



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.

## EFFECT OF AIR EXTRACTION FOR COOLING AND/OR GASIFICATION ON COMBUSTOR FLOW UNIFORMITY

T. Wang and J.S. Kapat<sup>1</sup>  
Gas Turbine Research Laboratory  
Department of Mechanical Engineering  
Clemson University  
Clemson, South Carolina, U.S.A.



W.R. Ryan, I.S. Diakunchak, and R.L. Bannister  
Power Generation Business Unit  
Westinghouse Electric Corporation  
Orlando, Florida, U.S.A.

### ABSTRACT

Reducing emissions is an important issue facing gas turbine manufacturers. Almost all of the previous and current research and development for reducing emissions has focused, however, on flow, heat transfer, and combustion behavior in the combustors or on the uniformity of fuel injection without placing strong emphasis on the flow uniformity entering the combustors. In response to the incomplete understanding of the combustor's inlet air flow field, experiments were conducted in a 48% scale, 360° model of the diffuser-combustor section of an industrial gas turbine. In addition, the effect of air extraction for cooling or gasification on the flow distributions at the combustors' inlets was also investigated. Three different air extraction rates were studied: 0% (baseline), 5% (airfoil cooling), and 20% (for coal gasification). The flow uniformity was investigated for two aspects: (a) global uniformity, which compared the mass flow rates of combustors at different locations relative to the extraction port, and (b) local uniformity, which examined the circumferential flow distribution into each combustor. The results indicate that even for the baseline case with no air extraction there was an inherent local flow nonuniformity of 10 ~ 20% at the inlet of each combustor due to the complex flow field in the dump diffuser and the blockage effect of the cross-flame tube. More flow was seen in the portion further away from the gas turbine center axis. The effect of 5% air extraction was small. Twenty-percent air extraction introduced approximately 35%

global flow asymmetry diametrically across the dump diffuser. The effect of air extraction on the combustor's local flow uniformity varied with the distances between the extraction port and each individual combustor. Longer top hats were installed with the initial intention of increasing flow mixing prior to entering the combustor. However, the results indicated that longer top hats do not improve the flow uniformity; sometimes, adverse effects can be seen. Although a specific geometry was selected for this study, the results provide sufficient generality to benefit other industrial gas turbines.

### INTRODUCTION

Reducing emissions is an important issue facing gas turbine manufacturers. Almost all of the previous and current research and development for reducing emissions has focused, however, on the flow, heat transfer, and combustion behavior in the combustors or on the uniformity of fuel injection without placing strong emphasis on the flow uniformity entering the combustors. In laboratories, combustion research has been frequently conducted using uniform flow as the inlet condition. Therefore, the factor of nonuniform combustor inlet flow distribution has not been intensively investigated. The extent of flow uniformity at the inlet to the combustors has been identified as an important factor which significantly affects the emission (Lyons, 1981). The cause of the nonuniform combustor inlet flow of many gas turbines can be traced to the complex flow field inside the dump

<sup>1</sup>J. Kapat is currently at the University of Central Florida

diffuser.

The heavy-frame industrial gas turbines designed by several major manufacturers typically use a two-part diffuser-combustion section to decelerate air exiting the compressor. First, air from the compressor flows through an annular pre-diffuser and recovers some of its kinetic energy before being discharged into a comparatively large chamber called the dump diffuser or the dump chamber. The diffuser-combustor section has two major functions (i) to decelerate the high velocity air coming from the compressor in order to provide an adequate air velocity for the combustion process (Wilson, 1984) and (ii) to distribute air uniformly to the combustors to reduce the non-homogeneity of the combustion process and therefore reduce the level of the exhaust emissions. For aircraft engines and small industrial gas turbines, the diffuser-combustor section adopts an in-line design where air from the dump chamber flows around the dome of an annular combustor (e.g. Fishenden and Stevens, 1977; Stevens et al., 1978; Carrotte et al., 1993). As a result of the in-line geometry, air is distributed relatively uniformly to the combustor. However, overall machine length impacts the turbine package size and rotor dynamics and thus is an important design issue in heavy-frame industrial gas turbines. To minimize machine length, reverse-flow diffusers with can-annular combustors are generally used. In reverse-flow diffusers, air from the annular pre-diffuser turns approximately  $150^\circ$  before entering the combustors and must maneuver around the combustor and the transitional piece. As a result, the flow characteristics of these diffusers are entirely different from those of in-line diffusers.

Relatively few experimental studies have been performed on dump diffusers with flow reversal. In an earlier study, the authors experimentally investigated, in the same facility as used in this study, detailed three-dimensional flow characteristics and aerodynamic performance in the diffuser-combustor section (Kapat, Wang, et al., 1996a). They discovered that most flow tended to move toward the outer part (away from the gas turbine center axis) of the dump diffuser. As a consequence, more flow entered the combustor from the outer portion (away from the gas turbine center axis) of each combustor. A concurrent CFD study conducted by Zhou, Wang, and Ryan (1996) provided a more detailed flow pattern, which helped to improve the understanding of the extremely complex 3-D flow interactions between the dump diffuser and the combustor inlet flow characteristics. The experimental results, although not as detailed as the CFD results, were necessary for verifying and interpreting the CFD results. The nonuniform flow distribution, discovered by both experimental and computational studies, prevailed in all the combustors. In this study, this is identified as the "local nonuniformity" since it considers non-uniformity in the local flow field entering each individual combustor rather than the bulk distribution of mass flow to all of the combustors.

As air extractions are needed for cooling and/or for Integrated Gasification of Combined Cycle (IGCC), the flow characteristics become more complex and non-axisymmetric. Kapat, Agrawal, and Yang (1994) have reported the effect of air extraction on diffuser performance for another manufacturer's industrial gas turbines. They reported that use of a single port for extracting air adversely affects the cooling of the transition pieces of the gas turbine. It was discovered that some cooling air is sucked outward from the transition piece cooling holes by the extraction force instead of impinging on the

transition pieces as originally designed. Later, Kapat, Wang et al. (1996b) conducted experiments in the same facility used in this study and discovered that there was approximately a 35% mass flow nonuniformity occurring between the combustor located near the extraction port and away from it. This nonuniformity of overall air flow between combustors produced by single-port air extraction is identified as "global nonuniformity" in this paper.

Evidence both from the inherent local nonuniformity due to the design of the existing diffuser-combustor configuration and global nonuniformity generated by air extraction motivated the present study to investigate the local flow distribution at the combustors' inlets.

Three different conditions were studied: 0% (baseline), 5% (for airfoil cooling), and 20% (for coal gasification). The air extractions were made at the dump diffuser outer wall (sometimes referred to as the combustor shell). The flow uniformity was investigated for two aspects: (a) global uniformity that compared the mass flow rates of combustors at different locations relative to the extraction port and (b) local uniformity that examined the circumferential flow distribution of each combustor including the local flow distribution surrounding each of the swirlers at the combustor inlet.

