Int. J. Heat Mass Transfer. Vol. 32, No. 6, pp. 1155–1164, 1989
Printed in Great Britain

0017–9310/89 $3.00+0.00

Study of three-dimensional gas-turbine combustor flows

W. SHYY
Department of Aerospace Engineering, Mechanics and Engineering Science, University of Florida, Gainesville, FL 32611, U.S.A.

M. E. BRAATEN
GE Research and Development Center, P.O. Box 8, Schenectady, NY 12301, U.S.A.

and

D. L. BURRUS
GE Aircraft Engine, Cincinnati, OH 45215, U.S.A.

(*Received* 25 *April* 1988 *and in final form* 22 *November* 1988)

Abstract—Both computational and experimental results are presented for studying the three-dimensional flow in an annular gas turbine combustor. The computational approach attempts to strike a reasonable balance to handle the competing aspects of the complicated physical and chemical interactions of the flow, and the requirements in resolving the three-dimensional geometrical constraints of the combustor contours, film cooling slots, and circular dilution holes. The algorithm employs non-orthogonal curvilinear coordinates, second-order accurate discretizations, multigrid iterative solution procedure, the standard k–ε turbulence model, and a combustion model comprising of an assumed probability density function and the conserved scalar variable formulation. To assess the performance of the numerical algorithm, three different annular combustor flows with in-house experimental measurements are investigated. Overall, it is found that good theory/data agreement of the characteristic temperature pattern in the exit plane can be obtained. The influence of changing the dilution hole arrangements on the combustor performance is well predicted. The complicated mixing process can be better understood with more detailed information supplied by the numerical simulation. It is concluded that for the normal operating condition where the physical process is likely to be dominant, the performance of a gas turbine combustor can be predicted by the present methodology.

INTRODUCTION

In a continuous combustion device such as a gas-turbine combustor, there exists a range of complex, interacting physical and chemical phenomena. Included are fuel spray atomization and vaporization, turbulent transport, finite-rate chemistry of combustion and pollutant formation, radiation and particulate behavior. Rigorous description of these phenomena are, however, either not available or require mathematical models which are too complex for computation, when taken together in the context of multi-dimensional flows. For these reasons, models of varying degrees of sophistication have been used depending on the particular questions being asked. The level of sophistication in the models is continuously increasing with improvements in numerical methods, computer capabilities and physical understanding.

An important factor in the selection of sub-models is 'computational tractability', which here is intended to mean that the differential or other equations needed to describe a sub-model should not be so computation-intensive as to preclude use in three-dimensional flow calculations. It is a factor in the selection of methodologies because calculations of complex flows (multi-component, three-dimensional, turbulent) in complex geometries (multiple inlet streams, highly-curved boundaries) are currently limited by computer capabilities. Previous studies, as reviewed by Jones and Whitelaw [1], have established that the 'mixed-is-burned' model (equilibrium chemistry) along with the k–ε eddy viscosity turbulence model and an assumed shape probability density function (pdf)/moment equations for scalar fluctuations, are computationally tractable in complex flows.

In the present paper, a methodology for computing steady turbulent combusting flow in combustors of complex shape, developed in the last several years, is briefly outlined. The approach taken here attempts to strike a reasonable balance to handle two competing aspects of the modeling work, namely, the complicated physical and chemical interactions of the flow, and the requirements in resolving the three-dimensional geometrical constraints of the combustor contours, film cooling slots, and circular dilution holes. A package of computer programs, called CONCERT, has been produced based on the algorithm developed. Three combustor flows with different contours, dilution hole arrangement, as well as aerodynamic characteristics, are selected to assess the predictive capability of the CONCERT code. By varying the arrangement of dilution holes with the same combustor wall contours, one can also investigate the resulting impact on the exit temperature pattern. Besides demonstrating qualitatively satisfactory agreement between theoretical predictions and experimental data in terms of the exit temperature profiles, detailed mixing and flow patterns can be supplied through the numerical simulations, which are critically needed by combustor designers.

COMPUTATIONAL ALGORITHM AND MODELS

The key elements of the numerical algorithm and turbulent combustion models embodied in the CONCERT code are listed in the following. The details can be found in ref. [2] and references cited there.

(1) Conserved scalar (with assumed pdf to account for variance effect) and fast chemistry approach for turbulence/chemistry interaction.

(2) Standard k–ε two-equation model with wall function treatment for turbulence effects.

(3) Zonal method for three-dimensional non-orthogonal grid generation which yields good control on the local geometrical variations (holes, slots) and produces a grid system with a unified index notation.

(4) Semi-implicit interactive algorithm solving strong conservation form of transport equations (mass, momentum, and other scalar fields) in general non-orthogonal curvilinear coordinates.

(5) Second-order finite difference operator for all terms, including convection, pressure, and diffusion effects.

(6) Multi-step predictor–corrector method for the pressure correction equation.

(7) Multigrid method (with either line or point method) for solving the system of linear equations resulting from the discretization procedure.

The contribution of the CONCERT algorithm is that, for the first time, a single phase gas-turbine combustor flow calculation can be conducted with a reasonable combustion model on the one hand, and a satisfactory numerical procedure on the other. Spray and thermal radiation aspects are neglected for the time being. This is considered acceptable for analysis at high power engine operating conditions. There have been intensive efforts devoted to this direction in the research community [3–5]. However, most works suffered from several shortcomings, notably the employment of inflexible grid distribution and low numerical accuracy. For example, despite recognizing that the adoption of the curvilinear coordinate system is crucial in yielding a successful predictive tool for combustor flow analysis, Priddin and Coupland [3] used the 'orthogonal' curvilinear coordinate system which fails either to resolve all of the geometrical complexities, such as a circular dilution holes or to adjust the grid according to the flow characteristics to reduce the numerical errors. Besides the work of Priddin and Coupland, other reported studies essentially are all still in the stage of conducting the reacting flow calculation based on the Cartesian/polar coordinate systems, which is even less adequate.

