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

96-TA-10

EXPERIMENTAL STUDIES OF AIR EXTRACTION FOR COOLING AND/OR GASIFICATION IN GAS TURBINE APPLICATIONS

J.S. Kapat and T. Wang
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

This paper describes an experimental study on how the flow field inside the dump diffuser of an industrial gas turbine is affected by air extraction through a single port on the shell around the dump diffuser. A sub-scale, 360° model of the diffuser-combustor section of an advanced developmental industrial gas turbine was used in this study. The experiments were performed under cold flow conditions which can be scaled to actual machine operation. Three different conditions were experimentally studied: 0%, 5%, and 20% air extraction. It was found that air extraction, especially extraction at the 20% rate, introduced flow asymmetry inside the dump diffuser and, in some locations, increased the local flow recirculations. This indicated that when air was extracted through a single port on the shell, the performance of the dump diffuser was adversely affected with an approximate 7.6% increase of the total pressure loss, and the air flow into the combustors did not remain uniform. The global flow distribution was shown to be approximately 35% nonuniform diametrically across the dump diffuser. Although a specific geometry was selected, the results provide sufficient generality for improving understanding of the complex flow behavior in the reverse flow diffuser-combustor sections of gas turbines under the influence of various air extractions.

NOMENCLATURE

1, 2, 3	Longitudinal measurement planes (Figure 2)
B,...,G	Streamwise measurement locations (Figure 2)
D_h	Hydraulic diameter at the annular pre-diffuser inlet
IGCC	Integrated Gasification Combined Cycle
r	Radial distance measured from the centerline of the test section
U	Average axial velocity at the test section inlet
V	Velocity
x	Axial distance from the annular pre-diffuser inlet
X1, X2, X3	Extraction sites (Figure 3)

Subscripts

a	Axial component
c	Circumferential component (positive for clockwise motion when looking in the direction of the flow at the annular pre-diffuser inlet)
r	Radial component

INTRODUCTION

Integrated Gasification Combined Cycle (IGCC) systems are of specific interest to the governments and utility companies of many countries as well as to gasifier and gas turbine manufacturers. The United States government is especially interested in developing reliable and cost-competitive IGCC systems for generating power using its abundant coal reserves for the following reasons: (a) to reduce U.S. dependency on foreign energy sources, (b) to have a matured and cost-effective IGCC technology in place when the natural gas supply diminishes in the future, (c) to provide a clean coal technology, and (d) to help U.S. industries achieve and maintain leadership in advanced IGCC technologies, which will also play an important role in future power generation in countries with limited natural gas resources, like China and India. To emphasize the U.S. government's interest in using coal gas as a fuel for future gas turbines, the U.S. Department of Energy (DOE) specifically required that new gas turbines developed under the support of its Advanced Turbine Systems (ATS) program be adaptable to coal gas (DOE/Clemson Workshops I and II, 1991 and 1992, and DOE Congressional Report, 1993).

One of the most important technical challenges for an IGCC system is how to integrate the gasifier and the gas turbine systems to achieve cost-reduction and enhanced efficiency. The critical aspect of this integration is the removal of the compressor from the original gasifier system and supplying the gasifier with compressed air from the gas turbine. Following this integration process, the next technical question is how and where to extract the compressed air from the gas turbine. To investigate this issue, a collaborative research program between Clemson University and Westinghouse Electric Corporation, under the support of the Morgantown Energy Technology Center (METC) of DOE, was initiated at Clemson University.

In addition to extracting air for IGCC, air is extracted to cool the

Presented at the ASME Turbo Asia Conference
November 5–7, 1996 — Jakarta, Indonesia
This paper has been accepted for publication in the Transactions of the ASME
Discussion of it will be accepted at ASME Headquarters until February 28, 1997

rotor. Extracting air from the compressor to cool vanes and blades in the turbine has been a common practice. The locations of the air extraction ports are mostly dictated by two considerations: (a) the convenience and accessibility of the sites and (b) for matching the pressures at the sites of the cooling components.

In view of the gasification requirement for the highest pressure at which air can possibly be extracted, the section between the compressor exit and the turbine inlet, i.e. the diffuser-combustor section, has become the focus of the search for adequate sites for air extraction.

The diffuser-combustor section has two major functions: (a) to decelerate the high velocity air coming from the compressor (Wilson, 1984) and (b) to distribute air uniformly to the combustors (Lyons, 1981).