Although a specific geometry was selected for the present study, the results should provide sufficient generality for improving understanding of the complex flow behaviors in the reverse-flow-type diffuser-combustor sections of industrial gas turbines with/without air extraction.

## EXPERIMENTAL PROGRAM

### Test Facility

The desired air flow through the test model was provided by an open-circuit, suction-type wind tunnel. The overall layout is shown in Fig.1. Air from the test section went into a plenum box that was maintained at  $-40$ " H<sub>2</sub>O gage pressure by a suction fan rated at 15.6 m<sup>3</sup>/s (33,000 cfm) at 9.65 kPa (1.4 psi) and driven by a 220 hp motor. The plenum box isolated the test section from vibrations or oscillations of the fan and provided the necessary work space for installing a probe traversing system and changing the instrumentation. A detailed description of the experimental facility is provided in Kapat, Wang, et al. (1996a).

### Test Section

Fig. 2 shows a sectional view of the test section. A 48% sub-scale, 360° model of the diffuser-combustor section of a developmental heavy-frame gas turbine was constructed. The selection of a 360° model was necessary to investigate the effect of asymmetric air extraction (for cooling or IGCC) on the flow patterns in this study.

Before entering the model, the flow traveled through a straight annular section used to condition the flow. At the end of the developing section, the flow entered the annular pre-diffuser where it decelerated before being discharged into the dump diffuser.

After entering the dump diffuser, the air turned approximately  $150^\circ$ , maneuvered around the transition pieces and combustors, and entered the annular passage between the combustor and the top hat (Fig.2). The annular passage in the top hat helped to distribute the flow before it turned approximately  $180^\circ$  and entered the combustor. After passing

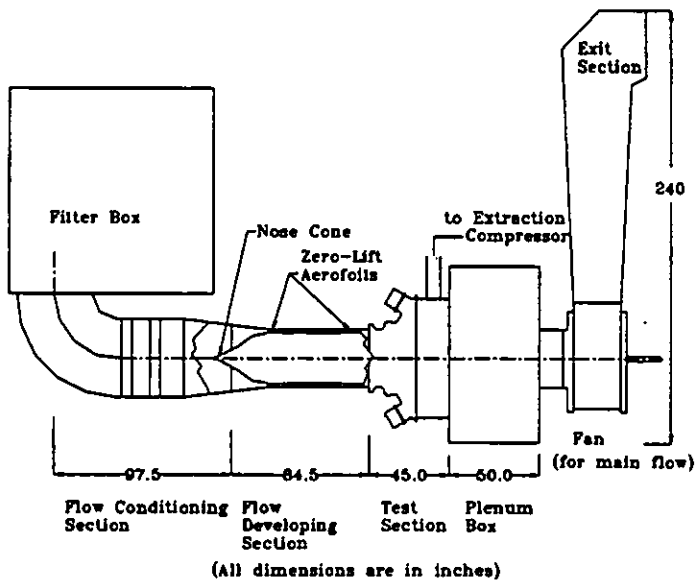


Fig. 1 Layout of test facility

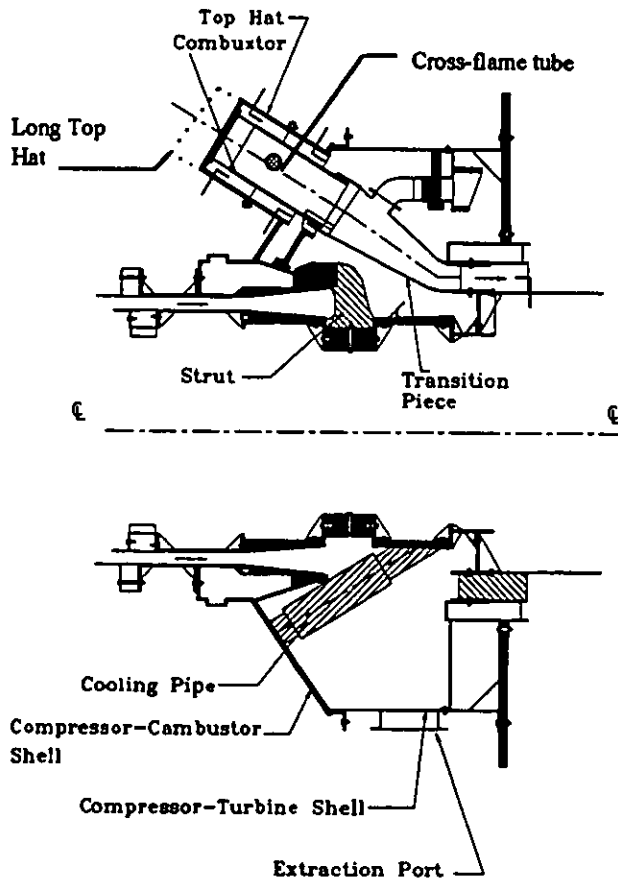


Fig. 2 Sub-scale test model

through the combustor and the transition piece, the flow was exhausted into a plenum.

The dump diffuser in this test model contained all the components in the actual engine including the cooling pipes and support struts. The annular passages in the top hats around the combustors contained the combustor cross-flame tubes with their complex angles to the flow maintained. In the actual engine, cross-flame tubes are used to sequentially ignite the combustors by circumferentially conducting flame through the tubes. The transition pieces were vacuum-formed to match the outer contours of the actual engine transition pieces. Cooling holes were drilled in the transition pieces such that the fraction of cooling air flow entering the transition piece lining was identical to that expected in the actual engine. Geometric similarity between the prototype and the model was maintained everywhere. One of the combustors was a true scaled model fabricated with all the swirlers and fuel injectors in place. The other fifteen mock combustors were fabricated with a circular orifice being placed at the combustor inlet. The diameter of the orifice for each combustor was calibrated to render the same drag coefficient as the "true scaled" model at various flow speeds. Detailed information on the test section and calibration of the combustor pressure loss coefficient is described in Kapat et al. (1996a).

#### Top Hat Length

To investigate if more room in the top hat area can promote better flow mixing, longer top hats with twice the mixing space of the existing design were installed for a portion of the experiment. The mixing space is the space between the end of the top hat and the inlet of the combustor.