The details of the CONCERT algorithm cannot be possibly covered here. However, issues that have been discussed with some confusion, namely, the choice of the primary dependent variables and its relation to the grid arrangement, deserve some special attention. With the use of the curvilinear coordinates, either the original Cartesian velocity components or the transformed velocity components (covariant or contravariant), can be used as the primary variables. Here, a combined use of the Cartesian velocity components and contravariant velocity components is devised. In momentum equations, the Cartesian components are treated as the primary variables [6], while in the continuity equation the contravariant velocity components are first updated directly to satisfy the continuity equation and then so-called D'yakonov iteration is used to yield the corresponding values between the contravariant and Cartesian components [7]. A staggered grid system [6] is adopted so that the CONCERT algorithm can maintain the full strength once uncovered back to the Cartesian coordinate system. This approach has worked out satisfactorily, and contrary to the suggestion of Priddin and Coupland [3] and Wittig *et al.* [8], many highly curved flow problems, such as those in a diffuser with 90° turning have been computed with no fundamental difficulties [9].

In terms of the sub-models accounting for the turbulence and chemistry effects, measurements of the flow and turbulent transport characteristics of gas turbine combustors have been reported by Heitor and Whitelaw [10], who measured both isothermal and reacting flow (propane fuel) characteristics of a model can-type gas-turbine combustor. According to them, a scalar effective viscosity turbulence model (e.g. the 'k–ε model') should be adequate for the flow in the dilution section of the combustor since the production of turbulent kinetic energy is largely caused by the iteration of shear stress with shear strain. In the upstream recirculation zone, however, the flow is characterized by a large mean radial pressure gradient; an eddy-viscosity model would underpredict the turbulent energy levels, and hence, is unlikely to provide correct trends for turbulent transport of the scalar quantities. Despite its weaknesses, however, for lack of a better alternative, the two-equation class of turbulence models remains in widespread use.

There have not been many studies of the relevance of the combustion models in the context of practical combustion equipment, either experimentally or theoretically. According to the measurements of Heitor and Whitelaw [10], in a gas turbine combustor under normal conditions the combustion is controlled more by physical than chemical processes in the primary zone and a partial equilibrium model is likely to provide realistic predictions. Comprehensive accounts of the various turbulence models and the conserved scalar with assumed pdf combustion models have been given by Jones [11] and Bilger [12]. Suffice it to note here that the β-function was chosen to represent the pdf distribution; once the mean and variance of the conserved scalar is computed, both the Favre and Reynolds averaged temperature, as well as the mean density fields can be obtained from the convolution of the pdf.

RESULTS AND DISCUSSION

Flows in three different types of annular combustors have been computed. First, the results of flow in the GE/SNECMA CFM56 engine combustor are presented. The high pressure core system of this engine is currently in use on several aircrafts, including the Boeing 737-300. A similar comparison will be conducted for another production type engine, CF6-80C2, which powers the Boeing 747. The predicted exit temperature patterns for two different configurations of a research type combustor are also compared with the in-house experimental measurements. Together with the results presented earlier for a similar combustor, but with different flow conditions as well as dilution hole patterns [13] more broadly based observations can be made to assess the strength and limit of the current three-dimensional combustor flow predictions.

For a CFM56 combustor, a mesh system comprising of $53 \times 25 \times 37$ (total of 49 025) grid points along the axial, radial, and circumferential directions are constructed first. Since overall there are 20 swirl cups evenly placed in the combustor inlet, the complete flow information throughout the whole combustor can be obtained by applying the periodic boundary conditions of the single-cup sector of 18°. With each sector, there are five circular holes on both the top and bottom walls. There are also seven and six film cooling slots on the top and bottom surfaces, respectively. Representative views of the grid system and the inlet velocity pattern out of the swirl cup and splash plate are shown in Fig. 1. The combustor operating conditions, including the flow levels through the various combustor inlet streams, were determined from available test data. The inlet velocity data were taken using a five-hole yaw probe. The inlet gas temperature is 1396°R, and the mean conserved scalar value out of the swirl cup is 0.1289. The thermochemical data of CH_2 are used to represent the fuel property. Figure 2 shows the calculated velocity field in both the planes in line with the inlet swirl cup center, as well as midway between consecutive swirl cups. A blow-up of the dome region in line with the swirl cup center, reveals the recirculation zones formed by the interaction of the swirl cup flow and the primary dilution jets.

