Check for updates

THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS
345 E. 47th St., New York, N.Y. 10017

98-GT-184

The Society shall not be responsible for statements or opinions advanced in papers or discussion at meetings of the Society or of its Divisions or Sections, or printed in its publications. Discussion is printed only if the paper is published in an ASME Journal. Authorization to photocopy for internal or personal use is granted to libraries and other users registered with the Copyright Clearance Center (CCC) provided $3/article or $4/page is paid to CCC, 222 Rosewood Dr., Danvers, MA 01923. Requests for special permission or bulk reproduction should be addressed to the ASME Technical Publishing Department.

Copyright © 1998 by ASME All Rights Reserved Printed in U.S.A.

CFD MODELING OF A GAS TURBINE COMBUSTOR FROM COMPRESSOR EXIT TO TURBINE INLET

D. Scott Crocker, Dan Nickolaus, and Clifford E. Smith
CFD Research Corporation
Huntsville, Alabama

ABSTRACT

Gas turbine combustor CFD modeling has become an important combustor design tool in the past few years, but CFD models are generally limited to the flow field inside the combustor liner or the diffuser/combustor annulus region. Although strongly coupled in reality, the two regions have rarely been coupled in CFD modeling. A CFD calculation for a full model combustor from compressor diffuser exit to turbine inlet is described. The coupled model accomplishes two main objectives: 1) implicit description of flow splits and flow conditions for openings into the combustor liner, and 2) prediction of liner wall temperatures. Conjugate heat transfer with nonluminous gas radiation (appropriate for lean, low emission combustors) is utilized to predict wall temperatures compared to the conventional approach of predicting only near wall gas temperatures. Remaining difficult issues such as generating the grid, modeling swirler vane passages, and modeling effusion cooling are also discussed.

INTRODUCTION

CFD modeling of combustors and fuel nozzles has progressed to the extent that CFD simulations are frequently included in the design process. Thus far, CFD models have generally been limited to isolated parts of the combustion system. Most models include only the reacting flow inside the combustor liner with assumed profiles and flow splits at the various liner inlets. Carefully executed models of this type can provide valuable insight into mixing performance, pattern factor, emissions and combustion efficiency. Some insight into liner wall temperatures can also be indirectly derived from the near wall gas temperatures. Perhaps the most important benefit of CFD modeling is the innovative ideas that CFD solutions sometime inspire.

There are numerous examples of combustor CFD solutions in the literature of which Lawson (1993), Fuller and Smith (1993) and Lai (1997) are a representative sample. Lawson was able to successfully match exit radial profile experimental data and then use the anchored CFD model to predict the radial profile that resulted from different cooling and dilution air patterns. Lawson used a 1-D code to predict flow splits and a 2-D CFD model to predict the flow profile at the exit plane of the swirl cup. This profile was then applied as a boundary condition in the 3-D model. Fuller and Smith were able to predict exit temperature profiles of an annular through-flow combustor that were in fairly good agreement with measurements. They also used a 2-D model to provide boundary conditions at the exit plane of the swirl nozzle. They did introduce the use of multi-block grids which are essential for modeling complex geometries. Lai was able to predict hot spots based on near wall gas temperatures that corresponded to locations in the combustor that had experienced deterioration. Lai included the swirler passages in his model which is an important step in reducing the uncertainty of prescribed boundary conditions. Although Lai modeled the swirler passages and the rest of the combustor with a single block grid, it is generally much easier and more efficient to use multi-block grids.

CFD models of the diffuser and annulus region have also been performed such as those of Srinivasan, *et al.* (1990), Karki, *et al.* (1992) and Little and Manners (1993). All three of these efforts at modeling the flow field outside of the combustor liner were reasonably successful in predicting velocity profiles with relatively coarse 2-D and 3-D models. However, Mongia (1994) points out that prediction of pressure losses is more difficult and the results are typically relatively poor.

A logical next step is to model the entire flow field from the compressor diffuser to the turbine inlet with the flow inside and outside the combustor liner fully coupled. There are at least two important reasons to attempt such an ambitious task:

1. flow splits and boundary conditions for the combustor liner inlets are modeled explicitly and no longer need to be approximated; and
2. liner wall temperatures can be predicted when the flow fields on both sides of the liner walls are modeled in a coupled fashion.

**Presented at the International Gas Turbine & Aeroengine Congress & Exhibition
Stockholm, Sweden — June 2–June 5, 1998
This paper has been accepted for publication in the Transactions of the ASME
Discussion of it will be accepted at ASME Headquarters until September 30, 1998**

Downloaded from http://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1998/78644/V003T06A007/2410774/v003t06a007-98-gt-184.pdf by guest on 18 May 2021

One-dimensional annulus codes have traditionally been utilized to provide a reasonable prediction of flow splits, but they cannot capture important two and three dimensional flow features that can significantly affect total pressure losses. Nonuniform profiles of flow conditions, including velocities, jet angles and turbulence properties, at inlet boundaries can also have a significant influence on the flowfield in a combustor. Turbulence properties at inlets are particularly difficult to estimate. McGuirk and Spencer (1995) discuss the sensitivity of discharge coefficient and jet trajectory of dilution holes to small changes in the dilution hole geometry. They emphasize the need to provide realistic profiles at dilution hole inlets.