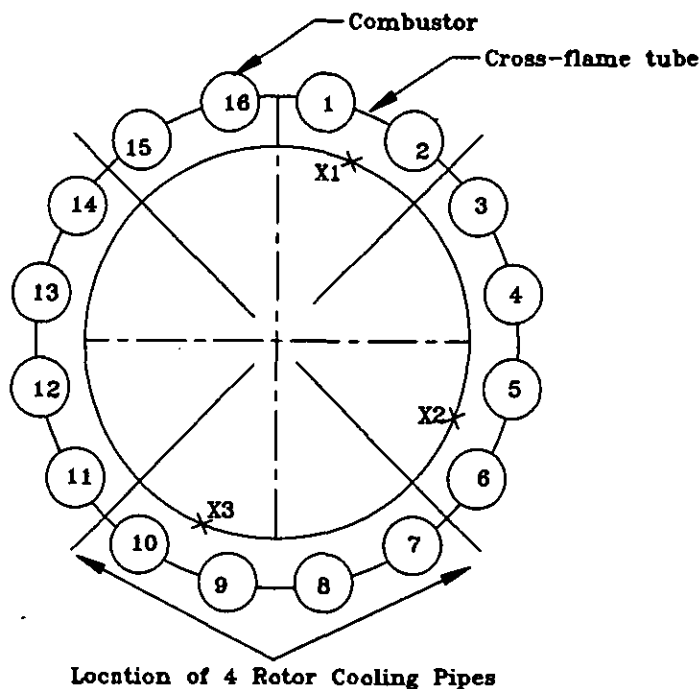
#### Air Extraction

The extraction ports were located on the outer casing around the dump diffuser as indicated in Figs. 1 and 2. Three different extraction ports (X1, X2, and X3) situated 90° apart circumferentially were built into the test section (see Fig. 3). Only one of these ports was used at a time for extraction. By extracting air at different circumferential locations while taking measurements at the same locations, circumferential variation of the effects of extraction was studied with minimal instrumented sites.

The extraction duct was equipped with a pitot-static probe that was used to monitor the air flow rate through the extraction duct. The desired flow rate through the extraction duct was achieved using both the extraction and the bypass ducts (Fig. 4). Small adjustments in the flow rate were performed by adjusting the extraction valve. For large variations, however, the bypass valve was used to maintain the total flow rate through the extraction compressor within the operating range. The extraction compressor is comprised of six stages and was rated at 2.36 m<sup>3</sup>/s (5,000 cfm) at 0.34 bars (5 psia).

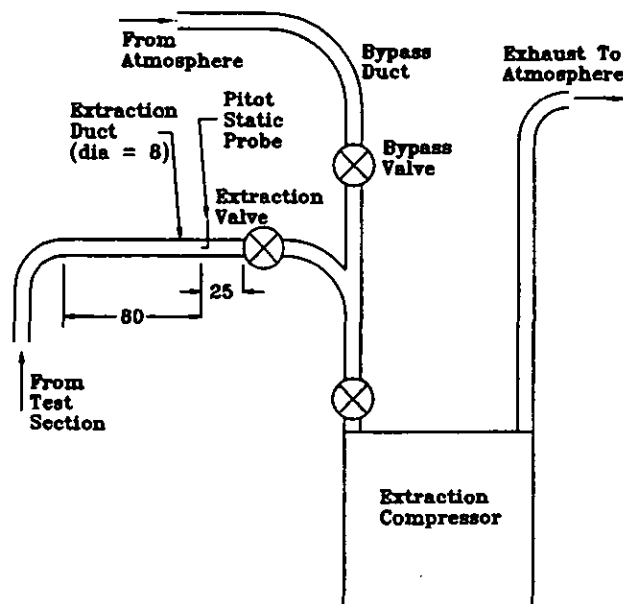
For different combination openings of the extraction and/or bypass valves, the pitot-static probe was traversed inside the duct, and velocity profiles were obtained. By integrating the profiles, the corresponding flow rates were calculated.

the extraction flow rate affected the main flow rate, the main flow needed adjustment again, and the process was repeated until both main and extraction flow rates were at the desired values. The process was repeated for each of the other two extraction ports.



Location of 4 Rotor Cooling Pipes

Fig. 3 Circumferential layout of air extraction locations X1, X2, and X3. Each circle represents the inlet cross-section of a combustor and is shown in detail in Fig. 6



(All dimensions are in inches)

Fig. 4 Bypass duct for controlling the air extraction rate

During the main experiments with 5% or 20% extraction, one of the three extraction ports X1, X2, and X3 was connected to the extraction duct. First, the main flow through the test section was adjusted with the extraction valve closed. Then, the extraction flow rate was obtained by adjusting the extraction and/or bypass valves. Since the adjustment of

### Experimental Cases

Each experimental case was assigned a four digit number. The meaning of each digit is as follows:

**1st digit:** "S" or "L" represents short or long top hats. **2nd digit:** "0, 5, or 2" represents zero, 5%, or 20% air extraction respectively. **3rd digit:** (i) for the baseline case, it represents the number of test runs, eg. 1 is the first run and 2 is the repeated run; (ii) for the extraction cases, it represents the location of extraction ports X1, X2, or X3. **4th digit:** represents the instrumented combustor 1 or 2 which is located away or near the rotor cooling pipe respectively (see combustor 1 or 2 in Fig. 3). For example, S012 represents short top hat baseline case (0% air extraction) at first test run for combustor number 2 that is located adjacent to the rotor cooling pipe. S022 is the repetitive run of S012. L232 represents long top hat case with 20% air extraction applied at extraction port X3 for combustor 2. There were a total of 24 test cases conducted.

### Instrumentation and Measurements

A custom-made tungsten hot wire, 2.5  $\mu\text{m}$  in diameter, was used to measure the flow distribution at the combustor inlet. The measurements were performed in the only combustor which was a true scaled model of the prototype containing eight swirlers housed within eight circular ducts through which combustion air has to flow before mixing with the fuel. In the true scaled combustor model, fuel nozzle models were fabricated with plexiglas cylindrical rods, which were situated at the middle of the cylindrical duct. Only one of the eight fuel nozzles was instrumented with a hot wire sensor which was placed approximately 4 mm from the fuel nozzle surface and 5 mm upstream of the corresponding swirler as shown in Fig. 5. The fuel nozzle can be rotated around its own axis so the hot wires can measure at any circumferential angle. For each swirler location eight angular locations at a 45° interval (numbered 1 to 8 in Fig. 6), were measured. The hot-wire measurements were performed one at a time at each of the eight swirler ducts. Similar measurements were then made on the rest of the seven swirlers by rotating the fixture that held the eight fuel nozzles to the appropriate locations. Each test case was completed when 64 measurements were made. For the first case (S011), the measurements were repeated at two different sampling rates: 2000 Hz for three seconds and 200 hz for 30 seconds. The difference of the mean velocities between the two sets was less than 2%. For this reason, in the rest of the cases, measurements were taken at 200 Hz for 30 seconds.

A constant temperature hot-wire anemometer, IFA-100, made by TSI, Inc. was used. The hot-wire sensor was calibrated against a pitot-static tube in a separate calibration wind tunnel over similar ranges of air velocities. The temperature difference between the calibration and operating conditions was corrected during data reduction process.