Figure 3 presents the calculated velocity and theory/data comparison of the temperature pattern distribution in the combustor exit plane. The strong vertical flows induced by the multiple jets are clearly seen and the predicted temperature pattern is characteristically agreeable to the measurement. The measured data were obtained from four arrays of seven thermocouple measurements between the top and bottom liners, rotated around the entire combustor exit annulus with a 1.5° interval. To illustrate the extremely complicated mixing process within the combustor, Fig. 4 depicts some representative streamlines of the mean velocity field. In Fig. 4(a) the streamlines are issued from the top and lower region of the inlet plane. The pattern is obviously extremely complicated. For example, it is evident that a large-scale mixing process is created by the primary jets which

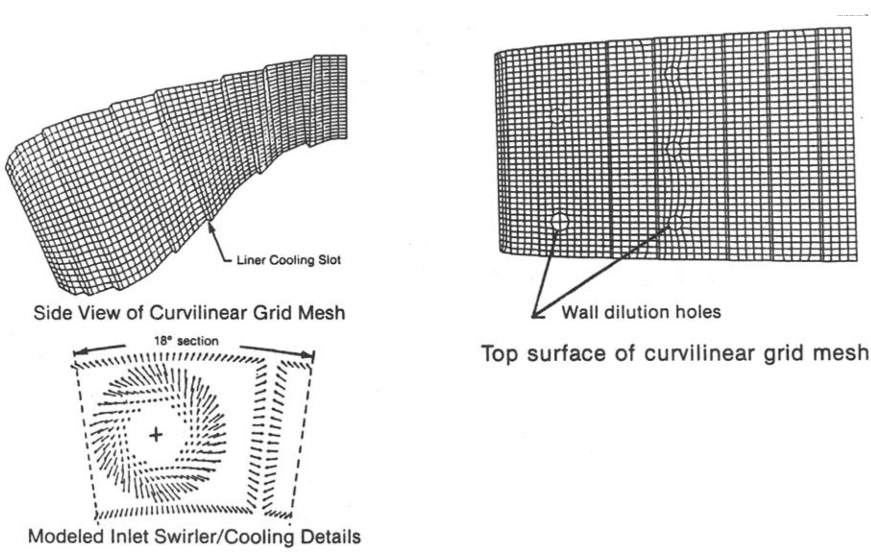

FIG. 1. Representative views of the grid system and the inlet velocity pattern of a CFM56 combustor.

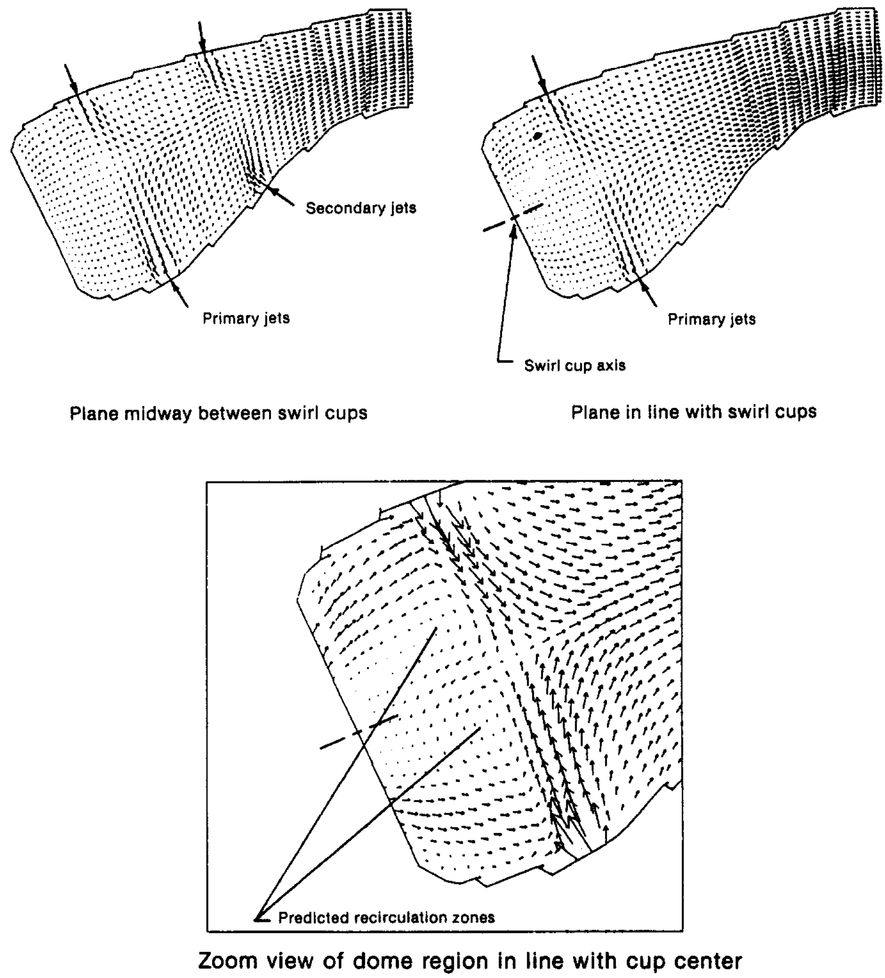

FIG. 2. Calculated velocity fields in side view planes.

