



Experimental and Computational Study of Trapped Vortex Combustor Sector Rig With Tri-Pass Diffuser

R.C. Hendricks
Glenn Research Center, Cleveland, Ohio

D.T. Shouse and W.M. Roquemore
Wright-Patterson Air Force Base, Dayton, Ohio

D.L. Burrus and B.S. Duncan
General Electric Aircraft Engines, Cincinnati, Ohio

R.C. Ryder and A. Brankovic
Flow Parametrics, LLC, Bear, Delaware

N.-S. Liu and J.R. Gallagher
Glenn Research Center, Cleveland, Ohio

J.A. Hendricks
Diligent Design, Toledo, Ohio

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA's counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.
- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.

- **CONFERENCE PUBLICATION.** Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.
- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at <http://www.sti.nasa.gov>
- E-mail your question via the Internet to help@sti.nasa.gov
- Fax your question to the NASA Access Help Desk at 301-621-0134
- Telephone the NASA Access Help Desk at 301-621-0390
- Write to:
NASA Access Help Desk
NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076



Experimental and Computational Study of Trapped Vortex Combustor Sector Rig With Tri-Pass Diffuser

R.C. Hendricks
Glenn Research Center, Cleveland, Ohio

D.T. Shouse and W.M. Roquemore
Wright-Patterson Air Force Base, Dayton, Ohio

D.L. Burrus and B.S. Duncan
General Electric Aircraft Engines, Cincinnati, Ohio

R.C. Ryder and A. Brankovic
Flow Parametrics, LLC, Bear, Delaware

N.-S. Liu and J.R. Gallagher
Glenn Research Center, Cleveland, Ohio

J.A. Hendricks
Diligent Design, Toledo, Ohio

National Aeronautics and
Space Administration

Glenn Research Center

Acknowledgments

The authors would like to thank Dr. Marvin Goldstein, NASA Glenn Research Center, Strategic Research Fund, the Air Force Research Laboratory (AFRL), and GE Aircraft Engines (GEAE), for their assistance in making this research possible.

Available from

NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22100

Available electronically at <http://gltrs.grc.nasa.gov>

Experimental and Computational Study of Trapped Vortex Combustor Sector Rig With Tri-Pass Diffuser

R.C. Hendricks
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

D.T. Shouse and W.M. Roquemore
Wright-Patterson Air Force Base
Dayton, Ohio 45433

D.L. Burrus and B.S. Duncan
General Electric Aircraft Engines
Cincinnati, Ohio 45215

R.C. Ryder and A. Brankovic
Flow Parametrics, LLC
Bear, Delaware 19701

N.-S. Liu and J.R. Gallagher
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

J.A. Hendricks
Diligent Design
Toledo, Ohio 43614

Summary

The Trapped Vortex Combustor (TVC) potentially offers numerous operational advantages over current production gas turbine engine combustors. These include lower weight, lower pollutant emissions, effective flame stabilization, high combustion efficiency, excellent high altitude relight capability, and operation in the lean burn or RQL modes of combustion. The present work describes the operational principles of the TVC, and extends diffuser velocities toward choked flow and provides system performance data. Performance data include EINO_x results for various fuel-air ratios and combustor residence times, combustion efficiency as a function of combustor residence time, and combustor lean blow-out (LBO) performance. Computational fluid dynamics (CFD) simulations using liquid spray droplet evaporation and combustion modeling are performed and related to flow structures observed in photographs of the combustor. The CFD results are used to understand the aerodynamics and combustion features under different fueling conditions. Performance data acquired to date are favorable compared to conventional gas turbine combustors. Further testing over a wider range of fuel-air ratios, fuel flow splits, and pressure ratios is in progress to explore the TVC performance. In addition, alternate configurations for the upstream pressure feed, including bi-pass diffusion schemes, as well as variations on the fuel injection patterns, are currently in test and evaluation phases.

Introduction

The trapped vortex combustor (TVC) is a novel design concept radically departing from the conventional swirl stabilized gas turbine engine annular combustor design approach that has been in use for the past 40 years. In the TVC concept, combustion is stabilized within cavities that are incorporated into the combustor liners. The cavities are designed to trap a vortex flow structure established through the use of driver air jets located along the cavity walls. Some amount of the overall combustor fuel is injected into the cavities where it mixes and burns quickly in the stable trapped vortex flow structure. The hot gases from these stable combustion zones exhaust from the cavities along radial struts spanning the gap between the inner liner and outer liner cavities. These hot gases then spread out across the face of the dome serving as pilots for igniting the remaining fuel that is injected into the main air stream entering the burner through the dome. The TVC design approach allows use of simple fuel injection techniques. It has demonstrated significantly enhanced ignition and combustion stability characteristics, provided high combustion efficiency over a much wider operating range, and provided significant reductions in pollutant emissions.

The TVC concept has been pioneered and developed by the Air Force Research Laboratory (AFRL) and GE Aircraft Engines (GEAE) under the joint sponsorship of AFRL, the Navy, and the Strategic Environmental Research Development Program (SERDP) (Roquemore et al., 2001). As part of this development effort, AFRL and GEAE jointly designed, fabricated, and tested a full scale 12-inch-wide rectangular prototypical TVC sector rig capable of operation at elevated inlet pressures and temperatures. A large quantity of performance data has been collected while running this prototypical test rig at operating conditions representative of typical conventional aircraft gas turbine engine applications (Burrus et al., 2001).

In this NASA-sponsored research program, the TVC 12-inch sector test rig was operated at high inlet velocities similar to those obtained in high flight speed applications. Unlike practical high-speed combustion systems that use ram air, high pressure drops across the combustor were used to increase the burner flow through velocity. Although this is not a practical approach for a practical vehicle, it does provide insights into the combustion process with high-speed inlet flows. The objectives of this work are 1) to extend the operational range of the TVC for high speed applications, 2) to establish a basis for numerical code evaluations by developing a database of performance characteristics and operational behavior of the TVC under conditions representing high through-flow velocities, and 3) to apply and assess the capabilities of an advanced multidimensional compressible reacting flow computational fluid dynamics (CFD) code to qualitatively and quantitatively capture the flow field and physics within the operating TVC. This paper presents some of the results from these early experimental investigations along with results from some of the initial CFD modeling calculations.

The experiments were simulated using a CFD solver developed to compute multiphase, chemically reacting flows in complex geometries, such as those in aircraft gas turbine combustors. The FPVortex™ flow solver is applied to compute three cases of interest for the TVC test rig. Distinct fueling schemes were selected for the CFD runs, to illustrate differences in aerodynamic and combustion flow structures. Details of the CFD simulations are provided, and are discussed with regard to the photographic results obtained on the TVC rig at the same fueling conditions.

Background

Trapped Vortex Combustor

The experimental apparatus used was an available 12-in wide rectangular (planar) sector rig that was originally designed, fabricated, and tested by a team consisting of representatives from the Air Force Research Laboratory (AFRL) and GE Aircraft Engines (GEAE). This same group of AFRL and GEAE

representatives participated with NASA GRC representatives to conduct this current experimental testing. The TVC combustor rig is a planar sector rig consisting of 10 modules arranged linearly. Opposed upper and lower cavities are present, all with liquid spray pressure atomizing fuel injectors, numerous air jets, and film and slot cooling injections. The main combustor air is fed by a diffusion system consisting of three passages (tri-pass configuration), which split the incoming flow into approximately three equal flows. This flow enters the combustor at high, but subsonic speed. For the cavity-only fueled operating mode, only upstream plenum air flow passes through the diffuser passages. For the cavity-plus-main operating mode, main stream fuel is supplied to each passage via a pair of opposed liquid spray pressure atomizing fuel injectors. In the process of fueling the main stream, the fuel is injected into the high-speed cross-streams passing through the diffuser passages to provide some degree of premixing of the main fuel air before the mixture enters the combustion chamber for burning (Roquemore et al., 2001 and Burrus et al., 2001).