Another important issue is accurate definition of the flow through the dome/fuel injector air swirlers. Fuller and Smith (1993) demonstrated the sensitivity of the overall solution to the details of the swirler air flow. The aerodynamics of recent lean dome combustor designs are dominated by the large percentage of air that is admitted through the dome swirlers. Crocker, et al. (1996) discusses the importance of detailed prescription of fuel nozzle/dome swirler boundary conditions and describes an approach for accurate prediction of flow splits and effective area. Fuller and Smith modeled the air swirler and combustor flow fields in a decoupled manner which may be appropriate in some cases. In general, though, it is necessary to couple the combustor solution and at least part of the swirler passages because of strongly recirculating flow in the region of the swirler. Another important consideration is the effect of nonuniform feed pressure at the inlet of the dome swirlers. The advent of multi-block and many-to-one grid topology, discussed later in the paper, make it feasible to include a detailed model of the swirler passages in the overall combustor model.

The prediction of liner wall temperatures is becoming increasingly important as more aggressive cycles increase liner heat loads and low emissions designs reduce the available cooling air. Although conjugate heat transfer approaches that couple heat transfer in solid material with the fluid flow solution have been available for some time, the authors are not aware of any attempt (in the open literature) to directly model gas turbine combustor liner wall temperatures using CFD.

This paper describes CFD analyses of a model combustor from the deswirl vane exit of a centrifugal compressor to the turbine nozzle inlet. The combustor configuration is representative of a lean, low emission design. The model includes an airblast fuel nozzle, dome and liner walls with dilution holes and cooling louvers. Since the combustor is a representative model and there are no experimental data, the emphasis is placed on the modeling methodology. The results of the analyses illustrate the influence of flow nonuniformities on the static pressure distribution around the liner and the resulting effect on flow splits and liner wall temperatures. The influence of non-luminous radiation and thermal barrier coating on wall temperatures is also examined.

COMPARISON OF CFD AND EMPIRICAL HEAT TRANSFER MODELS

The simple geometry of a pipe flowing with hot combustion products surrounded by a concentric annulus flowing with cooling air is employed to examine the relative influence of radiation and convection in comparison with an empirical formulation for predicting wall temperature. The geometry is illustrated in Figure 1. The empirical formula was originally derived by Lefebvre and Herbert (1960) and later restated by Lefebvre (1983). The equations, repeated below, utilize basic heat transfer principles and empirical data to account for radiation and convection on both sides of the liner.

$$\varepsilon_g = 1 - \exp(-290PL(ql_b))^{0.5}T_g^{-1.5} \tag{1}$$

$$R_1 = 0.5\sigma(1+\varepsilon_w)\varepsilon_g T_g^{1.5}(T_g^{2.5} - T_{w_1}^{2.5}) \tag{2}$$

$$C_1 = K_{C1}\frac{k_g}{D_L^{0.2}}\left(\frac{\dot{m}_g}{A_L\mu_g}\right)^{0.8}(T_g - T_w) \tag{3}$$

$$R_2 = K_{R2}(T_{w_2}^4 - T_3^4) \tag{4}$$

$$C_2 = K_{C2}\frac{k_a}{D_{an}^{0.2}}\left(\frac{\dot{m}_{an}}{A_{an}\mu_a}\right)^{0.8}(T_{w_2} - T_3) \tag{5}$$

$$R_1 + C_1 = R_2 + C_2 = \frac{k_w}{t_w}(T_{w_1} - T_{w_2}) \tag{6}$$

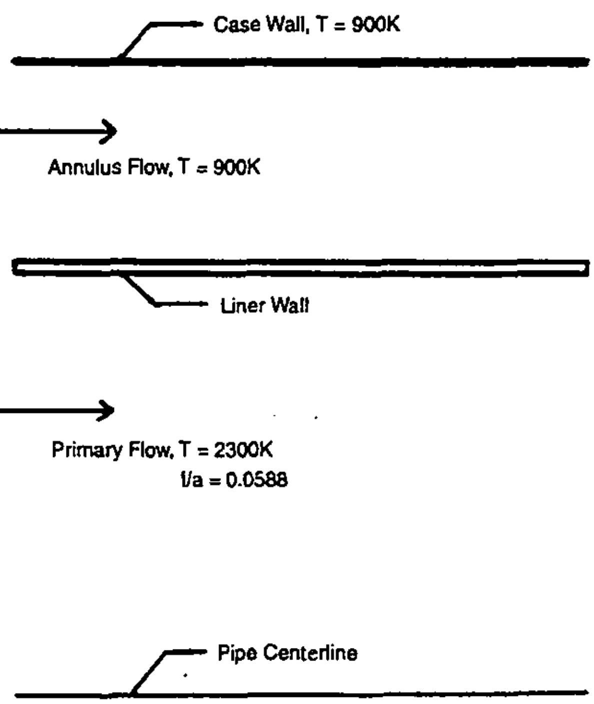

Figure 1. Pipe and Annulus Test Case Geometry

A detailed description of the elements of equations 1-5 is not repeated here since they are fully described in the widely available text by Lefebvre (1983). Each case discussed here had T_3 of 900K, a fuel/air ratio of 0.0588, and a hot product (gas) temperature T_g of 2300K.