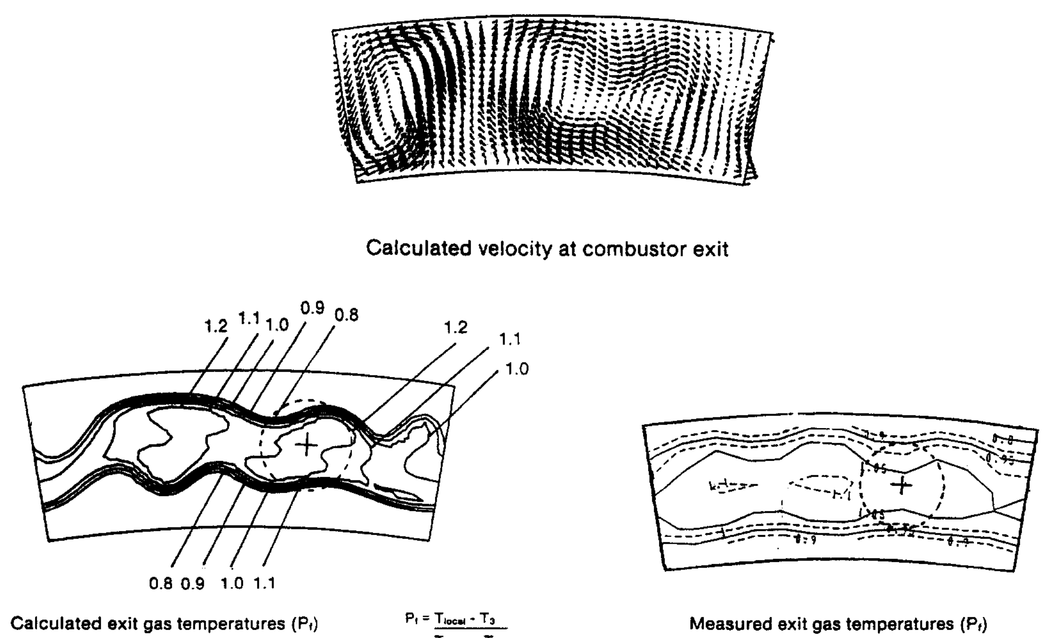

FIG. 3. Theory/data comparison in the combustor exit plane.

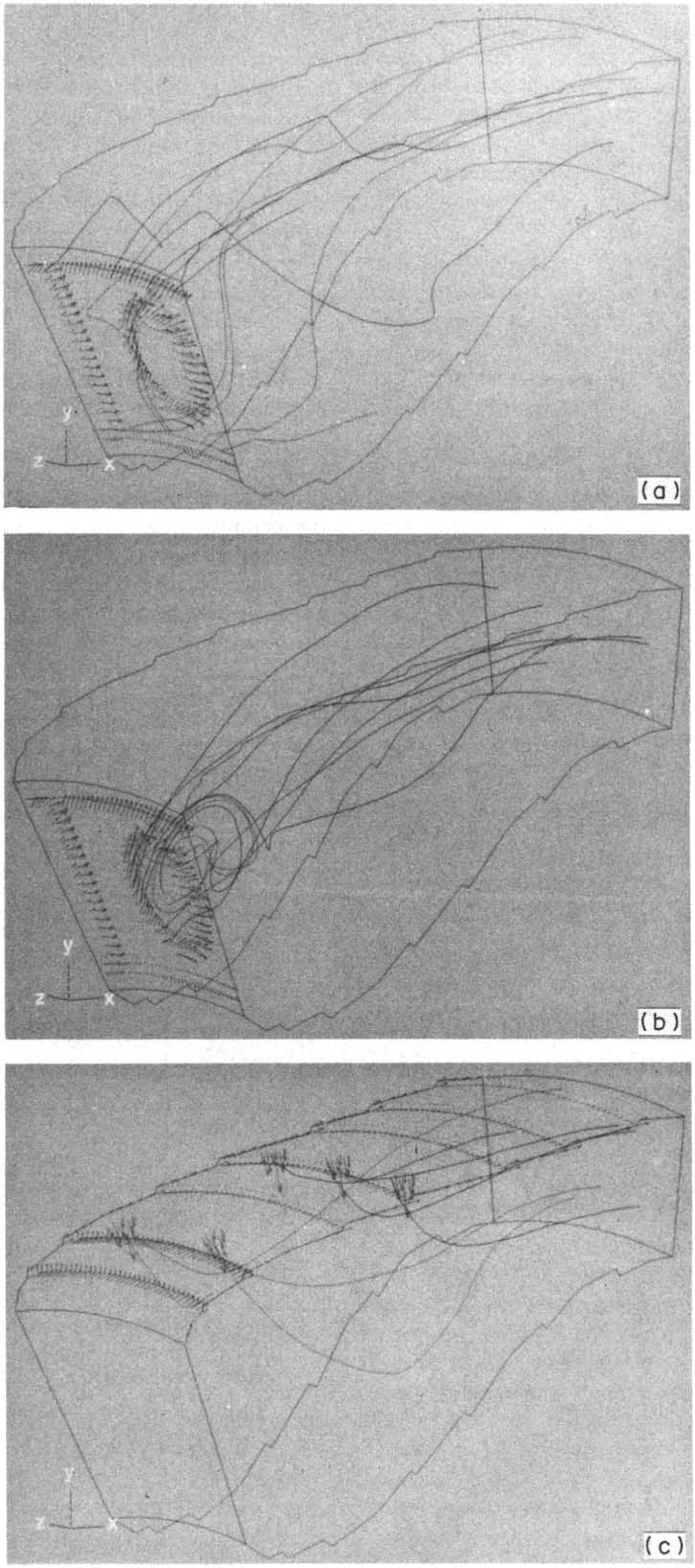

FIG. 4(a). Streamlines issued from the top and bottom of the main inlet. (b) Streamlines issued from the center of the main inlet. (c) Streamlines from dilution holes in the top wall.

can quickly exchange the fluid particles between the top and bottom wall regions. It is also evident that there is fluid exchange between adjacent sectors through the side boundary planes. Figure 4(b) shows the streamlines of the mean flow field issued from the centerline of the main inlet plane, where a recirculating zone located in the middle of the main dome can be clearly observed. Figure 4(c) shows the mean streamline issued from the primary and secondary dilution holes on the top wall. The differences between the incoming momentum of the two primary jets cause the depth of jet penetration to be considerably different. Collectively, Fig. 4 can help understand in a visually vivid manner the evolution of the key flow processes, such as fuel–air mixing, jet–main flow interactions, and the jet signatures in the exit plane.