The 12-in rectangular TVC sector rig hardware is shown in figure 1. In this photograph one of the air cooled sidewalls has been removed for viewing. The arrow shown on the figure indicates flow from left to right. Station 3 (plenum) air enters into the diffuser, and is split into three streams by the diffuser struts at the upstream end of the tri-pass branches. The flow is diffused, however, as it must wrap around a fuel strut perpendicular to the flow. The fuel strut is airfoil-shaped, splitting the flow within each branch in half. Liquid spray simplex-type pressure atomizing-fuel injectors are situated approximately midway down the fuel strut. When turned on, they inject directly into these tri-pass diffuser air streams. The upper and lower cavity fuel injection points (large openings on the cavity forward walls) can also be seen in the photo.

In the cavity only fueled-operating mode high-intensity combustion takes place and is largely confined to the cavity and the recirculation regions formed between the flows issuing from the tri-pass passages. For the cavity-plus-main operating mode, additional combustion takes place further downstream in the burner duct, as the liquid fuel from the main-stream injection evaporates and is ignited by the hot products transported out of the cavity.

Photographic results for the three cases under study here are shown in figures 2, 3, and 4. Case 1 results, taken at an inlet pressure of 406 kPa, inlet temperature of 539 K, overall equivalence ratio of $\phi = 0.53$, and fueling split between the cavity ($\phi = 0.65$) and main feed system ($\phi = 0.72$), are shown in figure 2. Case 2 results, taken at an inlet pressure of 396 kPa, temperature of 506 K, overall $\phi = 0.51$ and fueling split between the cavity ($\Phi = 1.86$) and main ($\Phi = 0.13$), are shown in figure 3. Finally, cavity-only fueling was used for Case 3, with cavity $\phi = 2.2$ and overall $\phi = 0.53$ at an inlet pressure of 314 kPa and temperature of 546 K. These results are shown in figure 4. The rich fueling condition for the cavity results in highly intense combustion, with high fuel residence times, and overall high combustion efficiencies.

Computational Fluid Dynamics Methodology

The CFD modeling approach described in Ryder and McDivitt (2000) is followed in this work. An important aspect of this approach is to model the combustion system geometry precisely as it is installed on the test stand. The starting point in the process is development of a fully-detailed computer aided design (CAD) solid model representation of the hardware. The CAD solid model is preferably based on the same CAD model as that used for structural and thermal finite element analysis, and also for manufacturing. This approach has the benefit of including not only all the major flow hardware features, but also additional and miscellaneous hardware such as fasteners, bolts, support brackets, and other assembly components which can often adversely affect the flow distributions and quality. A further benefit of this approach is that all flow paths which are aerodynamically coupled will be identified and included in the model. Once a satisfactory model has been created, an “air-solid” representation of the hardware is created, using a Boolean operation. This is a solid model representation of the gas flow path.

This model can then be automatically meshed using any of several commercial mesh generators, generally using tetrahedral elements. The resulting grid is tagged with boundary conditions, and outputted into an unstructured database read in by the flow solver. Intermediate results databases are created for assessing convergence, which can be viewed automatically in graphics post-processing packages. Further examples of this CFD methodology applied to gas turbine component flow fields may be found in the works by Ryder (2000) and Brankovic et al. (2001).

Physics-based modeling approaches are used to resolve the aerodynamics and combustion processes taking place within the combustion system. Turbulence is modeled using the two-equation or $k-\epsilon$ model of turbulence, together with wall functions. For combustion processes in which fuel-air mixing and flame stabilization are key phenomena to be resolved, turbulence wall functions provide adequate accuracy. If, however, pressure drop through the pre-diffuser or heat transfer along liner walls are of concern, near-wall grid clustering together with wall integration models of turbulence are used instead of wall functions.

The combustion process is modeled through detailed tracking of the liquid fuel spray droplets, evaporation, and chemical reaction. Individual spray droplets are tracked using a Lagrangian trajectory algorithm, developed and described by Raju (2000). The droplet model includes physics models for droplet drag, heat, and mass transfer to the liquid through established correlations. Droplet heating is modeled in detail, including the effect of internal circulation within the droplet, and temperature variations within the droplet. Droplet initial conditions are obtained from fuel injector experiments, and droplet initial velocity, radius, and temperature fields are specified for each injector. In the present case, the orifice injectors have been measured to produce 90° conical spray droplet flow, and these results were used as the cavity and the main injector fueling initial conditions. Discrete liquid droplet streams are clocked at 10° intervals, originating at the external lip of the fuel injector orifice. The droplet size distribution is modeled using a correlation taken from El Bahawy and Whitelaw (1980) applied to each droplet stream. The liquid spray droplet model is fully coupled to the aerodynamics and combustion models in the flow solver through continuity, momentum, and energy source terms. Typically, the smaller diameter droplets evaporate and react first, followed by the larger droplets, which penetrate further into the flow field.

A CFD simulation of the speed contour provides a convenient overview of the main aerodynamic features of the TVC with tri-pass diffuser system, and is shown in figure 5. The plenum conditions for the CFD simulations to be discussed differ from those in the photos in terms of the air mass flow rates through the diffuser and the secondary flow (or cooling air) paths. In the experimental runs shown in the photos, the tested mass flow rate resulted in peak flow velocities corresponding to Mach numbers near 0.7 within the diffuser branches. This velocity increase is expected to impact the computed temperatures within the combustor. For example, at the higher Mach number flow rate run in the experiment, shear velocity between the cross flow and the liquid fuel droplet would be greater, resulting in a higher fuel evaporation rate, and, consequently, more rapid combustion and heat release. CFD simulations with a monodispersed droplet distribution (60 micron diameter) gave lower droplet evaporation rates than would be anticipated. As such, some fuel would be still bound in the liquid droplets, and left the domain un-reacted when compared to a similar CFD solution with gaseous fuel. A Rosin-Rammler droplet distribution (bell-shaped) and particle tracking were applied to the simulation to mitigate these effects.

In figure 5, the aerodynamics definition sketch, the box drawn around the combustor indicates the region shown in the photographs in figures 2, 3, and 4. The upper figure cross section A-A offset from the combustor center plane clearly shows the interactions of the in-pass diffuser airflows, with the cavity flow, with subsequent mixing in the downstream region. Air jets issuing into the upper and lower cavities are clearly seen, with rapid bending of the jets due to strong interactions with the combustion flow fields. Large-scale recirculation regions can be seen between the upper, middle, and lower branches of the in-pass diffusion system. These recirculation regions hold hot products of combustion, which enter along the cavity back wall. These recirculation zones form exceptionally stable combustion regions, and are aerodynamically stable over a wide range of fuel-air ratios and inlet temperatures and pressures.

The lower figures show cross sections of the combustor taken at three axial positions within the cavity: near wall (1-1), mid-plane (2-2), and downstream wall (3-3), as indicated by the cross section cuts above. The high-speed jets are clearly observed, as are the liquid fuel injection and the air jets issuing from the plenums. Although not easily observable in these views, significant amounts of slot and film cooling airflows are also entering into the cavity.