Downloaded from http://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1998/78644/V003T06A007/2410774/v003t06a007-98-gt-184.pdf by guest on 18 May 2021

The luminosity factor was L=1.0 and the beam length was 0.08 m. The emissivity of both sides of the liner and the case were assumed to 0.7 and 0.4, respectively. The liner inner diameter was 0.132 m, the wall thickness was 1.2 mm and the case diameter was 0.192 m. The pressure and the inside and outside velocities were varied.

The commercially available CFD code CFD-ACE was used for all of the analyses. The CFD model used a modified k-e turbulence model based on the Renormalization Group (RNG) theory (Yakhot *et al.*, 1992) and standard wall functions (Launder and Spalding, 1974) to predict the convective heat transfer. It is well known that wall functions are limited in their ability to predict convective heat transfer, especially for flow fields that are not parallel to the wall. Nevertheless, alternative low Reynolds number models require very fine grids near the walls and are currently not very practical for a full combustor model. The grid density near the wall was set for each case such that y^+ was approximately 25. Note that the near wall grid density varied considerably as a function of pressure and inlet velocity.

The turbulent Prandtl and Schmidt numbers were set to 0.25 for the CFD models. This relatively low value has consistently demonstrated better agreement with combustor fuel-air mixing results. However, the low value also results in increased heat transfer compared to the typical value of 0.9 (about 50% higher than with Prandtl number of 0.25 for a typical case with radiation). Since the convective heat transfer rate is increased on both sides of the liner, the effect on the wall temperature is relatively small. The temperature gradient through the wall, however, is overpredicted. An improved approach would be to use conventional turbulent Prandtl and Schmidt numbers in the wall functions. The convection terms (equations 3 and 5) of the empirical model were calibrated so that the convective heat transfer for the empirical model and the CFD model was equal. The calibration was performed for primary and annulus velocities both set to 20 m/s. The constants K_{C1} and K_{C2} were set to 0.046 and 0.040, respectively. Both constants from Lefebvre (1983) are 0.020 based on fully developed pipe flow. Since the test case under consideration predicts heat transfer near the pipe entrance with an L/D less than 0.5, far from fully developed flow, the correction factors of 2.3 and 2.0 are reasonable.

Radiation heat transfer is calculated by solving the radiative transfer equation using a discrete-ordinate method. The discrete-ordinate method provides a reasonable balance between obtaining realistic results for complex geometries and computational cost. The approach, along with validation cases, is described by Giridharan *et al.* (1995). Only CO_2 and H_2O were assumed to contribute significantly to the nonluminous gas radiation. The local gas emittance was calculated as a function of species concentration, pressure and temperature based on the data of Hottel (1954). Luminous radiation was not considered for this effort since these cases only involved lean combustion.

Results of the empirical and CFD models are compared in Figure 2 as a function of pressure and primary/annulus velocities. The agreement is remarkably good considering the uncertainties in both models. The most notable difference is that the CFD predicted wall temperature begins to drop below the empirical prediction as the pressure increases. This is caused by the lower rate of increase of the gas radiation to the wall for the CFD model. This is certainly an area of uncertainty since the data from Hottel (which both the empirical and CFD radiation models are based on) must be extrapolated beyond 5 atmospheres.

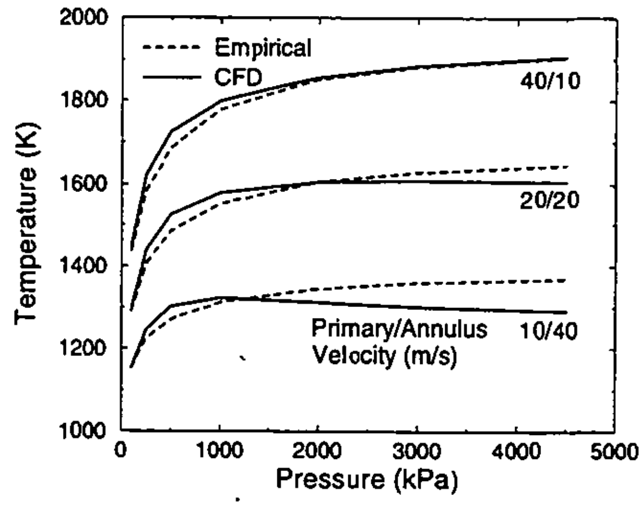

Figure 2. Comparison of Empirical and CFD Wall Temperature Predictions for Primary/Annulus Velocities of 10/40, 20/20, and 40/10 m/s

COMBUSTOR CFD MODEL DESCRIPTION

A single nozzle 24 degree periodic sector of the model combustor was analyzed. The combustor configuration included a centrifugal compressor diffuser section downstream of the deswirl vanes, a fuel nozzle and stem, typical cooling flow and a single row of dilution holes on both the inner and outer liner. The fuel nozzle was an airblast design with about 65% of the combustor air admitted through the two axial air swirlers. An axial cross-section of the 370,000 cell grid with 11 domains (blocks) is shown in Figure 3.