A typical heavy-frame industrial gas turbine uses a two-part diffuser-combustion section to decelerate the air exiting the compressor. Air from the compressor first 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. Different from the in-line annular combustor geometry in aircraft engines and small industrial gas turbines, heavy-frame industrial gas turbines employ reverse flow, can-annular combustors (Lefebvre, 1983) to reduce the overall system length and to make the combustion process relatively insensitive to inlet flow conditions. In these gas turbines, air from the annular pre-diffuser turns approximately 150° before entering the combustors and must maneuver around combustors and transition pieces as it travels to the inlet of the combustor. As a result, the flow characteristics in these diffusers are extremely complex and entirely different from those of in-line diffusers. Because of these complex flow fields, the irreversible losses in the diffuser-combustor sections generated from separation eddies in the dump diffuser and friction losses on the wall are significant. These losses manifest themselves as total pressure loss. Typically, one percentage point reduction in the total pressure loss (expressed as a fraction of the total pressure at the annular pre-diffuser inlet) translates to a 0.09 to 0.13 percentage point reduction in combined cycle efficiency (please note the distinction between percentage point and percentage). In addition, due to the nonaxisymmetric component arrangement in the dump diffuser, the flow entering the combustors is not ideal and may adversely affect the uniformity of combustion in the combustor. As a result, the emission levels may be compromised. Consequently, it is of vital importance to investigate the effects of air extraction on the flow pattern, the aerodynamic losses, and the uniformity of flow entering the combustors.

To the authors' knowledge, few experimental studies have been performed regarding air extraction for IGCC. Kapat, Agrawal, and Yang (1994) have reported the effect of air extraction on diffuser performance for another manufacturer's industrial gas turbine; however, only global performance has been studied. 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. These studies called our attention to the importance of improving our understanding of flow behavior under the influence of air extraction.

With the knowledge gained from the previous research, the current program was conducted through a series of studies with the overall objectives of (a) identifying potential locations for placing the air extraction ports, (b) assessing the advantages/disadvantages of various locations for extracting air, and (c) investigating the effects of air extraction on the aerodynamics, total pressure losses, and flow fields which affect the uniformity of combustion in the combustor.

First, a baseline study without air extraction was conducted computationally by Zhou, Wang, and Ryan (1996). The computational results were then used to guide the design and instrumentation of an experimental study of the baseline case without air extraction by Kapat, Wang, et al. (1996). For the air extraction study, the potential locations for the air extraction ports were identified at various locations on the walls of the pre-diffuser and the dump diffuser, taking into consideration the issues of accessibility and structural soundness. Objectives (a) and (b) were achieved through a numerical study (to be published at a later date) to compare the diffuser effectiveness under the conditions of applying air extraction at the inlet, in the middle, and at the exit of the annular pre-diffuser, respectively.

To achieve objective (c), a very complex experimental study for investigating the effects of air extraction on the detailed flow field inside the dump diffuser was conducted. This paper focuses on presenting the experimental results in the dump diffuser.

Two different air extraction rates, 20% and 5% of the flow at the dump diffuser outer wall (sometimes referred to as the combustor shell), were considered. The larger extraction simulates the conditions when air was extracted for gasification, and the smaller extraction simulates the conditions when air was extracted for cooling.

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 under the influence of air extraction.

EXPERIMENTAL FACILITY

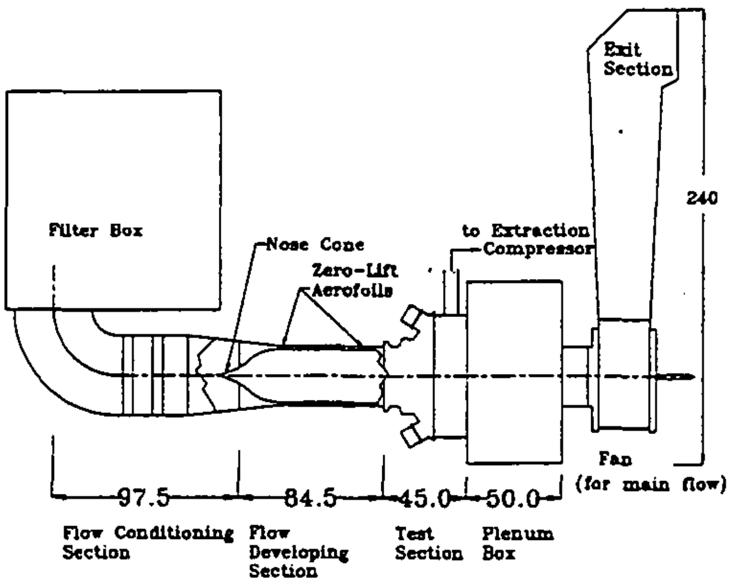
The desired air flow through the test model was provided by an open-circuit, suction-type wind tunnel. The overall layout is shown in Figure 1. Air from the test section went into a plenum box which was maintained at -40" H_2O gage pressure by a suction fan rated at 33,000 cfm (15.6 m^3/s) at 1.4 psi (9.65 kPa) 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. (1996).