Results

Comparison of Combustor Simulations with Photographs

CASE 1 (Run 2311): Main and cavity fueling.—In Case 1, fuel was injected into both the main tri-pass branches and into the TVC cavities. The TVC cavity equivalence ratio was $\phi = 0.65$, the main stream equivalence ratio was $\phi = 0.73$, and the overall system equivalence ratio was $\phi = 0.53$, based on a summation of all airflow in the cavity, the main stream, and downstream sections. The experimental photograph for the case is shown in figure 2, representing a time- and path-averaged spanwise view through all 10 sectors of the combustor. The small differences in ϕ are related to cavity and cooling air split estimations. The thermal-flow structure is represented in the photo through the luminosity of the flame. While not directly proportional to temperature, the observed flame luminosity shows high intensity burning within the cavity, particularly along the cavity back wall, as well as the central portion of the combustor. The flow simulation results for Case 1 are shown in figure 6, in terms of cross sections through the combustor center plane (figs. 6(a) to (d), and axial cuts through the cavity (figs. 6(e) to (g)).

The model calculation confirms that combustion is intense within the cavity, and particularly along the center plane, as seen in figures 6 (a) and (b). Hot products of combustion are entrained from the cavity region into recirculation regions between the tri-pass diffuser branches, and also along the downstream wall of the cavity, which wrap around a 90° turn, mixing with the film cooling flow. In addition, with significant fueling into the main tri-pass diffuser passages, some combustion takes place in the near field of those flows. Most of the fuel entering through the main stream reacts further downstream in the straight duct, as seen by the hot streaks in figure 6. Strong mixing in the lateral or span wise direction takes place away from the center plane, with well-mixed high temperature flow throughout the straight duct. Interaction of the hot combustion gases with the liner film cooling and slot flow is clearly observed along the upper and lower portions of the combustor duct, flushed by a cool layer of fresh air.

The cross sections through the TVC cavity in figures 6(e) to (g) give further detail of the combustion flow structure. Near the cavity back wall, seen in figure 6(e), there is strong fluid coupling of the fuel jet with the upper, central air jet issuing from the plenum. This acts to enlarge the effective combustion area of the liquid jet stream. Figure 6(e) also shows the hot flow evolving from the outer tri-pass legs, indicated by the two hot spots. These are due to rapid combustion of the fuel as it enters the combustion chamber. Interestingly, the center passage of the tri-pass diffuser flow features no combustion at this plane, but instead, that flow experiences a significant combustion delay until that fuel-air mixture reaches the downstream duct. These overall features can also be observed in the luminosity photo of this case in figure 2. There is clearly a cooler zone in the central region, corresponding to the issuing of the center passage air-fuel stream, followed by a delayed combustion zone further downstream in the burner duct. Distinct combustion flow structures are seen in figure 2, including a hot streak due to flow from the upper tri-pass branch together with entrained flow from the cavity, and also the hot central jet which eventually reacts and combusts quickly once ignited. A distinct cool region also exists in the upper and lower right corners of the cavity, both in the experimental results and in the computational simulation. Also, intense burning along the back cavity wall is seen in the experiment and computational simulation.

CASE 2 (Run 2318): Reduced main fueling, increased cavity fueling.—In Case 2, fuel was again injected into both the trapped vortex cavity and through the tri-pass diffuser passages, but in different ratios. Inlet pressure and temperature were nearly the same as in Case 1. The cavity was richly fueled,

with local equivalence ratio of $\phi = 1.8$, the main stream local equivalence ratio of $\phi = 0.14$, and the overall system equivalence ratio of $\phi = 0.52$. The experimental photograph for the case is shown in figure 3, while the CFD simulation results for this case are shown in figure 7, in the same format as the cross sections in figure 6.

The combustion processes in both the cavity and in the central combustion zone are more varied than in Case 1. In the cavity, the rich fueling results in a strong interaction of the liquid fuel jet with the opposite cavity wall, with that fuel reacting quickly and intensely with incoming air flow from the cavity aft wall driver holes. The hot products of combustion associated with this interaction are purged rapidly into the downstream combustor section and interact with the slot and film cooling flow along the combustor duct. There is an indication that hot cavity flow comes in contact with the upper and lower cavity walls, despite significant film cooling and air jet flow. There is also a suggestion in the photo in figure 3 that this occurs in the rig as well.

The combustion process in the central combustor zone is also more varied than in Case 1. Larger combustion recirculation regions are present between the tri-pass diffuser outer and center passage flows, with hot flow also engendered from the tri-pass passages due to direct fueling. The shear layer formed between the cavity and tri-pass diffuser outer branch supports a high temperature gradient, separating the driven cavity flow and the high speed, slightly heated tri-pass jet flow. Rapid mixing of the combustion gases from the various sources, and with the film and slot cooling flows, occurs in the combustor duct further downstream.

CASE 3 (Run 2142): Cavity fueling only.—In Case 3, fuel was injected into the trapped vortex cavity only, with no fuel injected into the diffuser passages. For this case, the cavity was richly fueled at a local equivalence ratio of $\phi = 2.2$, while overall system equivalence ratio was maintained at $\phi = 0.53$. Inlet pressure and temperature were again nearly the same as in Case 1. The experimental photograph for Case 3 is shown in figure 4, while the CFD simulation results for the case are shown in figure 8, in the same format as for the previous cases.

The combustion process for this case forms a highly intense combustion zone that originates entirely within the cavity. As in the other cavity flow fields, the flame is attached to the edges formed by the spray cone and anchored along the periphery of the hole near the injector orifice. The flame is turned by the various jet and film cooling flows and wraps around the in-pass airflows, filling the recirculation regions between the outer, center, and inner passages of the tri-pass diffuser. These regions are also very hot, with products of combustion entrained from the cavities and the incoming in-pass diffuser air flows. As in the previous cases, hot products mix out well in the downstream burner duct.

Combustor experimental performance data.—To establish the engineering performance parameters required for evaluation of the combustor, data were obtained for NO_x emissions, combustion efficiency, and lean blow-out at different inlet temperatures, different fuel-air ratios, and different cavity-to-main-stream fuel splits. The majority of the measured data were obtained in the cavity-only fueled operating mode. The measured NO_x emissions have been corrected to standard day humidity levels.

Results for humidity-corrected EINO_x versus burner bulk cold flow residence time are shown in figure 9. These results indicate that a factor of three reduction in the cold flow bulk residence time can reduce NO_x emissions by 33 to 50 percent. The observed 50 percent reduction in NO_x emissions is in line with anticipated benefits of using TVC technology in advanced military gas turbine engine applications.

Combustion efficiency versus bulk cold flow residence time is shown in figure 10. High values of combustion efficiencies (>99 %) are achieved over almost the entire range of residence times, over different fuel-air ratios, and cavity-to-main fuel splits. The effect on measured combustion efficiency is significantly amplified for overall lean operation ($f/a = 0.012$) at low operating inlet temperature (500 °F).

Overall burner exit equivalence ratio at lean blow-out (LBO) performance versus burner cold flow reference velocity is shown in figure 11. Values of 0.1 are generally considered good performance for conventional swirl-stabilized combustion devices, which typically operate at reference velocities of 50 ft/sec or less. The TVC generally performs at or below this value at reference velocities up to a factor of three higher, with most data points between the 0.06 and 0.08 levels. Based on prior test experience

with this prototypical TVC test rig, it was anticipated that LBO performance would be exceptionally good for this device.