Generation of a grid for such a complex geometry in a timely manner and with sufficient grid resolution is a challenging task. Mongia (1994), in his review of combustion modeling, estimated that as many as 12 million cells may be required to provide a near grid independent solution for a comprehensive one nozzle sector combustor model. Such a model would include detailed modeling of swirler vane passages. Second order numerical schemes can reduce that requirement to less than 5 million cells. (Second order upwind differencing was used for the current analyses.) Mongia's estimate is reasonable given the assumption of one-to-one grid cell interfaces between blocks. Many-to-one grid cell interfaces can dramatically reduce the total number of required cells. Many-to-one is illustrated by a close-up of the fuel nozzle grid interfaced with the primary zone grid shown in Figure 4. Detailed structures, such as swirler passages and even swirler vanes, can be modeled with a fine grid without transferring the high density grid into the rest of the combustor. (The grid quality could be improved by continuing the fine grid a short distance into the primary zone.) The swirler passages shown in Figure 4 were modeled with 35,000 cells which does not include swirler vanes. Swirler vanes could be included with approximately 100,000 additional cells. The difficulties associated with constructing such a grid are considerable, but not insurmountable, and will be addressed in the last section of the paper.

Downloaded from http://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1998/78644/V003T06A007/2410774/v003t06a007-98-gt-184.pdf by guest on 18 May 2021

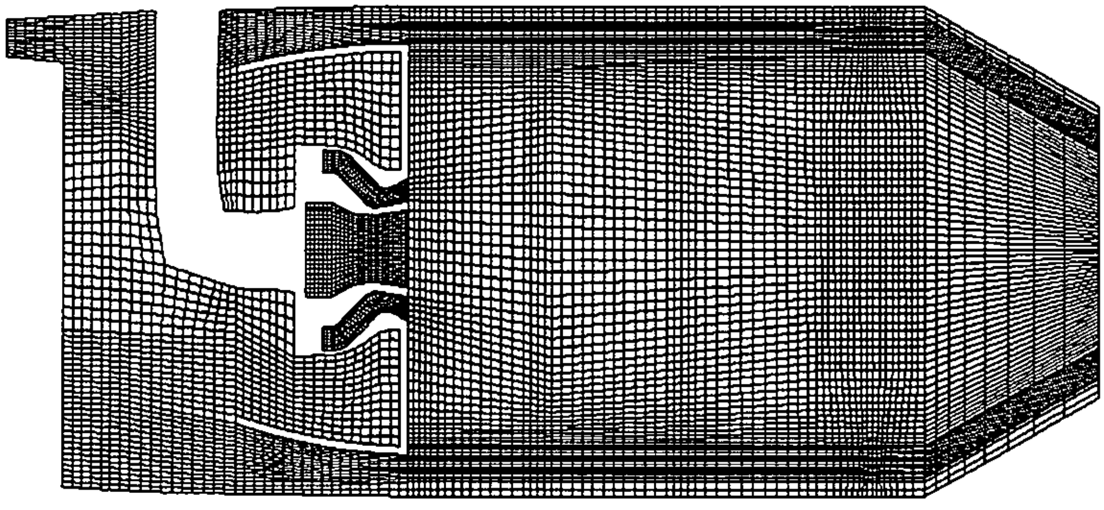

Figure 3. Centerline Axial Plane of Fine Grid (370,000 cells)

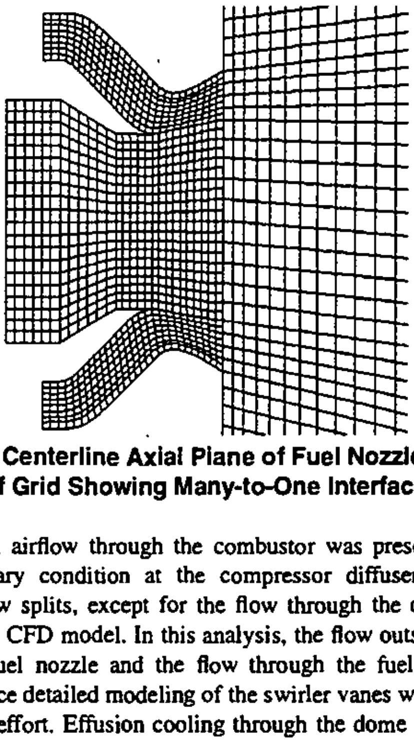

Figure 4. Centerline Axial Plane of Fuel Nozzle Portion of Grid Showing Many-to-One Interface

The total airflow through the combustor was prescribed by the single boundary condition at the compressor diffuser. All of the remaining flow splits, except for the flow through the dome, were a product of the CFD model. In this analysis, the flow outside the dome feeding the fuel nozzle and the flow through the fuel nozzle were decoupled since detailed modeling of the swirler vanes was beyond the scope of this effort. Effusion cooling through the dome face was also modeled in a decoupled manner. (Conjugate heat transfer modeling of effusion cooling is briefly discussed in the last section of the paper.) The flow rates feeding the air swirlers and effusion cooled dome walls were modeled as exit boundaries and were matched to the inlet flow rates on the downstream side of the dome. The swirler inlet boundaries were modeled according to Crocker, et al. (1996). Of course, the effect of variations in static pressure across the dome could not be captured. The liner walls with film cooling and dilution holes were modeled in a fully coupled manner. The holes feeding the film cooling slots were modeled by concentrated resistances since it was not practical to model the individual holes. A close-up of part of the outer liner grid is shown in Figure 5. The walls containing the dilution holes were three cells thick and the upstream walls were two cells thick. The liner wall thermal conductivity was assumed to be 23 W/(m-K), typical of metallic liners. The case wall was assumed isothermal at the combustor inlet temperature. Other walls such as the atomizer stem and the dome cowl were assumed to be adiabatic.