Next, the theory/data comparison of the exit temperature pattern of the three-dimensional turbulent reacting flow in a GE CF6-80C turbofan engine combustor is presented. The combustor is annular in geometry and the calculation was performed for a single swirl-cup sector of 12° with the periodic boundary conditions being imposed on the two side planes. Within the computational domain, there are eight round holes of different sizes on both the top and bottom surfaces. A schematic of the combustor side view, the grid system (with $65 \times 21 \times 21$, or 34 125 grid points) and a theory/data comparison of the exit temperature profile are shown in Fig. 5, where $\bar{T}_4$ and $\bar{T}_3$ designate the overall averaged temperature in the exit and inlet of the combustor, respectively, and $\bar{T}_4(r)$ designates the circumferentially averaged temperature at each local radial position. The profiles shown in Fig. 5 have been averaged along the circumferential direction. The measured data were obtained from four arrays of seven thermocouple measurements between the top and bottom liners, rotated around the entire combustor exit annulus at 1.5° intervals. Very good agreement has been obtained for this extremely complicated flow.

The above cases are for production type gas turbine combustors. The third combustor studied is a research type where more extensive experimental information is available. Similar to above, 20 swirl cups are evenly placed along the annulus. A schematic illustration of the combustor side view is shown in Fig. 6. Two different configurations in the primary dilution pattern have been investigated. The first configuration has single opposed dilution holes in line with the swirl cup centers in each 18° sector. The second configuration has two opposed dilution holes in each 18° sector, one in-line and one midway between the swirl

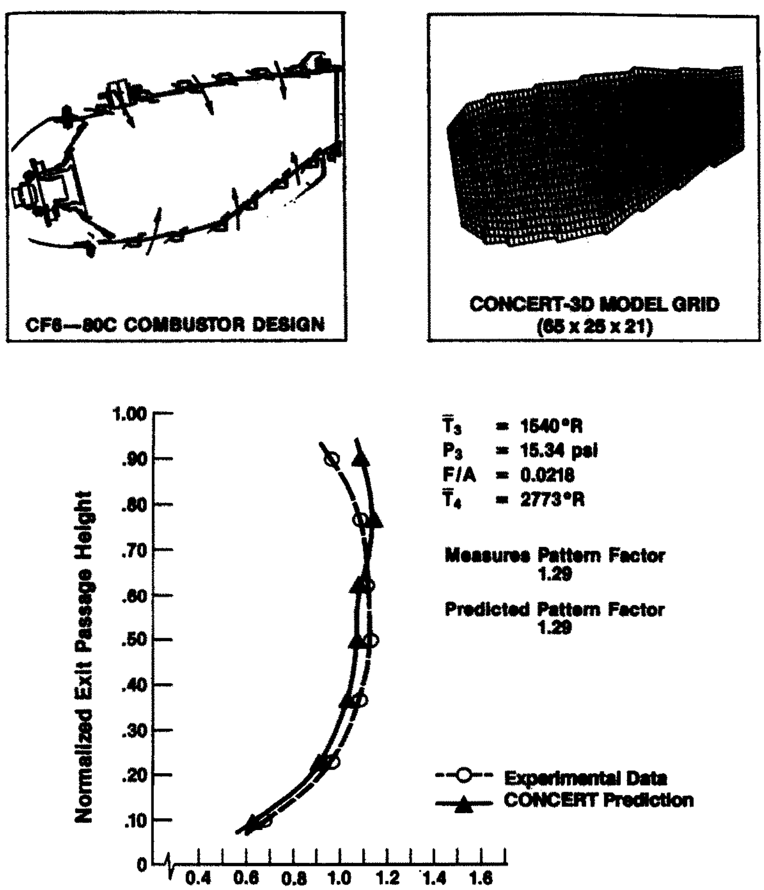

FIG. 5. Circumferentially-averaged normalized temperature $((\bar{T}_4(r)-\bar{T}_4)/(\bar{T}_4-\bar{T}_3))+1$.