Summary and Conclusions

This paper described experimental testing and numerical simulation of a trapped vortex combustor rig with diffuser flows to local Mach numbers near 0.7, with a view towards understanding the fundamental aerodynamic and combustion processes under different fueling conditions and evaluation of engineering performance factors, such as EINO_x combustion efficiency and lean blow-out. Photographs of the combustor hardware and photographs of the observed flame luminosity under three differing cavity-to-main-stream fueling splits led to insights into flame structure and potential hot spots within the combustion zone. CFD simulation results for the liquid fuel spray droplet evaporation and combustion process showed in detail the complex aerodynamic flow interactions and captured the very different combustion processes at work in the combustor main and cavity zones. It was noted that the CFD runs with monodispersed droplet distributions (60 micron diameter) gave reduced droplet evaporation rates and, consequently, reduced overall combustion efficiency; the Rosin-Rammler droplet distribution (bell-shaped) was applied to mitigate these effects. Additional experimental testing and data reduction showed the baseline TVC configuration with tri-pass diffuser to be very promising with respect to key performance parameters, and in comparison to values for conventional, swirl-stabilized combustors. The TVC combustor is continuing to undergo testing at AFRL and is being systematically and vigorously tested over a wider range of fueling and inlet conditions. Additional design work is also underway in tailoring the cavity and main stream fuel injection features in ways that will enhance fuel-air mixing and further improve performance characteristics of the TVC class of combustors.

References

- Brankovic, A., Ryder, Jr., R.C., Sturgess, G.J., Lee, J., Kushari, A., Lubarsky, E., Zinn, B.T., 2001, Computational and Experimental Study of Aerodynamics and Heat Release in a Liquid Fueled Combustor, AIAA Paper 2001-0976.
- Burrus, D.L., Johnson, A.W., Roquemore, W.M., and Shouse, D.T., 2001, Performance Assessment of a Prototype Trapped Vortex Combustor Concept for Gas Turbine Application, ASME IGTI Paper 2001-GT-0087, ASME IGTI Turbo Expo 2001, 4-7 June 2001, New Orleans, Louisiana.
- El Banhawy, Y., and Whitelaw, J.H., 1980, Calculation of the Flow Properties of a Confined Kerosene-Spray Flame, AIAA J., vol. 18, no. 12, pp. 1503-1510.
- Raju, M.S., 2000, An Overview of the NCC Spray/Monte Carlo PDF Computations, AIAA Paper 2000-0337.
- Roquemore, W.M., Shouse, D., Burrus, D., Johnson, A., Cooper, C., Duncan, B., Hsu, K.-Y., Katta, V.R., Sturgess, G.J., and Vihinen, I., 2001, Trapped Vortex Combustor Concepts for Gas Turbine Engines. AIAA Paper 2001-0483. 39th AIAA Aerospace Sciences Meeting and Exhibit, 8-11 January 2001, Reno, Nevada.
- Ryder, Jr., R.C., 2000, Application of the National Combustion Code Towards Unsteady Mixing and Combustion Modeling, AIAA Paper 2000-0335.
- Ryder, Jr., R.C., and McDivitt, T., 2000, Application of the National Combustion Code Towards Industrial Gas Fired Heaters, AIAA Paper 2000-0456.

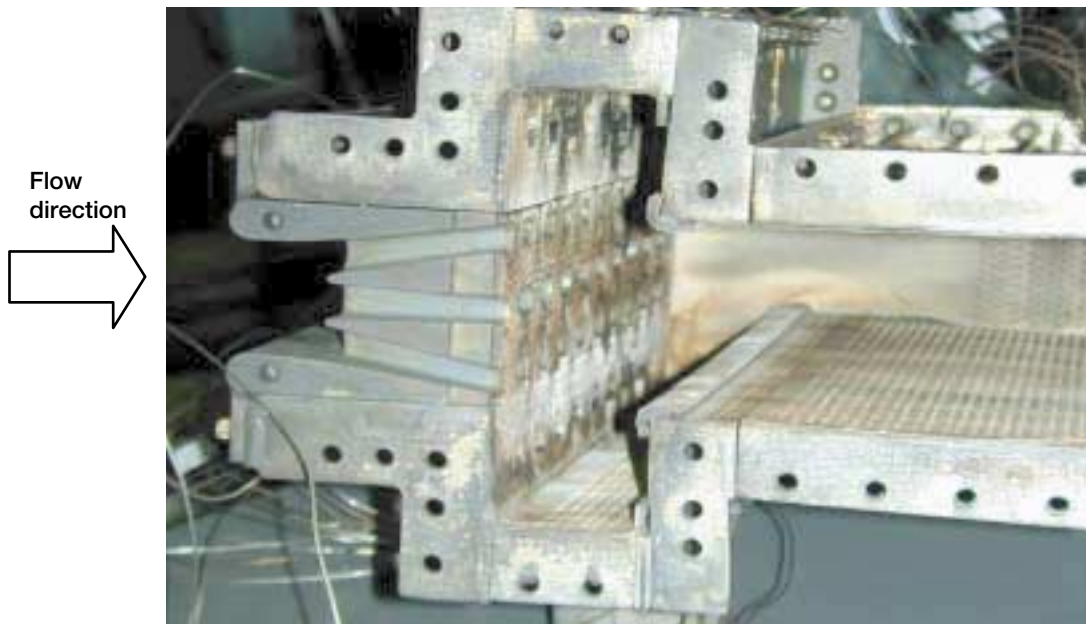


Figure 1.—Test apparatus photo for the Trapped Vortex Combustor with Tri-Pass diffuser system, featuring a rectangular array of 10 fuel injector modules in a planar combustor sector rig. The Near-field side wall has been removed for visualization. The fuel injectors can be seen on rear plate of the cavity. Carbon deposits are visible on the ceramic combustor coating, reflecting extensive hot-fired testing. Film cooling holes can be seen along the cavity walls, combustor core, and combustor liner walls.

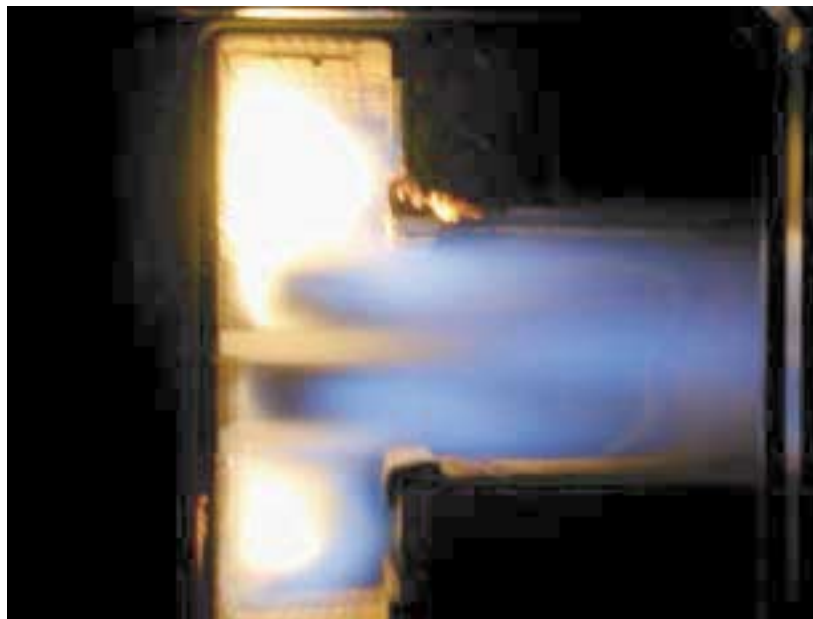


Figure 2.—Photo of the Trapped Vortex Combustor with Tri-pass diffuser, Run 2311. Inlet pressure 345 kPa (50 psia); fueling into cavity and main feed arms. Cavity $\phi = 0.65$, main feed arms $\phi = 0.73$, overall $\phi = 0.53$. $T_3 = 533 \text{ K}$ (500 °F); pressure drop = 56.4 kPa (8.17 psi). $[T_{\text{flame}}] = 3034 \text{ °F}$, fuel = liquid Jet-A, $[T_{\text{equil}}]_{\phi_{\text{cavity}}} = [3162 \text{ °F}]_{\text{JP8}} = [3068 \text{ °F}]_{\text{Jet-A}}$.