The overall mass flow rate entering each swirler in Fig. 6 was

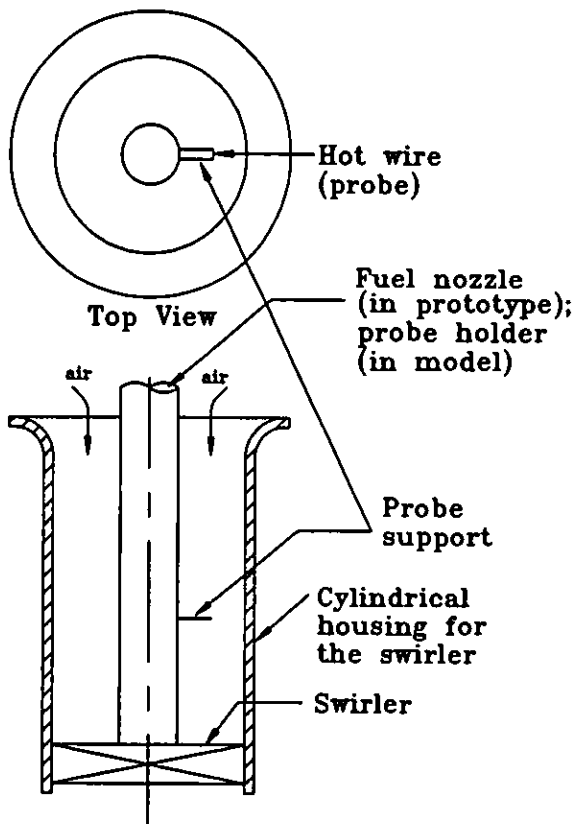


Fig. 5 Instrumentation of hot-wire sensor in the fuel nozzle-swirler assembly

calculated by integrating eight circumferential measurements and the overall mass flow rate of each combustor was calculated by adding the mass flow rate of each swirler. Fig. 6 is drawn as the flow enters the paper. The top of Fig. 6 is away from the turbine axis.

#### Experimental Uncertainty

For each of the baseline cases, at least two replications were performed. The third replications were further performed for cases S011, S021 and L011. Uncertainty due to replications for different locations varied from 1.5% to 8% with the average at 3.5%.

#### RESULTS AND DISCUSSION

**Short top hat, no air extraction (baseline cases)** — The flow distribution results of S012 are shown in polar coordinates in Fig. 7 with an arrangement matching that of Fig. 6. The scale of each concentric circle is in velocity (ft/s) on the left-hand side of each polar diagram. Each polar diagram represents the local flow distribution around a swirler. Integration of the mass flow rate of each swirler gave the total mass flow rate of that swirler and was represented as a single dot in the central polar diagram in terms of fraction of total mass flow rate of the combustor. The central polar diagram represents the overall mass flow distribution in the combustor. Since the density and representative area of each point are the same, the velocity is shown

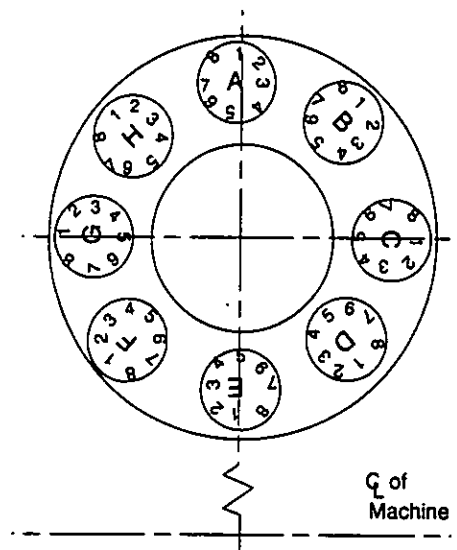


Fig. 6 End view of combustor inlet showing hot-wire measurement locations at inlet of each of the eight swirlers

in the figures instead of mass flow rate.

It can be seen in Fig. 7 that swirlers A and H had better uniformity than other swirlers. Swirler E had the worst nonuniform distribution. Most swirlers had more flow in the inner portion; this was specifically true for swirlers C, D, and E. The overall distribution, shown in the central diagram, indicated that there were approximately 20% more flow in the region covered by swirlers G, H, A, and B (upper-left half circle from 225° to 45°) than the lower-right half circle covered by swirlers C, D, E, and F. This case (S012) was measured at combustor 2, which is closer to the rotor cooling pipe. The effect of rotor cooling pipe to flow distribution can be obtained by comparing the results with those of case S011. The results of S011 had a similar trend in local (not shown) and overall distributions (Fig. 8a) as S012 (see Fig. 7) but approximately one half less nonuniform in overall distributions than S012.

**Short top hat, 5% air extraction** — The effect of 5% air extraction for case S512 was shown in Fig. 9. Comparison between Figs. 7 and 9 did not reveal much difference, except that swirlers A and B had approximately 3% less flow than the baseline case. This indicates that the effect of 5% air extraction on the combustor flow distribution is negligible. The similarity of the local distribution of corresponding swirlers between Fig. 7 and 9 also validated a good repeatability of the test results within a span of three months. Although the effect of 5% air extraction is not significant, other 5% air extraction cases, for example, case S531, with air being extracted at port X3 that was located diametrically from the combustor did show some variations on local flow distributions of swirlers D and E (not shown). This indicated that the air extraction could marginally affect the inner part of the combustor (i.e. swirlers D and E) diametrically across the dump diffuser. However, the overall combustor flow