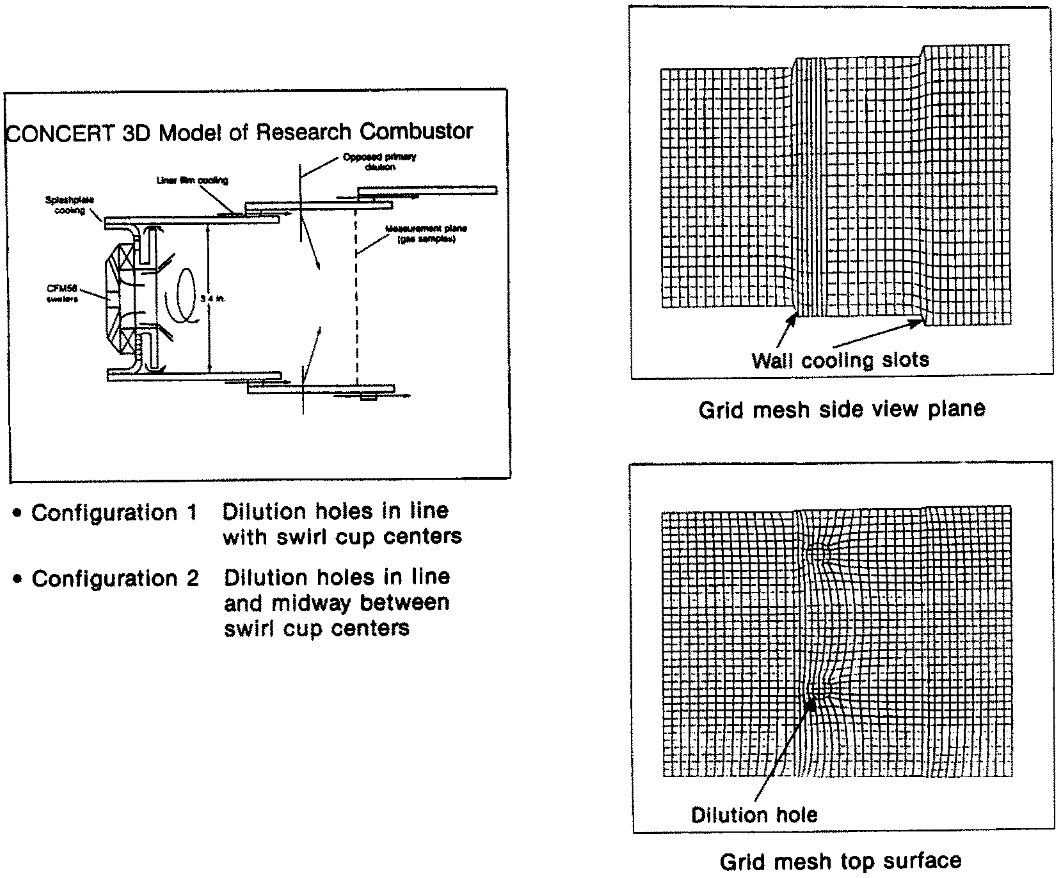

FIG. 6. Grid system of the research combustor configuration (41 × 25 × 27 points).

cup centers. Flow distributions were generated from measured data obtained on a calibrated flow stand. Gas temperature data were obtained from gas samples extracted at a plane just downstream of the primary holes. The inlet gas temperature was 1100°R for both cases modeled. The values of the mean conserved scalar out of the swirl cup are 0.0702 and 0.0783 for the two cases. A mesh system comprising of 45 × 25 × 27 (total of 41 625) grid points was generated, as shown in Fig. 6. Theory/data comparison at the measurement plane for both flow configurations is shown in Fig. 7. The agreement for both cases is good, both in terms of the hot/cold spot distribution, as well as the quantitative temperature levels. Figure 7 further demonstrates that the CONCERT algorithm is capable of correctly predicting the impact of the change of dilution hole pattern on the exit temperature pattern. This is the most critical information a designer is seeking in improving the performance of a combustor. The change of the temperature patterns in two configurations can be better understood by investigating the accompanied velocity field computed as a part of the solution, as shown in Fig. 8. The strong vortical flow motions caused by the interaction of main inlet flow and dilution jets can considerably enhance the mixing process and the chemical reactions. For both configurations, the cold spots are located at the positions outside the vortex cores. The introduction of the extra row of dilution holes produces a greater number of vortices in the exit plane creating more islands of higher temperature, especially in the lower half of the exit plane. It is noted that predicted temperature patterns generally show a larger degree of variation than those indicated by the measurements in all three cases studied here. This phenomenon can result from a combination of an under-prediction of the mixing process in the theoretical part and a lack of adequate spatial resolution in the experimental data.

CONCLUDING REMARKS

The results presented here suggest that for computing the complicated recirculating turbulent reacting flows in realistic combustor configurations, while virtually all of the modeling and computational aspects can be improved, the presently developed pragmatic approach, as embodied in the CONCERT algorithm, is capable of predicting characteristically correct temperature patterns. Detailed information yielded by the numerical solution can be used to help understand, explain, and parameterize the interactions among the many variables contained in the combustor