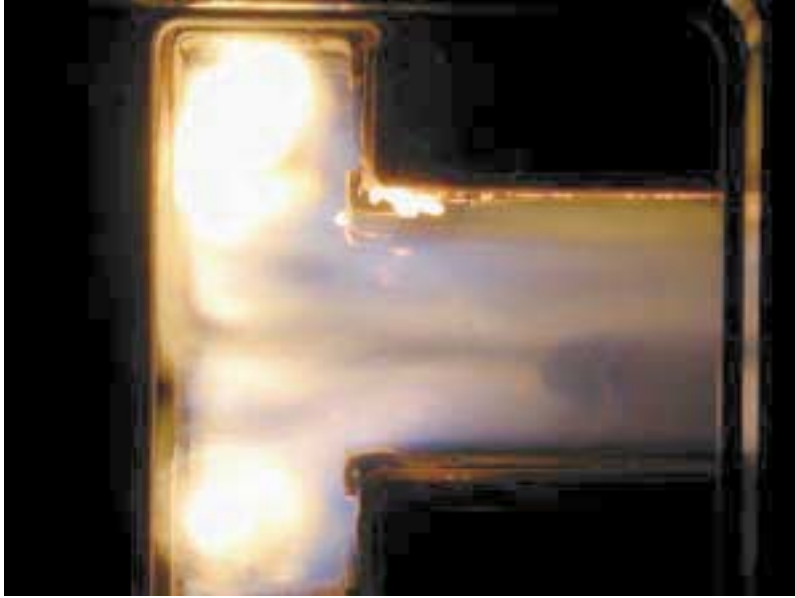


Figure 3.—Photo of the Trapped Vortex Combustor with Tri-pass diffuser, Run 2318. Inlet pressure 345 kPa (50 psia); fueling into cavity and main feed arms. Cavity $\phi = 1.8$, main feed arms $\phi = 0.14$, overall $\phi = 0.52$. $T_3 = 533$ K (500 °F); pressure drop = 53.3 kPa (7.72 psi). $[T_{\text{flame}}] = 2840$ °F, fuel = liquid Jet-A, $[T_{\text{equil}}]_{\phi_{\text{cavity}}} = [3068$ °F] $_{\text{JP8}} = [2836$ °F] $_{\text{Jet-A}}$.

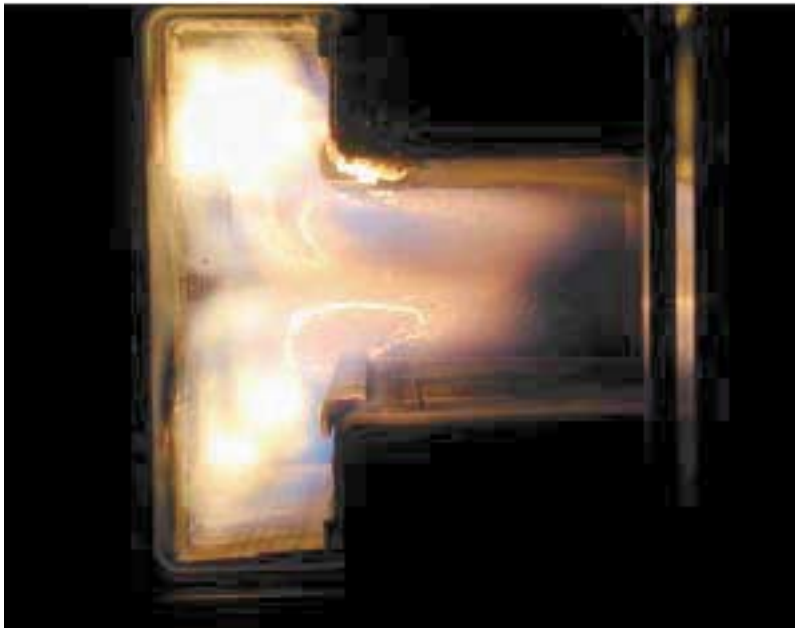


Figure 4.—Photo of the Trapped Vortex Combustor with Tri-pass diffuser, Run 2142. Inlet pressure 345 kPa (50 psia); cavity fueling only. Cavity $\phi = 2.2$, overall $\phi = 0.53$. $T_3 = 533$ K (500 °F); pressure drop = 47.3 kPa (6.86 psi). $[T_{\text{flame}}] = 2741$ °F, fuel = liquid Jet-A, $[T_{\text{equil}}]_{\phi_{\text{cavity}}} = [2730$ °F] $_{\text{JP8}} = [2465$ °F] $_{\text{Jet-A}}$.

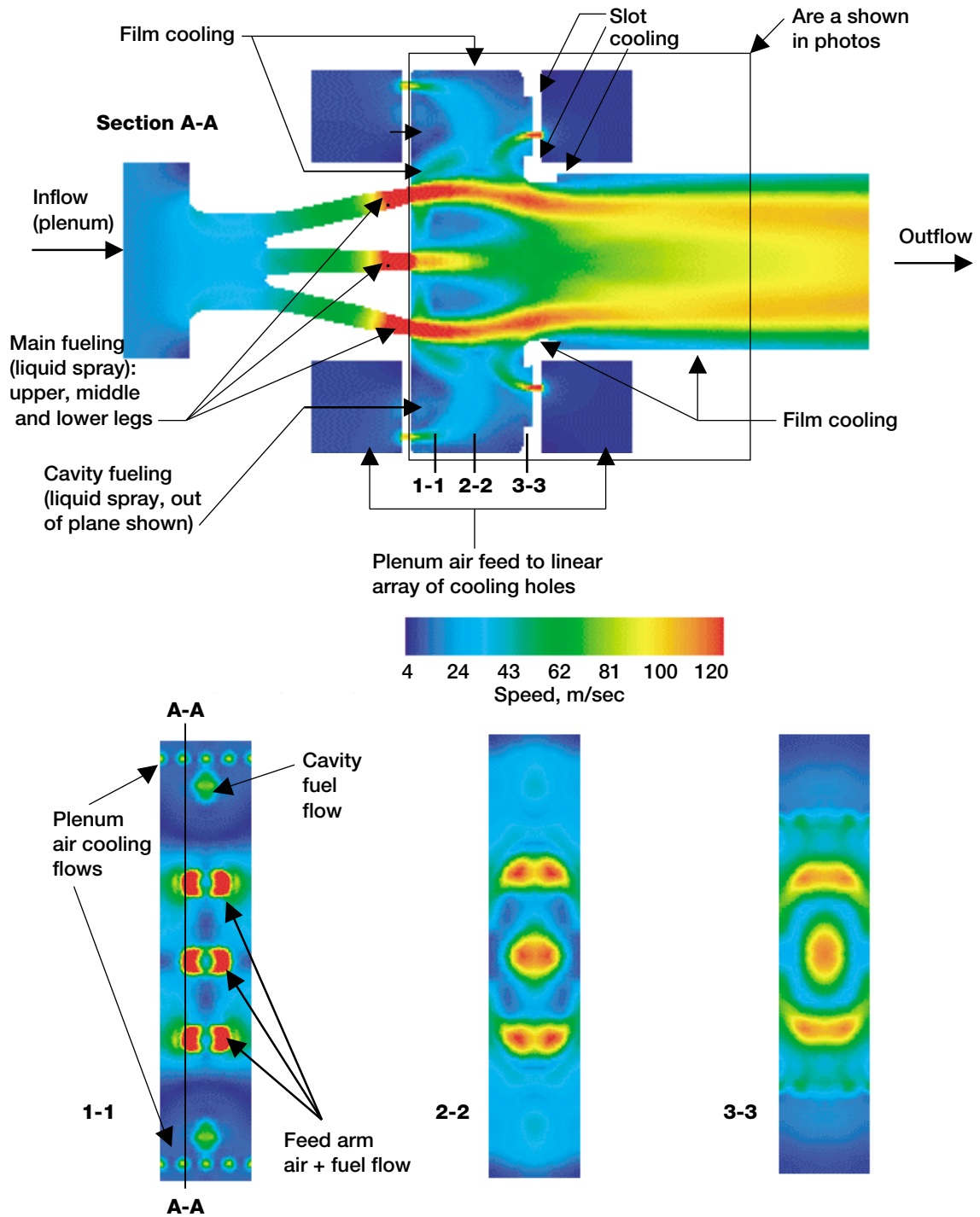
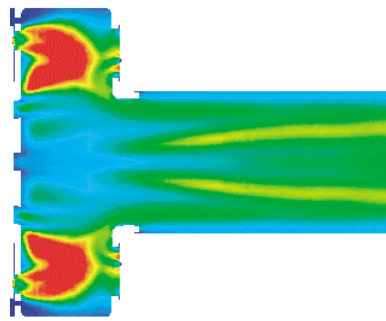
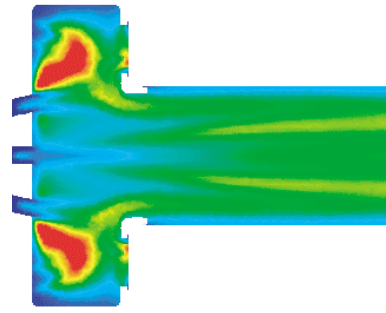