The dilution holes were modeled as round holes (not castellated approximations) with 6 by 6 grid cells. This is a typical grid density for dilution holes in most current combustor models, however, it may not be enough to achieve grid independence. McGuirk and Spencer (1993) compared models of a dilution hole with the castellated approximation and with fitted round holes, both with about a 12x12 and 20x20 mesh in the hole. There were significant differences between the castellated and fitted solutions. The solutions for the coarse and fine grids with fitted round holes, based on the velocity profile across the hole, were quite similar. This indicates that a 12x12 mesh, and possibly less, is adequate for resolving dilution holes. Though not utilized for this analysis, many-to-one grid topology can again be used to keep the total number of grid cells to a manageable level.

Kinetics were modeled using a one-step finite rate reaction with equilibrium reaction products. This efficient model accounts for the significant effects of finite reaction time and provides a much better prediction of flame temperature compared to simple one-step models with only H_2O and CO_2 as products. The one-step kinetics is also amenable to the prescribed pdf model which was used to account for turbulent/chemistry interaction. The two-variable pdf was applied to the fuel mixture fraction and to the progress variable. The fuel mixture fraction pdf is of primary importance for diffusion flames and the progress pdf is most important for premixed flames. Lean direct injection, as employed in this case, is often a hybrid condition for which both are significant. The liquid fuel spray motion and evaporation was modeled as Lagrangian droplet packets that influence the Eulerian gas phase equations through mass, energy, and momentum source terms.

Downloaded from http://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1998/78644/V003T06A007/2410774/v003t06a007-98-gt-184.pdf by guest on 18 May 2021

Figure 5. Close-up of Part of the Outer Liner Grid with Wall Cells Shaded

RESULTS

Selected results of the CFD calculations are discussed below primarily for the purpose of illustrating the qualitative nature of the predictions of a full combustor model. The flow pattern in the primary zone is dominated by a strong centerliner recirculation zone. Temperature contours in an axial plane along the nozzle centerline are shown in Figure 6. Flame temperatures are relatively low as a consequence of the lean primary zone equivalence ratio. Relative static pressure contours (exit pressure is 3.0 MPa) for the same axial plane are shown in Figure 7. Significant pressure variations of 1 to 2% of static pressure are evident around the liner. The most notable variation, typical of a centrifugal compressor configuration, is the approximately 1% higher pressure in the outer annulus. There are also significant circumferential variations of pressure, particularly around the fuel injector/dome region (not shown in any graphic). Hot side liner wall temperatures and velocity vectors are shown in Figure 8. An increase in wall temperature with distance away from the cooling slots is predicted along with significant circumferential variation.

Gas temperatures as a function of radius at a location 2.54 cm downstream from the dome and in line with the nozzle centerline are shown in Figure 9. Temperatures for a coarse grid and fine grid case with and without nonluminous gas radiation are compared. The coarse grid had just 80,000 cells compared with 370,000 cells for the fine grid. The coarse grid case had a slightly flatter profile as might be expected because of greater numerical diffusion. The effect of radiation on the gas temperatures was very small; a reducion of at most 5K near the combustor centerline was predicted. The effect of soot radiation, not modeled here, could potentially be more significant in many cases. In this case, however, soot should not play an important role because of the lean primary zone conditions.

Outer liner wall temperatures for the coarse and fine grid cases with and without radiation at a location 2.54 cm downstream of the dome are shown in Figure 10. The metal wall temperature for a coarse grid case with a 1.25 mm thick thermal barrier coating (TBC) (k=0.66 W/m-K) is also shown. The effect of the TBC was to reduce the hot side metal wall temperatures and to smooth out the variations considerably. The hot side of the TBC was considerably higher than the comparable metal wall case, so the heat transfer through the wall was reduced. Wall temperatures were consistently about 40 K higher for the radiation cases. There is considerable difference between the fine and coarse grid results. To further compare the fine and coarse grid cases, temperature contours for a transverse plane 2.54 cm downstream of the dome are compared in Figure 11. The overall pattern is similar, but the variations near the wall are significant as reflected by the different wall temperature patterns.

CONCLUSIONS AND FUTURE DIRECTION

CFD calculations of a model combustor from compressor diffuser exit to turbine inlet were successfully performed. The comprehensive model made it possible to predict flow splits for the various openings into the combustor liner and remove the guesswork required for prescribing accurate boundary conditions for those openings. The coupled modeling of the flow inside and outside the liner, combined with conjugate heat transfer analysis and participating gas radiation, provided a direct prediction of liner wall temperatures. Nonluminous gas radiation added approximately 40 K to the hot side metal wall temperature. TBC reduced peak metal wall temperatures by more than 200 K.

Several issues that were not addressed in this effort need to be resolved for a complete, comprehensive combustor model. Numerous physical models such as turbulence, kinetics, atomization, and soot need improvement and/or development. These issues are not considered here. The issues of interest here are those unresolved or incomplete capabilities required to fully couple the flow from the compressor diffuser exit to the turbine inlet. They include grid generation and modeling of effusion cooling.