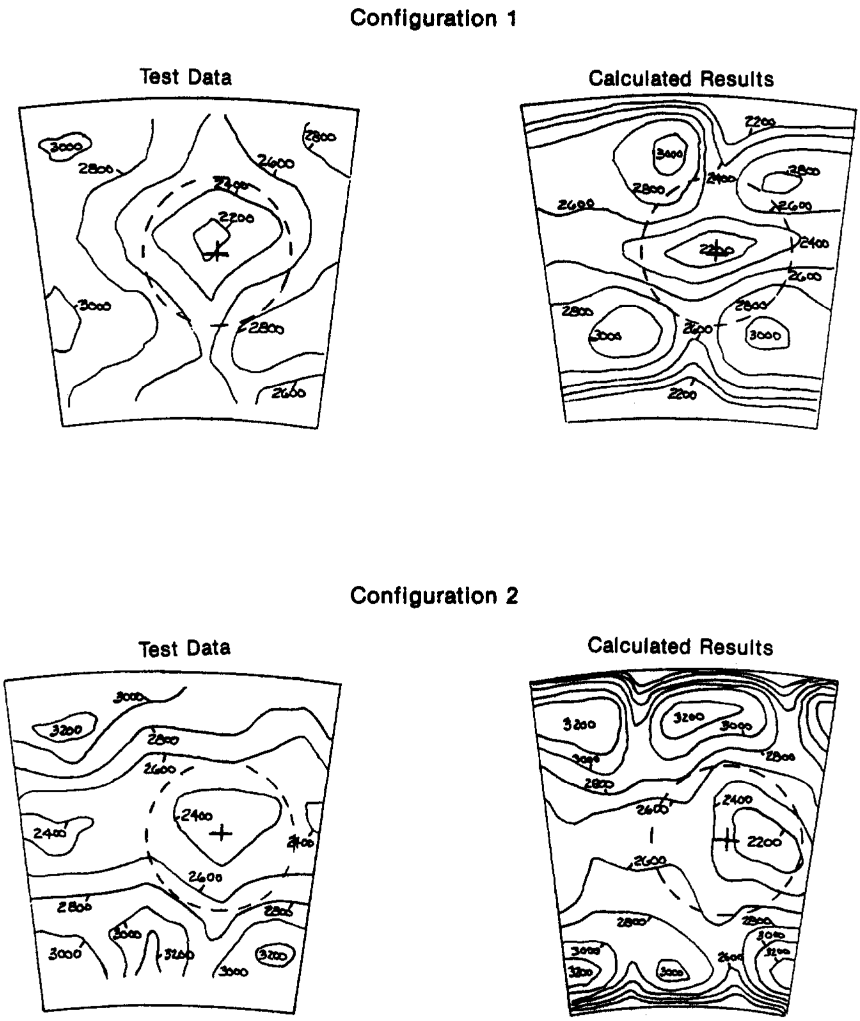

FIG. 7. Calculated vs measured gas temperatures (°F) at the measurement plane.

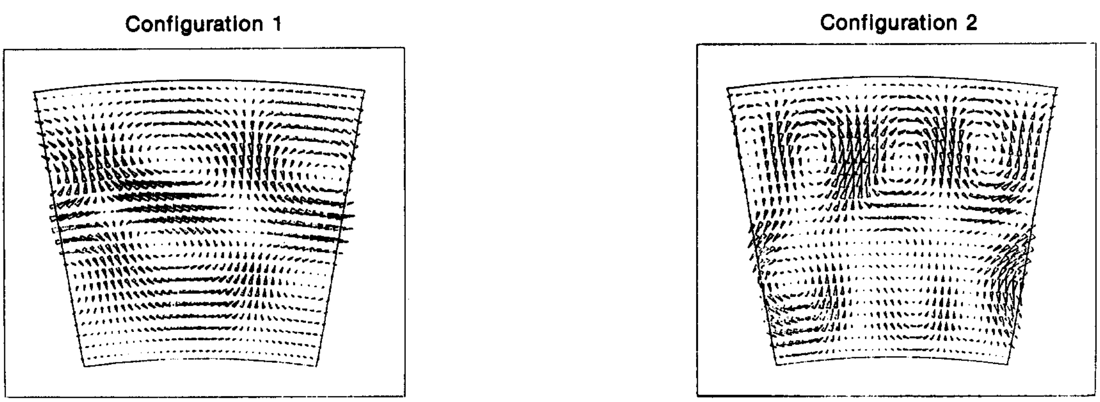

FIG. 8. Calculated velocity at the measurement plane.

flows. Accordingly, under the normal operating condition where the physical process is likely to be dominant, the major feature of combustor performance can be predicted, which is the conclusion of the present assessment.

Acknowledgements—Sincere thanks are due to Ms Kim Reksc and Ms Carol Cowsert for their expert typing. This work was conducted while W. Shyy was associated with GE Research and Development Center.

REFERENCES

1. W. P. Jones and J. H. Whitelaw, Modeling and measurements in turbulent combustion. Twentieth Symp. (Int.) on Combustion, pp. 233–249. The Combustion Institute, Pittsburgh, Philadelphia (1984).
2. W. Shyy, W. M. Correa and M. E. Braaten, Computational models for gas-turbine combustors. In *Calculation of Turbulent Reactive Flows* (Edited by R. M. C. So, J. H. Whitelaw and H. C. Mongia), pp. 141–183. ASME, New York (1986).
3. C. H. Priddin and J. Coupland, Impact of numerical methods on gas turbine combustor design and development. In *Calculation of Turbulent Reactive Flows* (Edited by R. M. C. So, J. H. Whitelaw and H. C. Mongia), pp. 335–348. ASME, New York (1986).
4. F. Joos and B. Simm, Comparison of the performance of a reverse flow annular combustion chamber under low and high pressure conditions. AGARD Propulsion and Energetics 70th Symp., AGARD CP No. 422 (1987).
5. M. J. Kenworthy, S. M. Correa and D. L. Burrus, Aerothermal modeling phase I, NASA CR-168296 (1982).
6. W. Shyy and S. S. Tong, Numerical recirculating flow calculation using a body-fitted coordinate system, *Numer. Heat Transfer* **8**, 99–113 (1985).
7. M. E. Braaten and W. Shyy, A study of recirculating flow computation using body-fitted coordinates: consistency and mesh skewness, *Numer. Heat Transfer* **9**, 554–574 (1986).
8. S. Wittig, H.-J. Bayer and B. Noll, On the application of finite-difference technique for the computation of the flowfield in gas turbine combustor with complex geometries. AGARD Propulsion and Energetics 70th Symp., AGARD CP No. 422 (1988).
9. W. Shyy and M. E. Braaten, Three-dimensional analysis of the flow in a curved hydraulic turbine draft tube, *Int. J. Numer. Meth. Fluids* **6**, 861–882 (1986).
10. M. V. Heitor and J. M. Whitelaw, Velocity, temperature, and species characteristics of the flow in a gas-turbine combustor, *Combust. Flame* **64**, 1–32 (1986).
11. W. P. Jones, Models for turbulent flows with variable density. In *Prediction Methods for Turbulent flows* (Edited by W. Kollmann), pp. 379–421. Hemisphere, Washington, DC (1980).
12. R. W. Bilger, Turbulent flows with non-premixed reactant. In *Turbulent Reacting Flows* (Edited by P. A. Libby and F. A. Williams), p. 65. Springer, Berlin (1980).
13. W. Shyy, S. M. Correa and M. E. Braaten, Computation of flow in a gas turbine combustor, *Combust. Sci. Technol.* **58**, 97–117 (1988).