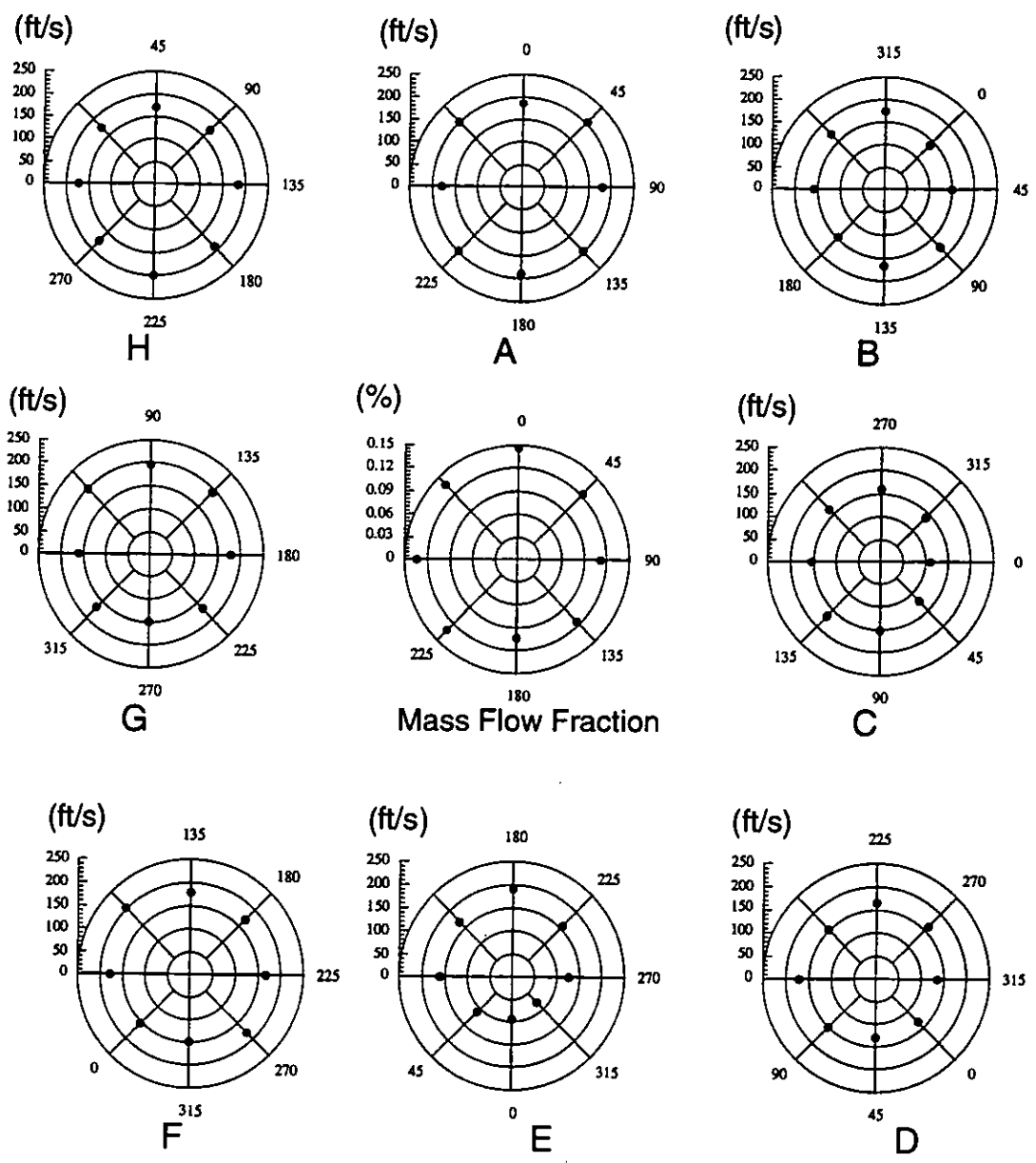


Fig. 7 Local and overall combustor inlet flow distribution of case S012 (short hat, no extraction, test 1 at combustor 2)

distribution of case S531 (Fig. 8b) is similar to case S512 (central diagram in Fig. 9).

**Short top hat, 20% air extraction** — The effect of 20% air extraction was significant, which can be seen by comparing case S212 in Fig. 10 with the baseline case S012 in Fig. 7. Case S212 had air extraction employed at port X1 located close to combustor 2 near one of the rotor cooling pipes. The most obvious difference is that flow is significantly retarded in the outer portion (the quadrant  $45^{\circ}$ - $0^{\circ}$ - $315^{\circ}$ ) of every swirler. For example, the velocities at  $0^{\circ}$  and  $45^{\circ}$  reduced from

135 ft/s at swirler D in Fig. 7 to approximately 100 ft/s in Fig. 10; similarly, at the same quadrant of swirler F, velocities were reduced from 145 ft/s in Fig. 9 to 95 ft/s in Fig. 10. Although the 20% air extraction significantly distorted the local distribution of each swirler, the nonuniformity of overall mass flow rate distribution (central diagram in Fig. 10) was about the same as that in the baseline case shown in Fig. 7.

The effect of the rotor cooling pipe can be seen by comparing the results of combustor 1 of case S211 (Fig. 11a) with those of case S212 (Fig. 10). The comparison indicated that the rotor cooling pipe had

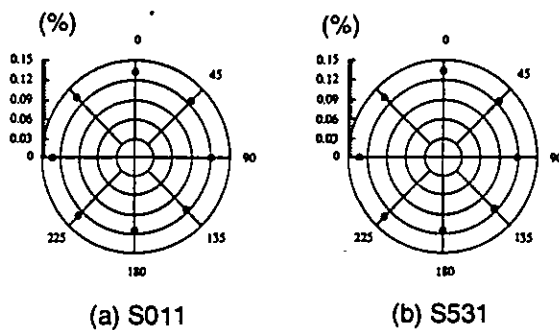


Fig. 8 Overall combustor inlet flow distribution for cases S011 and S531

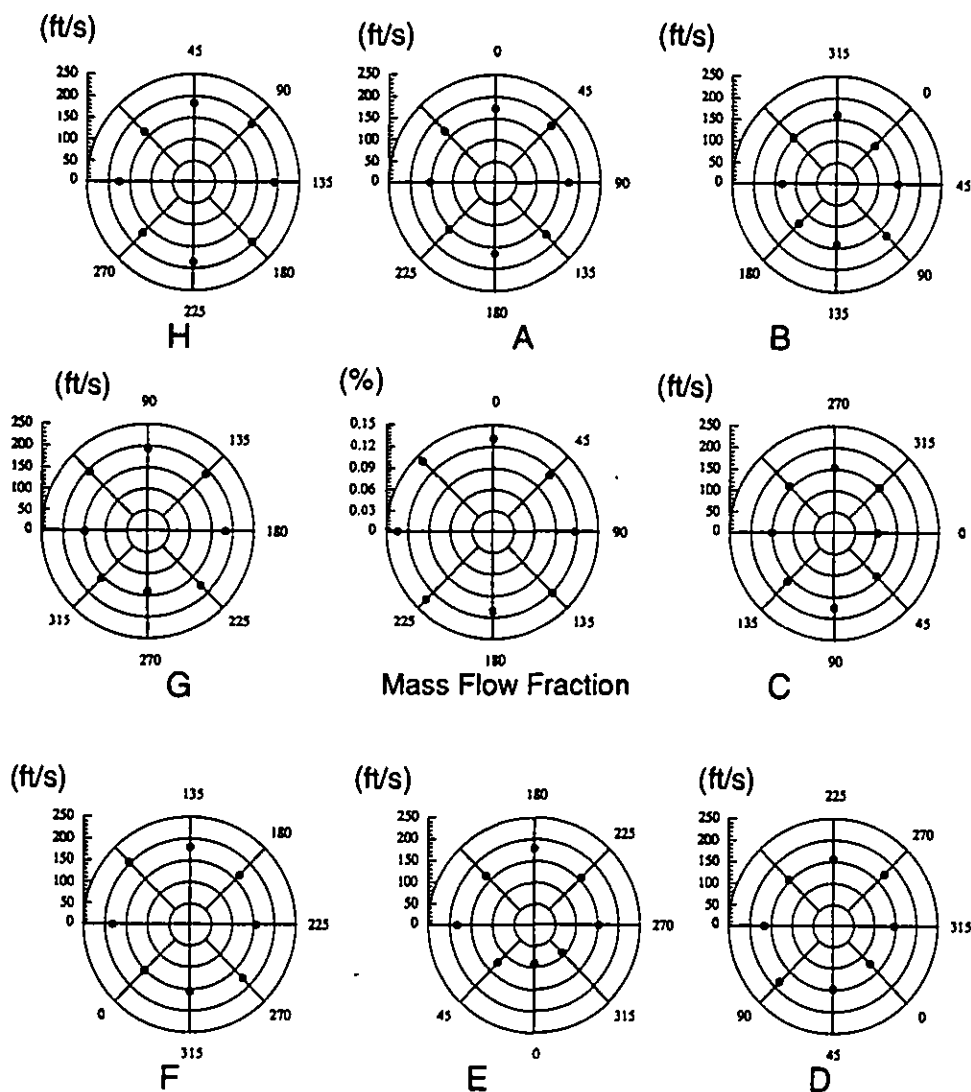


Fig. 9 Local and overall combustor inlet flow for case S512 (short hat, 5% extraction, test 1 at combustor 2)