The challenge in generating the grid is twofold: 1) it must be possible to generate a quality grid in a reasonable amount of time, and 2) the grid must have adequate resolution to capture the relevant physics but be small enough that 24 hour or less execution time can be maintained. Most CFD codes, including CFD-ACE, that have adequate physical models for solving reacting flow with spray, utilize a structured grid topology. It is possible to model virtually any geometry using a multi-block, many-to-one structured approach, however the process is quite painful for complex geometries such as swirler vane passages. Unstructured grids are generally easier to generate for complex geometries and the potential for automation is greater. Automaion of grid generation (e.g. swirler vanes, dilution holes, cooling slots) is an important requirement for making comprehensive combustor modeling a practical design tool. Unstructured grids also usually require fewer grid cells since the grid packing can be isolated in the appropriate areas. (Many-to-one accomplishes the same thing to a lesser extent in structured grids.) The need for mixed element 'brick' cells near walls is one of the primary difficulties. Physical models in unstructured grid solvers are generally less mature than for structured grid solvers, but should catch up in the relatively near future. One interesting approach is to use an unstructured solver for the nonreacting and generally more geometrically complex regions feeding the liner, and a more mature structured solver for the reacting region inside the liner in a fully coupled manner. An additional advantage to this approach is that the computational expense associated with reaction and radiation can be limited to the appropriate regions of the flow.

Downloaded from http://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1998/78644/V003T06A007/2410774/v003t06a007-98-gt-184.pdf by guest on 18 May 2021

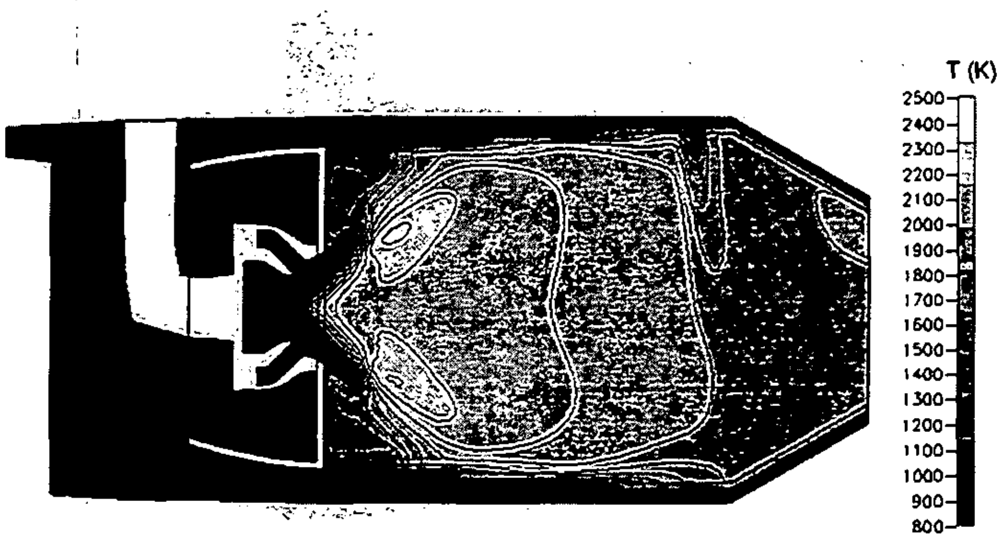

Figure 6. Temperature Contours for an Axial Plane Along the Nozzle Centerline

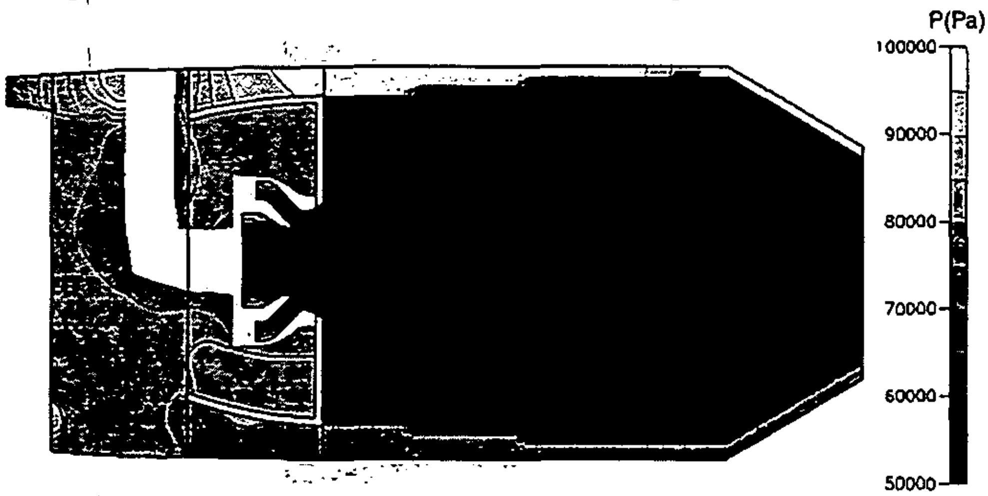

Figure 7. Relative static Pressure Contours for an Axial Plane Along the Nozzle Centerline (Exit Pressure = 3.0 MPa)