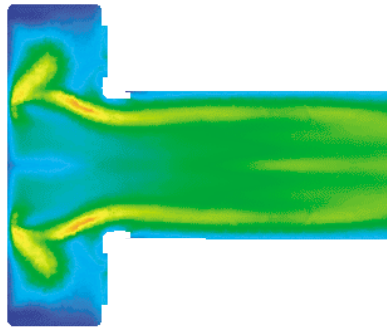
Figure 5.—Speed contours and definition sketch for CFD simulation of the trapped vortex combustor with tri-pass diffuser, showing air and fuel flows. Upper figure is a cross-section taken midway between combustor center-plane and end-wall. Lower figures represent cuts at three planes within the cavity.



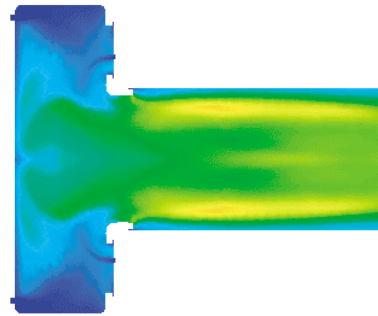
(a) Combustor center plane, $Z = 0.0$ m.



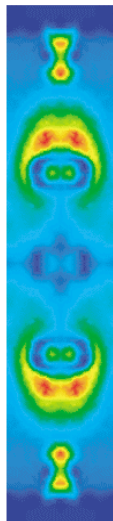
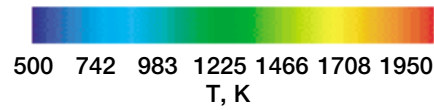
(b) $Z = 0.005$ m off center plane.



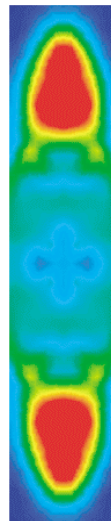
(c) $Z = 0.010$ m off center plane.



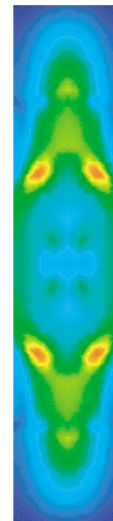
(d) $Z = 0.015$ m off center plane.



(e) $X = 0.001$ m off upstream cavity wall

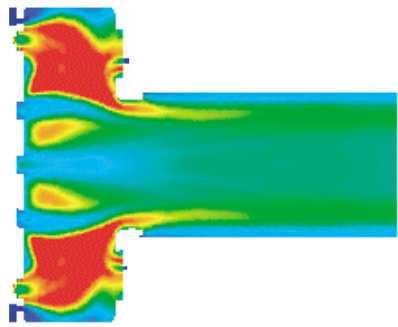


(f) Midway through cavity

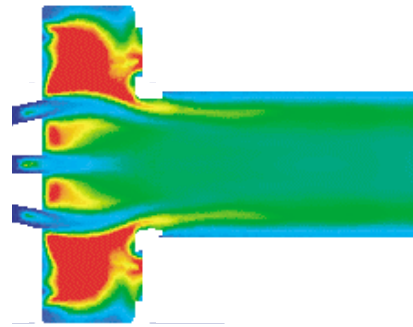


(g) $X = 0.001$ m off downstream cavity wall

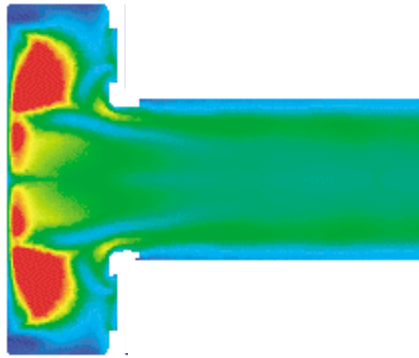
Figure 6.—Computed temperature contours for Case 1. (a)-(d) Lengthwise cuts through the combustor. (e)-(g) Axial cuts through cavity.



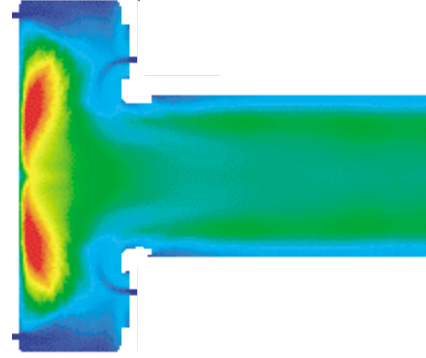
(a) Combustor center plane, $Z = 0.0$ m.



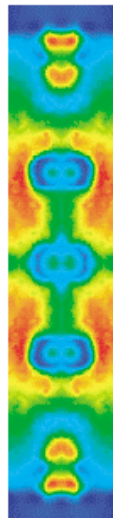
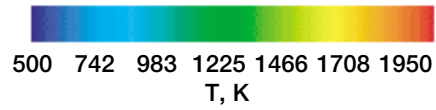
(b) $Z = 0.005$ m off center plane.



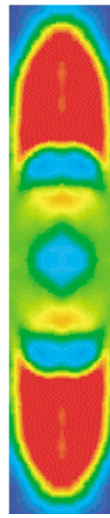
(c) $Z = 0.010$ m off center plane.



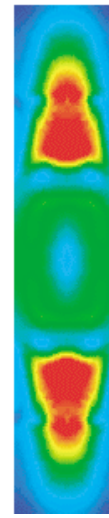
(d) $Z = 0.015$ m off center plane.