ETUDE DES ECOULEMENTS TRIDIMENSIONNELS DANS LES CHAMBRES DE COMBUSTION DE TURBINE A GAZ

Résumé—On présente les résultats expérimentaux et numériques de l'étude de l'écoulement tridimensionnel dans une chambre annulaire de combustion d'une turbine à gaz. L'approche numérique tente de dresser un bilan des aspects compétitifs des interactions complexes physiques et chimiques et de tenir compte dans la résolution, des contraintes géométriques tridimensionnelles dues au contour de la chambre, des fentes de refroidissement et des trous circulaires de dilution. L'algorithme emploie des coordonnées curvilignes non orthogonales, des discrétisations précises de second ordre, une procédure de résolution itérative multigrille, le modèle de turbulence k–ε, et un modèle de combustion renfermant une fonction de densité de probabilité. Pour vérifier la qualité de l'algorithme numérique, on expérimente sur trois écoulements annulaires différents. Globalement on constate un bon accord entre théorie et expérience pour la configuration de température dans le plan de sortie. L'influence d'un changement dans l'arrangement des trous de dilution est bien prédite. Le mécanisme compliqué du mélange peut être mieux compris à partir de la simulation numérique qui donne une information plus détaillée. On conclut que dans les conditions normales où le mécanisme physique est prédiminant, la performance de la chambre de combustion d'une turbine à gaz peut être prédite par la présente méthodologie.

UNTERSUCHUNG DER DREIDIMENSIONALEN STRÖMUNG IN EINER GASTURBINEN-BRENNKAMMER

Zusammenfassung—Es werden sowohl Berechnungs- als auch experimentelle Ergebnisse zur Untersuchung der dreidimensionalen Strömung in der ringförmigen Brennkammer einer Gasturbine vorgestellt. Der rechnerische Ansatz versucht eine brauchbare Bilanz aufzustellen, um die konkurierenden Aspekte einerseits der komplizierten physikalischen und chemischen Wechselwirkungen der Strömung zu berücksichtigen, und andererseits die Erfordernisse bei der Beschreibung der dreidimensionalen Kontur der Brennkammer, der Film-Kühlungsschlitze und der kreisförmig angeordneten Verdünnungsbohrungen. Der Algorithmus benutzt nicht-orthogonal gekrümmte Koordinaten, eine Diskretisierung zweiter Ordnung, einen mehrfach iterativen Lösungsweg, das Standard k–ε Turbulenz–Modell und ein Verbrennungsmodell, das eine angenommene Wahrscheinlichkeits-Dichtefunktion enthält. Um die Leistungsfähigkeit des numerischen Verfahrens abzuschätzen, wurden drei verschiedene ringförmige Verbrennungsströmungen experimentell im Labor untersucht. Generell wurde herausgefunden, daß eine gute Übereinstimmung zwischen Theorie und Meßdaten besteht bezüglich des charakteristischen Temperaturfeldes in der Auslaßebene. Der Einfluß eines Wechsels der Anordnung der Verdünnungsbohrungen auf die Brennkammerleistung wird gut vorhergesagt. Kompliziertere Mischungsvorgänge können mit mehr detaillierteren Informationen aus der numerischen Simulation besser verstanden werden. Abschließend kann gesagt werden, daß die Leistung einer Gasturbinen-Brennkammer durch das vorliegende Verfahren für normale Betriebsbedingungen, wo die physikalischen Vorgänge ziemlich dominierend sind, bestimmt werden kann.

ИССЛЕДОВАНИЕ ТРЕХМЕРНЫХ ТЕЧЕНИЙ В ГАЗОТУРБИННОЙ КАМЕРЕ СГОРАНИЯ

Аннотация—Представлены расчетные и экспериментальные данные исследования трехмерного течения в кольцевой газотурбинной камере сгорания. При помощи вычислительного метода предпринята попытка нахождения достаточного равновесия для изучения конкурирующих аспектов сложных физических и химических взаимодействий течения, а также требований к решению трехмерных геометрических связей контуров камеры сгорания, каналов для охлаждения пленок жидкости и круглых отверстий для разбавления. В алгоритме используются неортогональные криволинейные координаты, дискретизации с точностью до второго порядка, многосеточный метод итераций, стандартная турбулентная модель k–ε и модель сгорания, включающая предполагаемую форму функции плотности вероятности и концепцию консервативной скалярной переменной. Для оценки характеристик численного алгоритма исследованы три различных вида течения в кольцевых камерах сгорания с измерениями течения внутри корпуса. В целом найдено, что можно получить хорошее согласование теоретических и экспериментальных данных для характерного температурного режима в выходной плоскости. Рассчитано влияние изменения расположения отверстий для разбавления на режим работы камеры сгорания. Сложный процесс смешения может быть лучше понят на основе более подробной информации, полученной при численном моделировании. Сделан вывод, что данная методология позволяет рассчитать рабочие характеристики газотурбинной камеры сгорания при нормальном режиме работы.