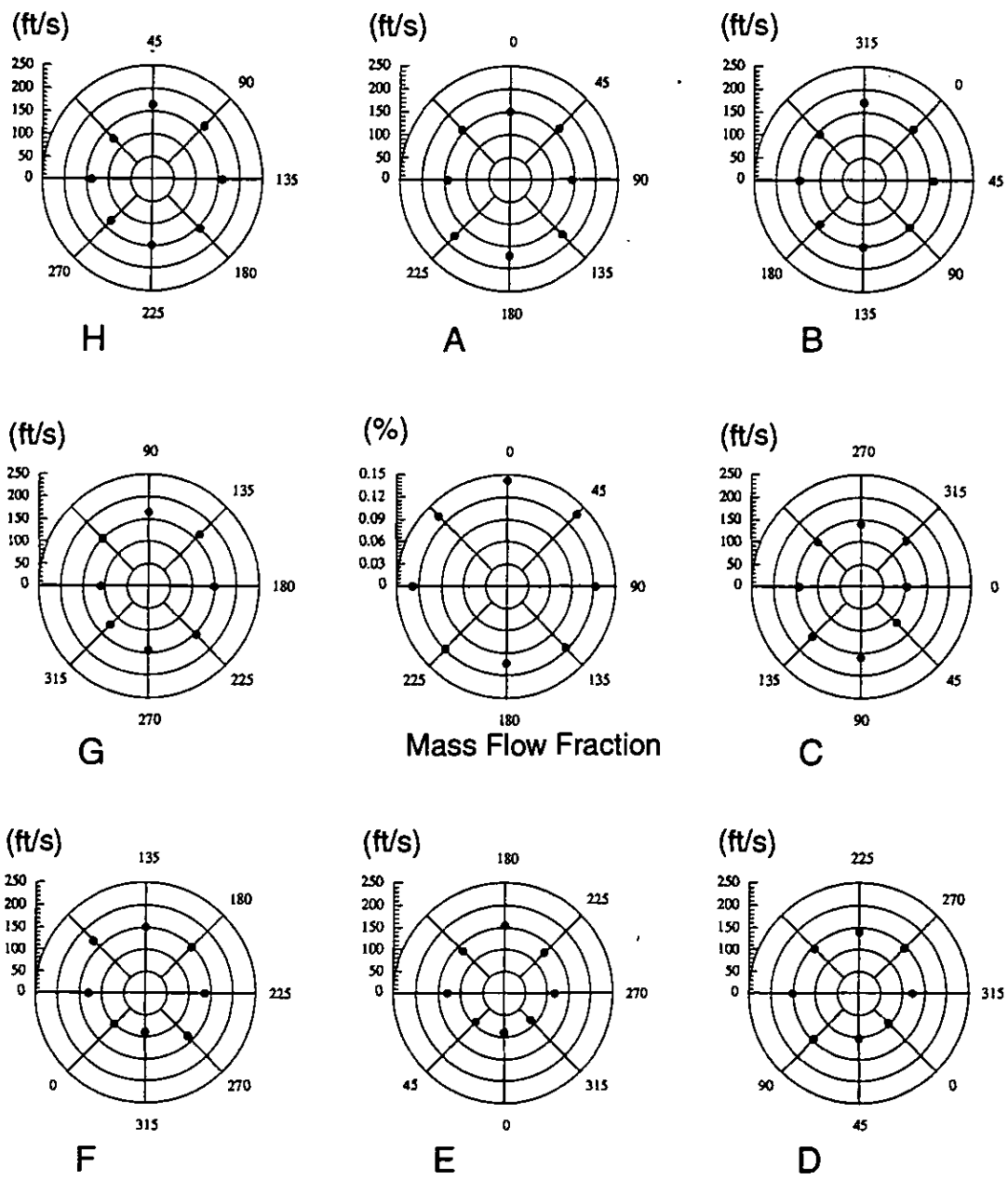


Fig. 10 Local and overall combustor inlet flow for case S212 (short hat, 20% extraction, test 1 at combustor 2)

affected the local flow distribution of each swirler (not shown) but had negligible effect on the overall combustor flow distribution (Fig. 11n). Both cases (S211 and S212) had approximately 15% more flow in the upper-left half circle (including swirlers G, H, A, and B) than the lower half of the circle (including swirlers C - F).

The effect of the distance from the air extraction port can be seen by comparing cases S232 (Fig. 11c) and S212 (Fig. 10). When the air extraction was changed in case S212 (Fig. 10) the distortion of local flow distribution that appeared in case S212 (Fig. 10) could still be seen in the outer portion (the quadrant 45°-0°-315°) of every swirler in case

S232 (not shown). The uniformity of the overall combustor mass distribution of case S232 (Fig. 11c) was, however, approximately within 8% between the upper-left and lower-right half circles; this was better than the 15% nonuniformity for case S212 (central diagram in Fig. 10). The improved flow uniformity in case S232 could be explained as follows: when the air was extracted diametrically across the dump diffuser, the flow moving toward outer part of the dump diffuser (away from the turbine axis) experienced an opposite force from the low pressure site near port X3; therefore, the flow on the upper half of the combustor reduced and overall flow distribution



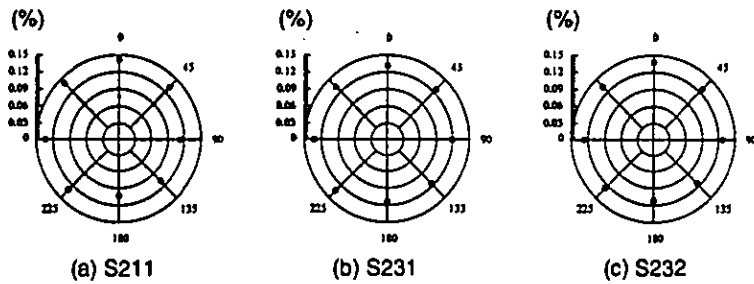


Fig. 11 Overall combustor inlet flow distribution for cases (a)S211 (b)S232 and (c) S231

