

\[ U_2 = 15 \text{ m/s} \]
\[ \phi_{0.865} = 5.7 \]
\[ \phi_{0.01} = 1.9 \]
\[ \phi_{0.089} = 8.0 \]

\[ \text{GAP} = 15.4 \text{ m/s} \]

ALL DIMENSIONS IN CM

FIGURE 2. - CONCLUDED.
Figure 3. Comparison of measured mean axial velocity profiles for coannular jets downstream of an axisymmetric sudden expansion, with calculations made using three turbulence models.
Figure 4. - Comparison between measured and calculated dimensionless temperature difference ratios downstream from a row of cool jets injected into a constant-temperature cross flow from the upper wall of a constant area duct (J = 25.32, S/H = 0.25, H/D = 8, 19,890 nodes.)
FIGURE 5. COMPARISON BETWEEN MEASURED AND CALCULATED DIMENSIONLESS TEMPERATURE DIFFERENCE RATIOS DOWNSTREAM FROM A ROW OF COOL JETS INJECTED INTO A CONSTANT-TEMPERATURE CROSS FLOW FROM THE UPPER WALL OF A CONSTANT AREA DUCT (J = 25.37, S/H = 0.75, H/D = 8, 5615 NODES).
FIGURE 6. - ANNULAR COMBUSTOR SCHEMATIC AND CALCULATION GRID CONFIGURATION.
FIGURE 7. - COMPARISON OF MEASURED AND PREDICTED PERFORMANCE AND EMISSIONS FOR COMBUSTOR IN FIG. 6.
\[ K = 23 \]
\[ K = 14 \]
\[ K = 5 \]

**Figure 8.** Calculated liner wall temperatures at maximum power condition for combustor in Fig. 6.
FIGURE 9. - CALCULATIONS OF LAMINAR FLOW IN A SQUARE (2-D) DRIVEN CAVITY.
FLOW CONTROL VALVE FOR EACH PRIMARY JET

TO EXHAUST FAN

PRIMARY JET FEED PIPE

60° SWIRLER

SWIRLER FEED PIPE WITH FLOW CONTROL VALVE

FIGURE 10. - TEST SECTION GEOMETRY FOR EXPERIMENTAL STUDY OF INTERACTION BETWEEN FLOW FROM MULTIPLE SWIRLERS AND TRANSVERSE JETS.
Figure 11. - Radial profiles of gas-and solid-phase mean flow components and particle number density at a particle-to-gas mass loading ratio of 1.0.
### Title and Subtitle
Assessment, Development, and Application of Combustor Aerothermal Models

### Author(s)
J.D. Holdeman, H.C. Mongia, and E.J. Mularz

### Performing Organization Name and Address
NASA Lewis Research Center
Cleveland, Ohio 44135-3191

and

U.S. Army Aviation Research and Technology Activity—AVSCOM
Cleveland, Ohio 44135-3191

### Abstract
The gas turbine combustion system design and development effort is an engineering exercise to obtain an acceptable solution to the conflicting design trade-offs between: combustion efficiency, gaseous emissions, smoke, ignition, restart, lean blowout, burner exit temperature quality, structural durability, and life cycle cost. For many years, these combustor design trade-offs have been carried out with the help of fundamental reasoning and extensive component and bench testing, backed by empirical and experience correlations. Recent advances in the capability of computational fluid dynamics (CFD) codes have led to their application to complex three-dimensional flows such as those in the gas turbine combustor. A number of U.S. Government and industry sponsored programs have made significant contributions to the formulation, development, and verification of an analytical combustor design methodology which will better define the aerothermal loads in a combustor, and be a valuable tool for design of future combustion systems. The contributions made by NASA Hot Section Technology (HOST) sponsored Aerothermal Modeling and supporting programs are described in this paper.

### Key Words (Suggested by Author(s))
Aerothermal modeling; Combustor models; Gas turbine combustors; Hot section technology

### Distribution Statement
Unclassified—Unlimited
Subject Category 07