(e) $X = 0.001$ m off upstream cavity wall

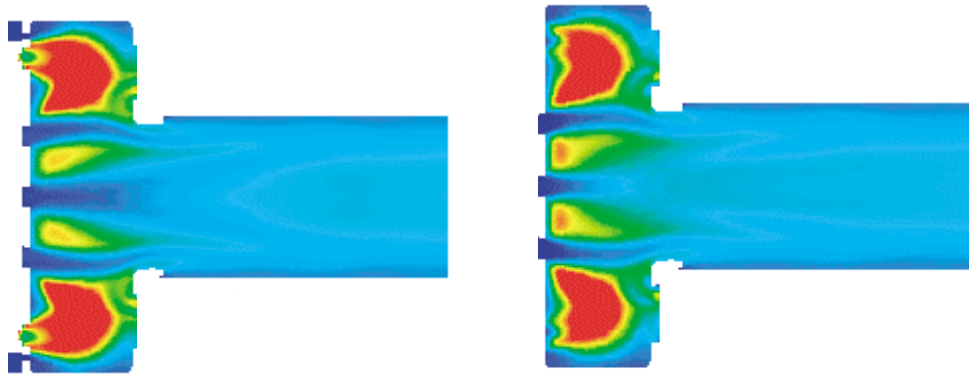


(f) Midway through cavity



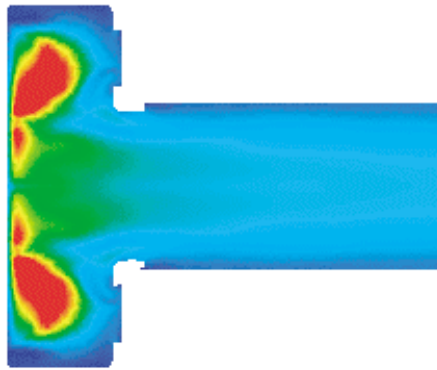
(g) $X = 0.001$ m off downstream cavity wall

Figure 7.—Computed temperature contours for Case 2. (a)-(d) Lengthwise cuts through the combustor. (e)-(g) Axial cuts through cavity.



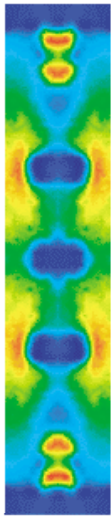
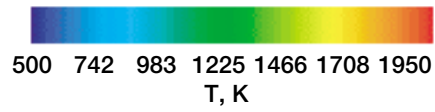
(a) Combustor center plane, $Z = 0.0$ m.

(b) $Z = 0.005$ m off center plane.

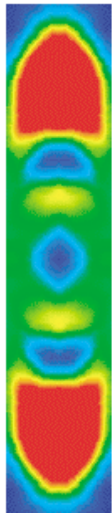


(c) $Z = 0.010$ m off center plane.

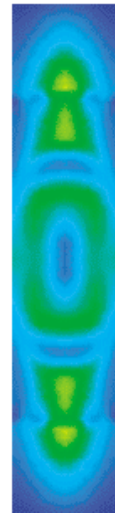
(d) $Z = 0.015$ m off center plane.



(e) $X = 0.001$ m off upstream cavity wall



(f) Midway through cavity



(g) $X = 0.001$ m off downstream cavity wall

Figure 8.—Computed temperature contours for Case 3. (a)-(d) Lengthwise cuts through the combustor. (e)-(g) Axial cuts through cavity.

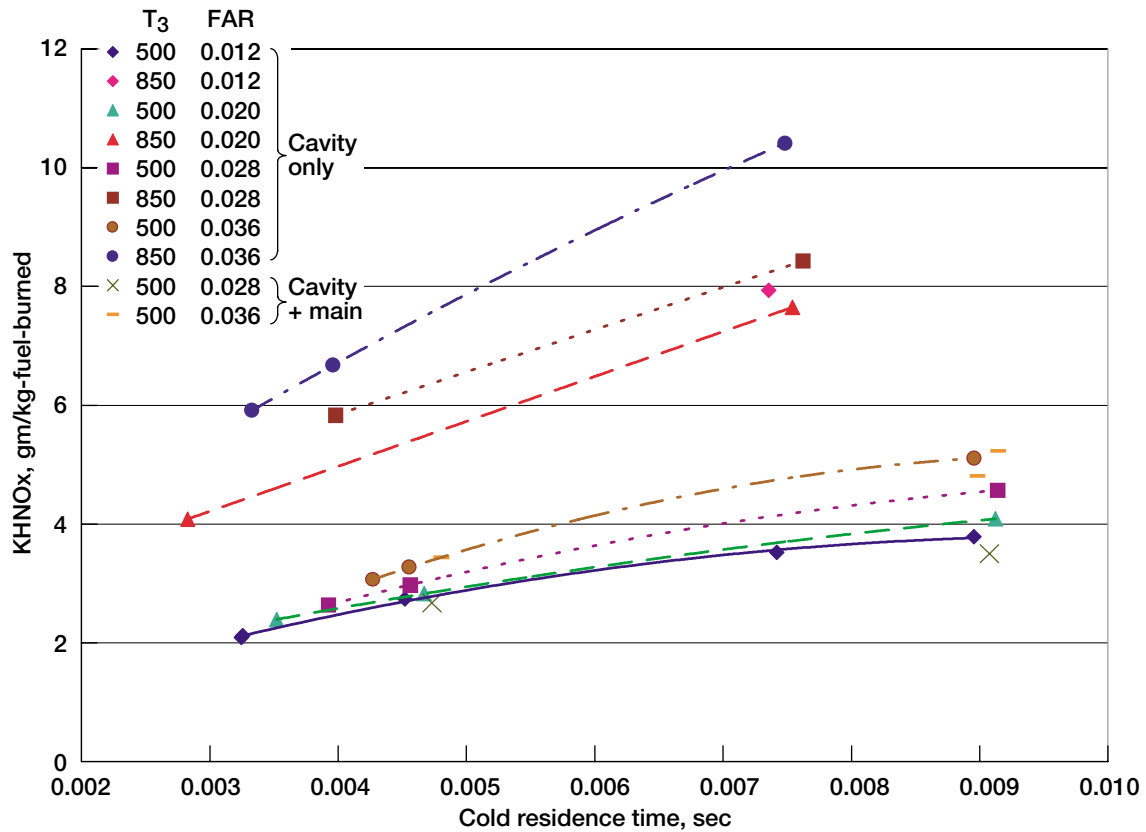


Figure 9.—Experimental TVC-Tri-pass data of humidity-corrected EINO_x versus burner bulk cold residence time, at various test conditions. Inspection of the plot reveals that NO_x can be reduced from 33% to 50% of baseline values, and that residence times can be controlled over a range of 2 to 3 times. NO_x is seen to be relatively insensitive to T₄. Pressure drop performance at P₄ is about 20% to 25% of P₃. Stable combustor operation observed over all parameter settings. FAR is fuel air ratio and T₃ is combustor air inlet temperature [(g/kg) EINO_x or KHNO_x ≈ (ppm NO_x) (1 + FAR)/(630 FAR)].