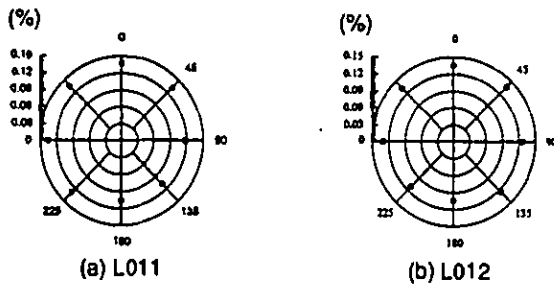


Fig. 12 Overall combustor inlet flow distribution for cases (a)L012 and (b)L011

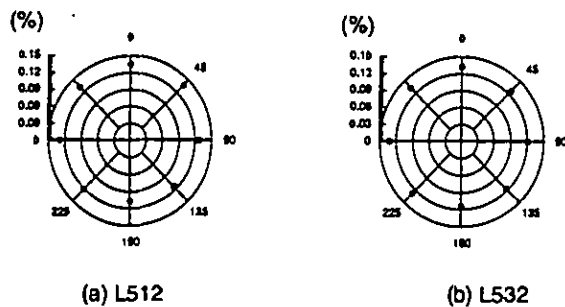


Fig. 13 Overall combustor inlet flow distribution for cases (a)L512 and (b)L532

improved. The mass flow for the lower half of the combustor was also affected by the extraction; however, since the rotor cooling pipe had already exerted resistance to the flow moving toward the lower portion of the combustor, the effect from the suction became less.

The effect of rotor cooling pipe with air extraction diametrically across the dump diffuser can be seen from the comparison between case S231 and S232 (Fig. 11b and 11c). With air being extracted at port X3, the flow distribution of combustor 1 of case S231, which was away from the rotor cooling pipe, showed better local flow uniformity at each swirler (not shown) than for case S232 (not shown). This was expected and was consistent with the results of baseline cases, which showed that combustor 1 had more uniform local flow distribution than

combustor 2. The overall distribution of case S231 became a little worse, however, with 10% more flow in the upper-left portion of the combustor 1 (Fig. 11b) compared to 8% of the combustor 2 in case S232 (Fig. 11c). A plausible explanation for this unexpected result could be that combustor 1 was away from the rotor cooling pipe, and the flow near the lower half of the combustor 1 was closer to port X3, so the suction force at X3 reduced flow more in the lower half of the circle. Nevertheless, a 2% difference was not that much, so this explanation should not be treated, however, as an absolute fact.

**Long top hat, no air extraction** —To improve the flow uniformity, longer top hats were installed based on the hypothesis that increasing the size of the end zone of the top hat may provide a longer time for better mixing. The results of overall combustor flow distribution for cases L012 and L011, using the long top hats without air extraction, are shown in Fig. 12. When they are compared with the corresponding short top hat cases of S012 (Fig. 7) and S011 (Fig. 8), there seems to be no significant changes even though the short top hats were replaced with the longer ones. Both L012 and L011 (Fig. 12) cases have approximately 15% more flow in upper-left half circle (from 225° to 45°) than in the lower-right half circle.

**Long top hat, 5% air extraction** —Long top hats somehow affect the local flow distributions for each swirler (not shown here) with 5% air extraction. As a consequence, the overall flow distributions for cases L512 and L532 (Fig. 13) also changed. In the short top hat cases of S531 (Fig. 8) and S512 (Fig. 9), 15% more flow enters the combustor in the upper-left half circle from 225° to 45° than in the lower half circle. The longer top hats shift the higher flow region to the upper half circle (from 270° to 90°). The flow uniformity is improved a bit, from 15% nonuniformity in case S512 (Fig. 9) to 10% in case L512 (Fig. 13), but maintains the same 15% nonuniformity between cases S531(Fig. 8) and L532 (Fig. 13).

**Long top hat, 20% air extraction** — The effect of long top hat on the local flow distribution at each swirler can be seen by comparing case S212 (Fig. 10) and case L212 (Fig. 14). No specific pattern is observed and an clear improvement of uniformity around each swirler is observed. By comparing the central diagrams in Fig. 10 and Fig. 14, the overall combustor flow distribution becomes less uniform in the long top hat case L212 than in the short top hat case S212. The S212 case has 15% more flow in the upper half circle (270°-0°-90°), whereas the L212 case has 25% more flow in the upper-right half circle (315°-0°-135°).

When air extraction was employed at port X3, the long top hats did not improve the flow uniformity, which can be seen by comparing case S232 and S231(Fig. 11) with cases L232 and L231 (Fig. 15). Both S231 and L231 have about 9% nonuniformity. However, the long top hat makes flow less uniform in case L232 that has 16% more flow in the upper half circle (270°-0°-90°) than case S232 that has 8% more flow in the upper-left half circle (225°-0°-45°).

From all the long top hat cases presented above, increased top hat size does not improve the flow uniformity, and may even make flow become less uniform. Therefore, the employment of long top hats is not recommended.