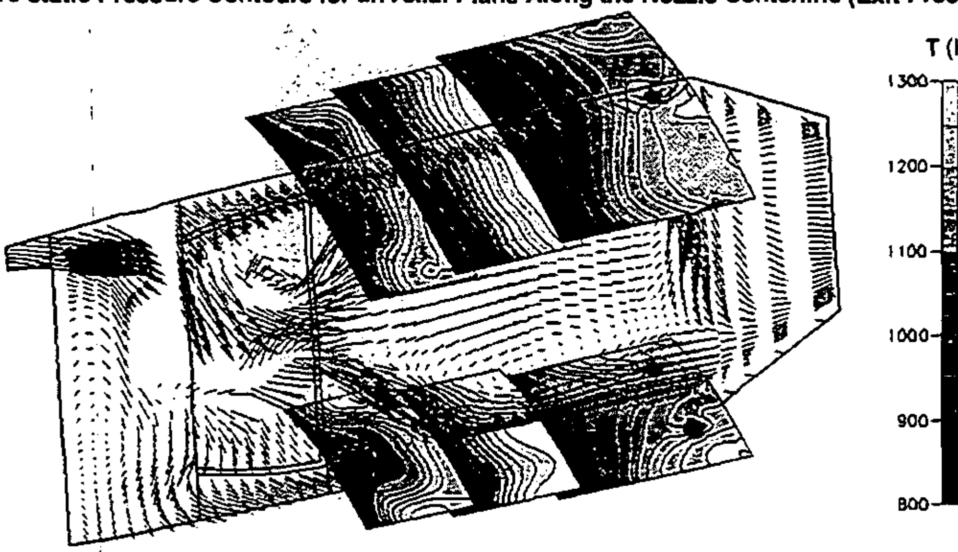

Figure 8. Hot Side Liner Wall Temperature Contours and Velocity Vectors for an Axial Plane Along the Nozzle Centerline

Downloaded from http://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1998/78644/V003T06A007/2410774/v003t06a007-98-gt-184.pdf by guest on 18 May 2021

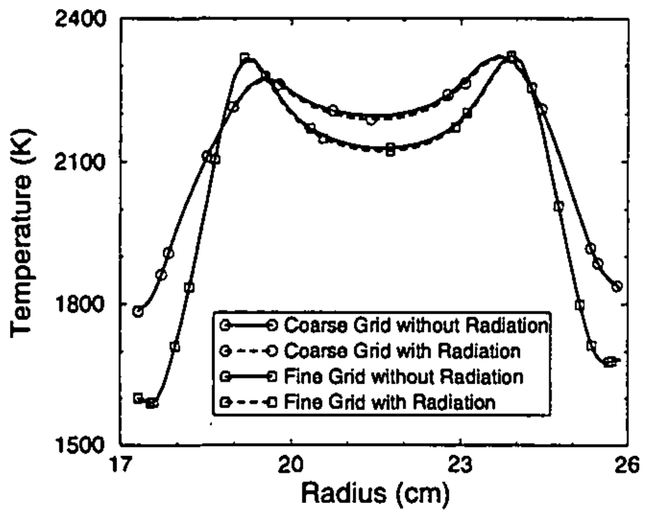

Figure 9. Gas Temperatures 2.54 cm Downstream of the Dome and In Line with the Nozzle Centerline

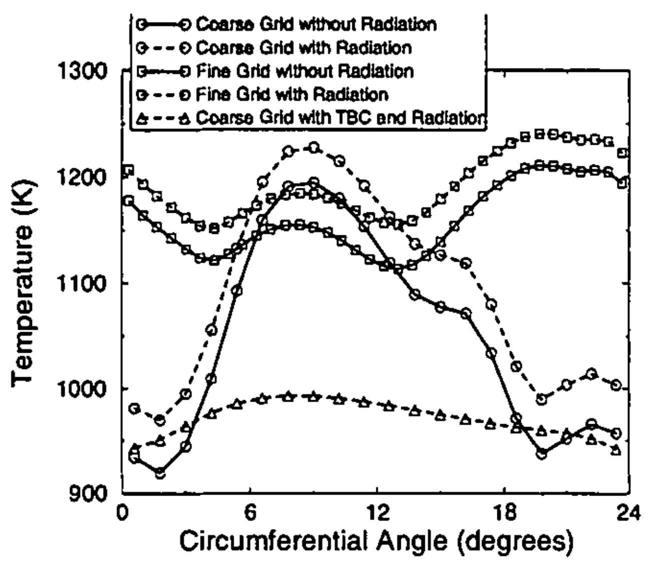

Figure 10. Hot Side Wall Temperatures on the Outer Liner 2.54 cm Downstream of the Dome (See Arrow in Figure 8 to Identify Location)

Effective modeling of effusion cooling, now used for liner wall cooling in many combustors, is essential for coupling the flow between the annulus and the combustor regions. The large number of very small holes associated with effusion cooling make it impractical in the foreseeable future to directly model effusion cooled walls. One solution is to develop a submodel that calculates mass flow through the wall and heat transfer from the wall and provides the appropriate mass, momentum, and energy source terms for the gas phase cells adjacent to the wall and the appropriate energy source terms for the wall.