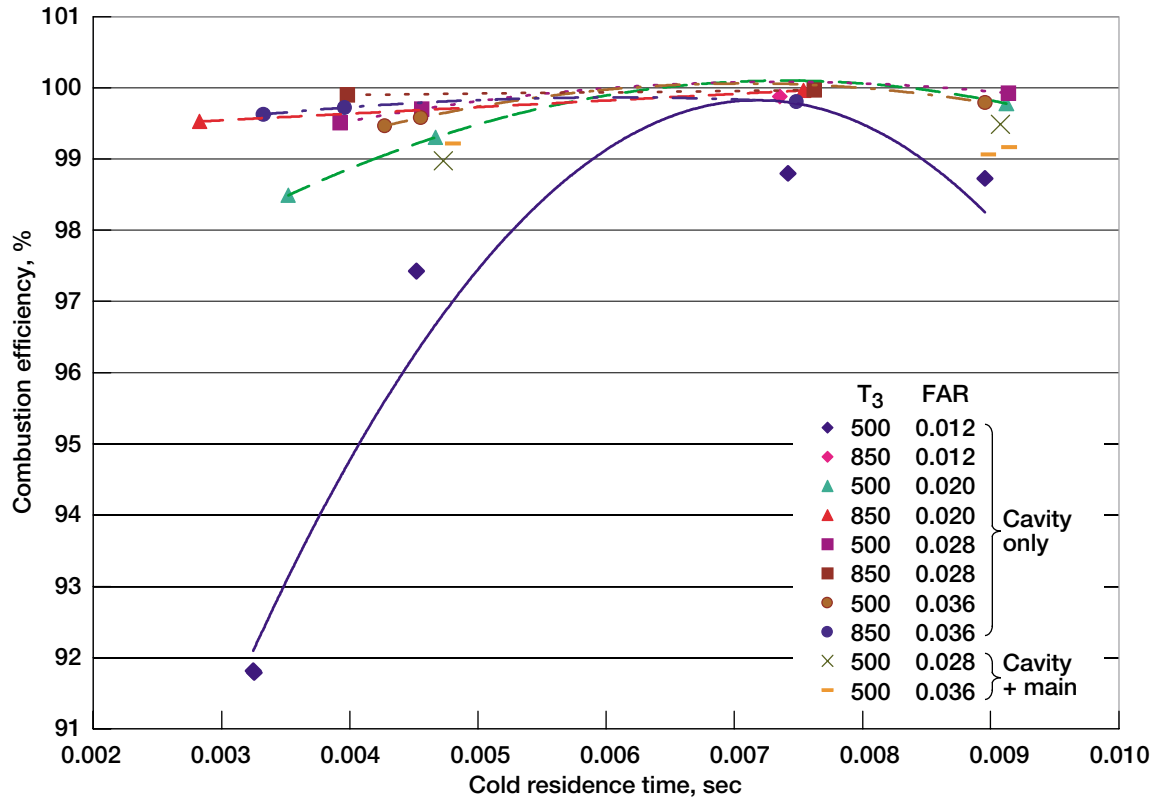


Figure 10.—Experimental TVC-Tri-pass data of combustion efficiency versus bulk cold residence time. Combustion efficiency decreases with residence time, significantly at low fuel-air ratios. For example, less than 98.5% for $f/a < 0.02$. Further measurements planned for the performance region of $0.012 < f/a < 0.02$. f/a FAR is fuel air ratio and T_3 is combustor air inlet temperature

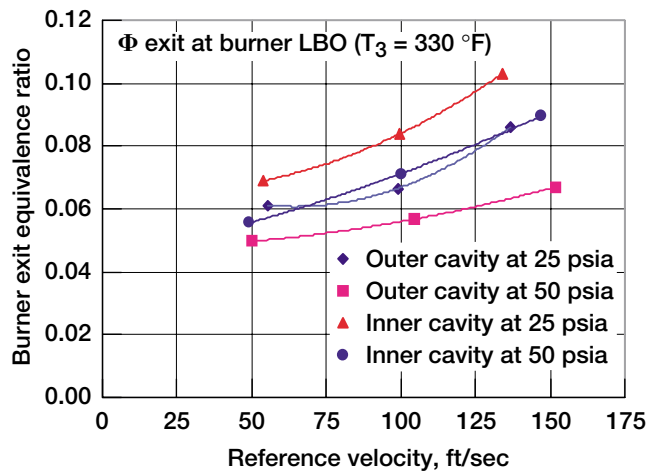


Figure 11.—Experimental TVC-Tri-pass data of combustor lean blow-out (LBO) versus reference velocity, at 172.5 kPa (25 psia) and 345 kPa (50 psia). For this case, cavity $\phi = 0.28$, overall $\phi = 0.359$. A value of 0.1 is considered good performance for conventional gas turbine combustor LBO performance. Trend observed is that LBO increases as reference velocity increases. T_3 is combustor air inlet temperature.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (<i>Leave blank</i>)		2. REPORT DATE January 2004	3. REPORT TYPE AND DATES COVERED Technical Memorandum	
4. TITLE AND SUBTITLE Experimental and Computational Study of Trapped Vortex Combustor Sector Rig With Tri-Pass Diffuser			5. FUNDING NUMBERS Cost Center 2250000013	
6. AUTHOR(S) R.C. Hendricks, D.T. Shouse, W.M. Roquemore, D.L. Burrus, B.S. Duncan, R.C. Ryder, A. Brankovic, N.-S. Liu, J.R. Gallagher, and J.A. Hendricks				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration John H. Glenn Research Center at Lewis Field Cleveland, Ohio 44135-3191			8. PERFORMING ORGANIZATION REPORT NUMBER E-12899-1	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA TM-2004-212507	
11. SUPPLEMENTARY NOTES R.C. Hendricks, N.-S. Liu, and J.R. Gallagher, NASA Glenn Research Center; D.T. Shouse and W.M. Roquemore, Wright-Patterson Air Force Base, Dayton, Ohio 45433; D.L. Burrus and B.S. Duncan, General Electric Aircraft Engines, Cincinnati, Ohio 45215; R.C. Ryder and A. Brankovic, Flow Parametrics, LLC, Bear, Delaware 19701; and J.A. Hendricks, Diligent Design, Toledo, Ohio 43614. Responsible person, R.C. Hendricks, organization code 5000, 216-977-7507.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Categories: 07, 28, and 64 Available electronically at http://gltrs.grc.nasa.gov This publication is available from the NASA Center for AeroSpace Information, 301-621-0390.			12b. DISTRIBUTION CODE	
13. ABSTRACT (<i>Maximum 200 words</i>) The Trapped Vortex Combustor (TVC) potentially offers numerous operational advantages over current production gas turbine engine combustors. These include lower weight, lower pollutant emissions, effective flame stabilization, high combustion efficiency, excellent high altitude relight capability, and operation in the lean burn or RQL modes of combustion. The present work describes the operational principles of the TVC, and extends diffuser velocities toward choked flow and provides system performance data. Performance data include EINO _x results for various fuel-air ratios and combustor residence times, combustion efficiency as a function of combustor residence time, and combustor lean blow-out (LBO) performance. Computational fluid dynamics (CFD) simulations using liquid spray droplet evaporation and combustion modeling are performed and related to flow structures observed in photographs of the combustor. The CFD results are used to understand the aerodynamics and combustion features under different fueling conditions. Performance data acquired to date are favorable compared to conventional gas turbine combustors. Further testing over a wider range of fuel-air ratios, fuel flow splits, and pressure ratios is in progress to explore the TVC performance. In addition, alternate configurations for the upstream pressure feed, including bi-pass diffusion schemes, as well as variations on the fuel injection patterns, are currently in test and evaluation phases.				
14. SUBJECT TERMS Combustion; CFD; Trapped vortex combustor; Emissions; Combustor performance			15. NUMBER OF PAGES 21	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	