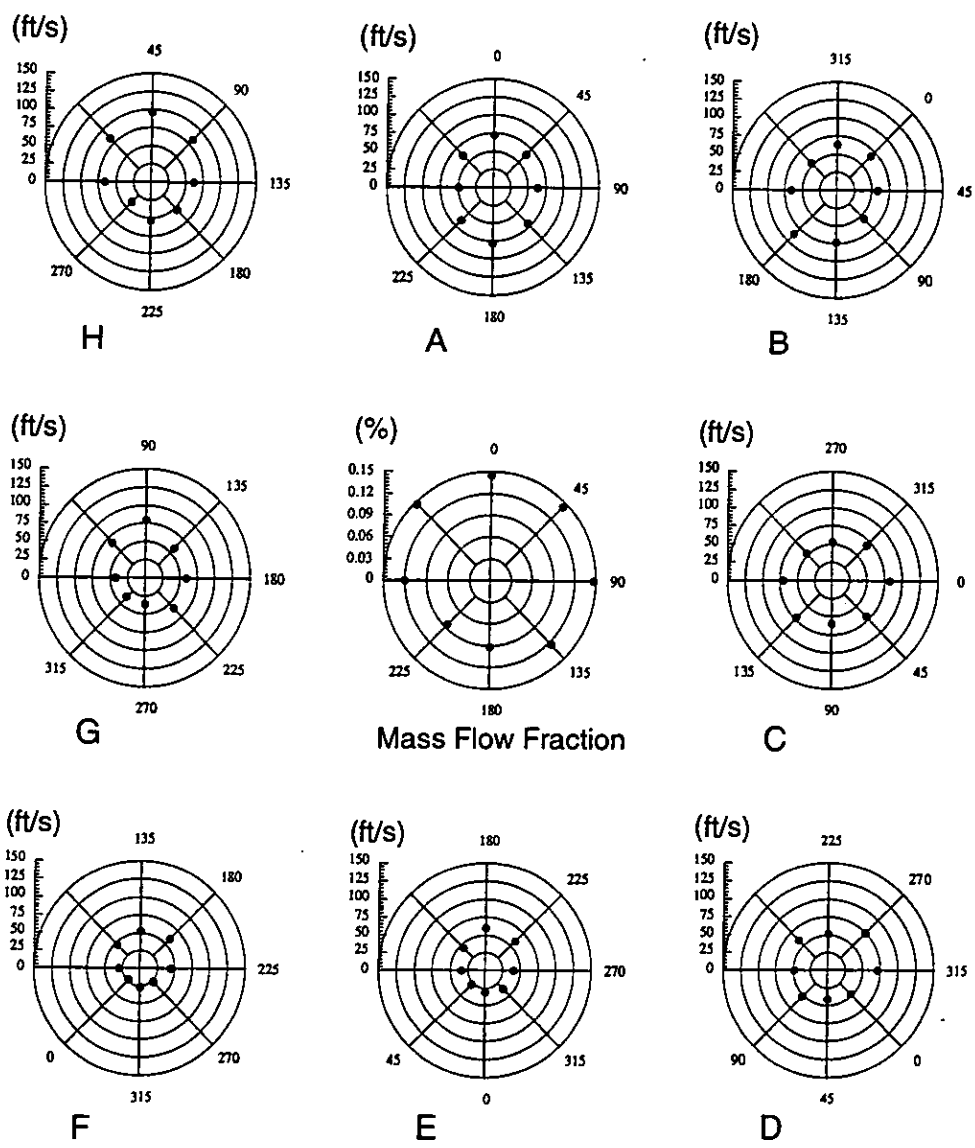


Fig. 14 Local and overall combustor inlet flow for case L212 (long hat, 20% extraction, test 1 at combustor 2

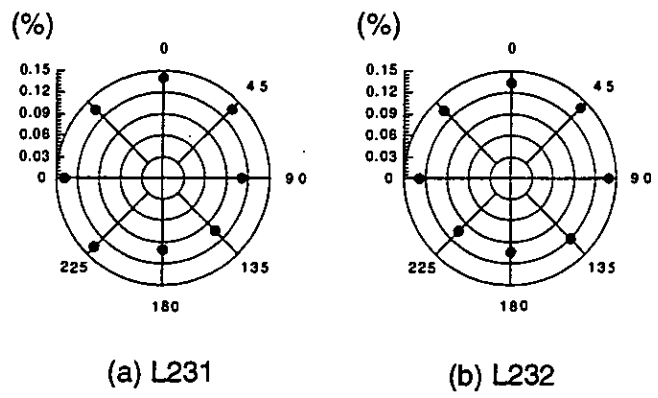


Fig. 15 Overall combustor inlet flow distribution for cases (a) L231 and (b) L232

## SUGGESTIONS FOR IMPROVEMENT

Since large top-hats did not improve airflow uniformity as was initially hypothesized, other possible means under consideration are listed below:

- (a) Uniform Resistance Method - provide circumferentially varying resistance in the top hat to counteract the effect of uniform flow resistance in the dump diffuser. Three possible ways are suggested: (i) vary the width of the flow passage circumferentially, (ii) add screen or honeycomb, (iii) add a long flow shield extending from the top hat radially into the dump diffuser.
- (b) Mixing Method - install turbulators near the combustor inlet to promote mixing.

## CONCLUSIONS

An experimental study was performed in a 48% scale, 360° model of the diffuser-combustor section of a developmental industrial gas turbine to investigate (a) the combustor inlet flow distribution in the reversed-flow type dump diffuser and (b) the effects of various air extractions for cooling (5%) and gasification (20%) on uniformity of the flow entering the combustors. The results indicated that even at the baseline case with no air extraction there was an inherent local flow nonuniformity of 10 - 20% at the inlet of each combustor due to the complex flow field in the dump diffuser and the blockage effect of the cross-flame tube. The combustors near the rotor cooling pipes have less uniformity than those away from the cooling pipes. More flow was seen in the portion further away from the gas turbine center axis. The effect of 5% air extraction was small. Twenty percent air extraction introduced approximately 35% global flow asymmetry diametrically across the dump diffuser. The effect of air extractions on the local flow uniformity of the combustors varied with the distances between the extraction port and each individual combustor. The most obvious difference is that flow is significantly retarded in the outer portion (the quadrant 45°-0°-315°) of every swirler. Although the 20% air extraction significantly distorts the local distribution of each swirler, the nonuniformity of overall mass flow rate distribution is about the same as that in the baseline case. This nonuniformity could be alleviated by extracting the air necessary for gasification at more than one location.

Longer top hats were installed with the initial intention to increase flow mixing. The results indicated that the longer top hats do not improve the flow uniformity. It was concluded that the degree of flow uniformity at the combustor's inlet was critical to the combustion uniformity and consequently emissions. It should be examined and improved for each gas turbine model. Although a specific geometry was selected in this study, the results provide sufficient generality to benefit other industrial gas turbines.

## ACKNOWLEDGMENT

The authors wish to thank Mr. Michael Thom for his help in constructing the experimental facility and the test model. This work was supported by the Morgantown Energy Technology Center of the U.S. Department of Energy through DOE/METC contract No. DE-AC21-93MC30247. The authors also wish to thank Dr. Richard A. Johnson at the Morgantown Energy Technology Center for his guidance in this program.

## REFERENCES

- Kapat, J.S., Agrawal, A.K., and Yang, T.T., 1994, "Air Extraction in a Gas Turbine for Integrated Gasification Combined Cycle (IGCC): Experiments and Analysis," ASME Paper 94-GT-193.
- Kapat, J.S., Wang, T., Ryan, W.R., Diakunchak, I.S., and Bannister, R.L., 1996a, "Cold Flow Experiments in a Sub-Scale Model of the Diffuser Combustor Section of an Industrial Gas Turbine," ASME paper 96-GT-518.
- Kapat, J.S., Wang, T., Ryan, W.R., Diakunchak, I.S., and Bannister, R.L., 1996b, "Experimental Studies of Air Extraction for Cooling and/or Gasification in Gas Turbine Applications," ASME paper 96-TA-10.
- Lefebvre, A.H., 1983, *Gas Turbine Combustion*, Hemisphere Pub. Corp., Washington, DC.
- Lyons, V., 1981, "Effect of Fuel-Air-Ratio Nonuniformity on Emissions of Nitrogen Oxides," NASA TP 1798.
- Welz, J.P., 1986, "An Algorithm for Using the Five-Hole Probe in the Non-Nullled Mode," AIAA/ASME 4th Fluid Mechanics, Plasma Dynamics and Lasers Conference, Atlanta, May 11-14.
- Wilson, D.G., 1984, *The Design of High-Efficiency Turbomachinery and Gas Turbines*, MIT Press, Cambridge, MA.
- Zhou, D., Wang, T., and Ryan, W.R., 1996, "Cold Flow Computations in a Sub-Scale Model of the Diffuser-Combustor Section of an Industrial Gas Turbine," presented at the ASME paper 96-GT-513