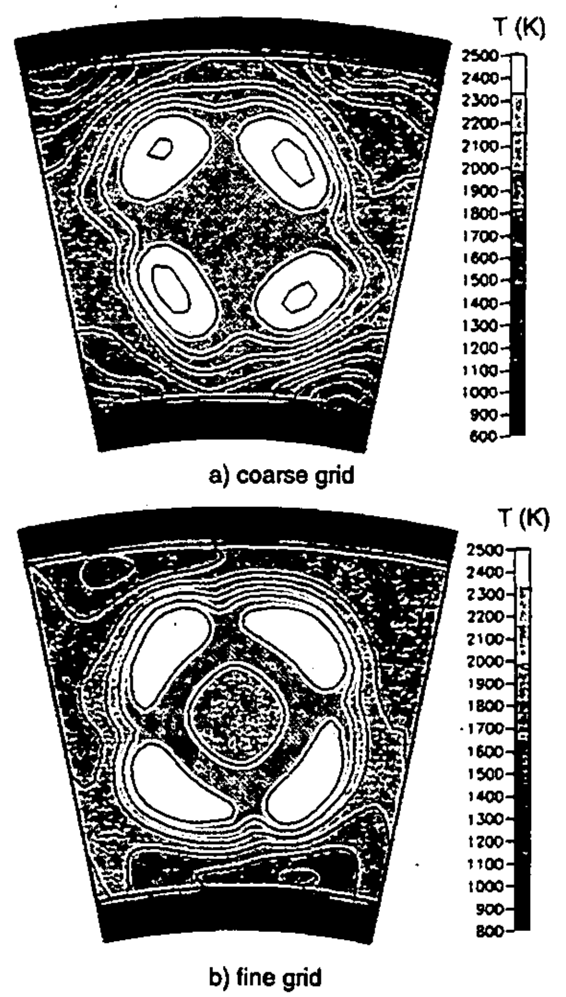

Figure 11. Temperature Contours for a Transverse Plane 2.54 cm Downstream of the Dome—Comparison of Coarse and Fine Grid Cases

In the near to mid term future, it should be possible to perform fully coupled combustion system CFD calculations. Adequate grid resolution can be achieved with roughly 1 million cells using advanced grid generation techniques and second order discretization schemes. Execution time for the 370,000 cell case described in this paper was about 48 hours (with radiation and with little optimization) on a 500 MHz DEC Alpha 500. If one assumes 4 times more execution time for a 1 million cell case, a factor of 8 improvement is needed to achieve one day turnaround time. If history is a guide, such improvement may be realized for a single workstation within 5 years. Overnight execution can be achieved now with 10 to 15 workstations operating in parallel.

REFERENCES

Crocker, D.S., Fuller, E.J., and Smith, C.E., 1996, "Fuel Nozzle Aerodynamic Design Using CFD Analysis," ASME paper 96-GT-127.

Fuller, E.J. and Smith, C.E., 1993, "Integrated CFD Modeling of Gas Turbine Combustors," AIAA paper 93-2196.

Downloaded from http://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1998/78644/V003T06A007/2410774/v003t06a007-98-gt-184.pdf by guest on 18 May 2021

Giridharan, M.G., Lowry, S., and Krishnan, A., 1995, "Coupled Conductive-Convective-Radiative Conjugate Heat Transfer Model for Complex Applications," ASME paper 95-WA/HT-5.

Hottel, H.C., 1954, "Radiant Heat Transmission," in W.H. McAdams (ed.), Heat Transmission, 3rd ed., McGraw-Hill.

Karki, K.C., Oeclsle, V.L., and Mongia, H.C., 1992, "A Computational Procedure for Diffuser-Combustor Flow Interaction Analysis," J. of Eng. for Gas Turbines and Power, Vol. 114, pp. 1-7.

Lai, M.K., 1997, "CFD Analysis of Liquid Spray Combustion in a Gas Turbine Combustor," ASME paper 97-GT-309.

Launder, B.E. and Spalding, D.B., 1974, "The Numerical Computation of Turbulent Flow," *Comp. Methods Appl. Mech. Engr.*, vol. 3, p. 269.

Lawson, R.J., 1993, "Computational Modeling of an Aircraft Engine Combustor to Achieve Target Exit Temperature Profiles," ASME paper 93-GT-164.

Lefebvre, A.H. and Herbert, M.V., 1960, "Heat-Transfer Processes in Gas-Turbine Combustion Chambers," Proc. Inst. Mech. Engr., vol. 174, no. 12, pp. 463-473.

Lefebvre, A.H., 1983, Gas Turbine Combustion, McGraw-Hill.

Little, A.R. and Manners, A.P., 1993, "Predictions of the Pressure Losses in 2D and 3D Model Pump Diffusers," ASME paper 93-GT-184.

McGuirk, J.J and Spencer, A., 1993, "CFD Modeling of Annulus/Port Flows," ASME paper 93-GT-185.

McGuirk, J.J and Spencer, A., 1995, "Computational Methods for Modelling Port Flows in Gas-Turbine Combustors," ASME paper 95-GT-414.

Mongia, H.C., 1994, "Combustion Modeling in Design Process: Applications and Future Direction," AIAA paper 94-0466.

Srinivasan, R., Freeman, W.G., Mozumdar, S., and Grahmann, J.W., 1990, "Measurements in an Annular Combustor-Diffuser System," AIAA paper 90-2162.

Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B., and Speziale, C.G., 1992, "Development of Turbulent Models for Shear Flows by a Double-Expansion Technique," *Physics of Fluids*, vol. 4, pp. 1510-1520.

Downloaded from http://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1998/78644/V003T06A007/2410774/v003t06a007-98-gt-184.pdf by guest on 18 May 2021