TEST SECTION

Figure 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 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°, maneuvered around the transition pieces and combustors, and



(All dimensions are in inches)

Figure 1. Layout of test facility

entered the annular passage between the combustor and the top hat (Figure 2). The annular passage in the top hat helped to distribute the flow before it turned approximately 180° and entered the combustor. After passing through the combustor and the transition piece, the flow was exhausted into the plenum.

The dump diffuser in this test model contained all of 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, the 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. Detailed information on the test section and calibration of the combustor pressure loss coefficient is described in Kapat et al. (1996).

AIR EXTRACTION

In the test section, all profile measurements were made around combustor 2 (Figure 3). In order to study the circumferential nonuniformity due to air extraction, three different extraction ports (X1, X2, and X3), situated 90° apart circumferentially, were built into the test section (Figure 3). Only one of these ports was used at a time for extraction. Thus by extracting air at different circumferential locations but taking measurements at the same locations, the study of circumferential variation of the effects of extraction was achieved. In this way, the number of instrumented sites was minimized

The extraction duct was equipped with a pitot-static probe which was used to monitor the air flow rate through the extraction duct (Figure 4). The desired flow rate through the extraction duct was achieved by using both the extraction and the bypass ducts. Small adjustments in the flow rate were performed by using the extraction valve. However, for large variations, the bypass valve was used so as to maintain the total flow rate through the extraction compressor within the operating range. The extraction compressor comprised six stages and was rated at 5,000 cfm (2.36 m^3/s) at 5 psia (0.34 bars).

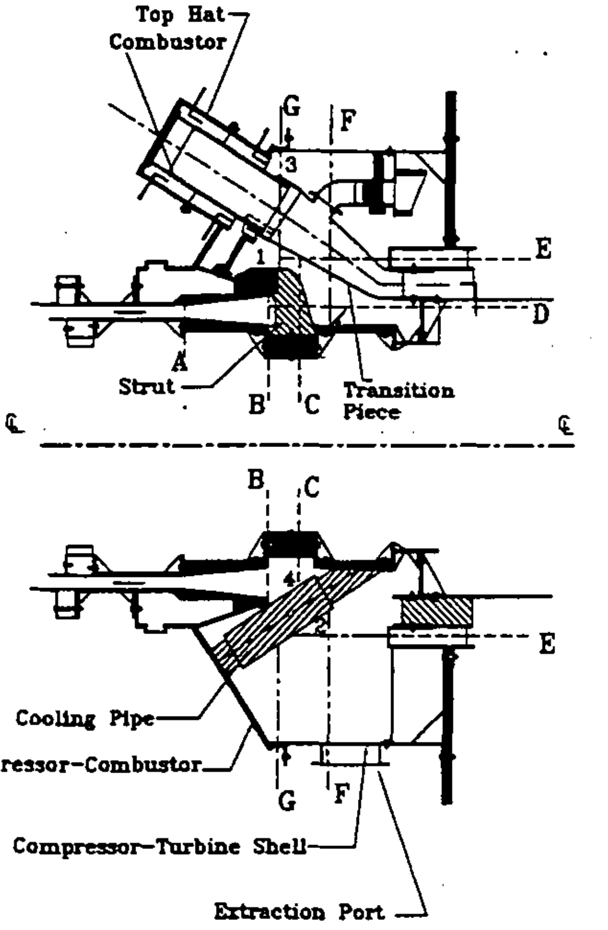
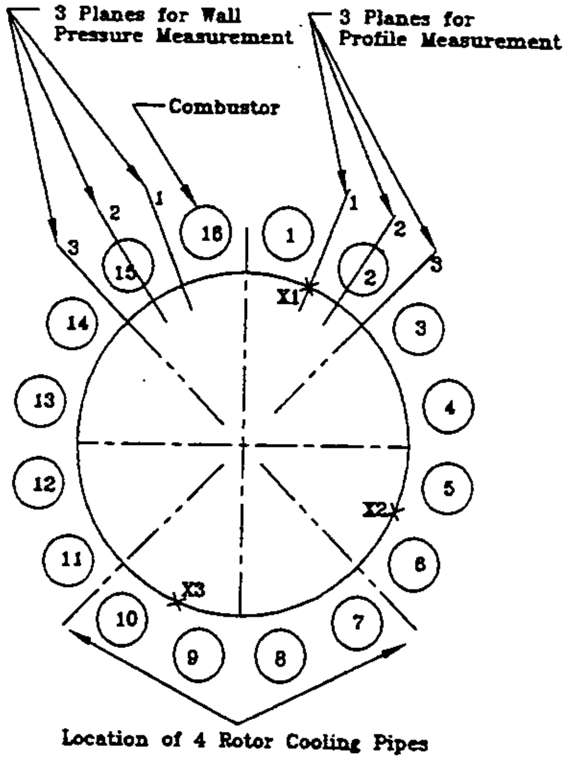


Figure 2. Sub-scale test model and measurement locations

For different combinations of 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.

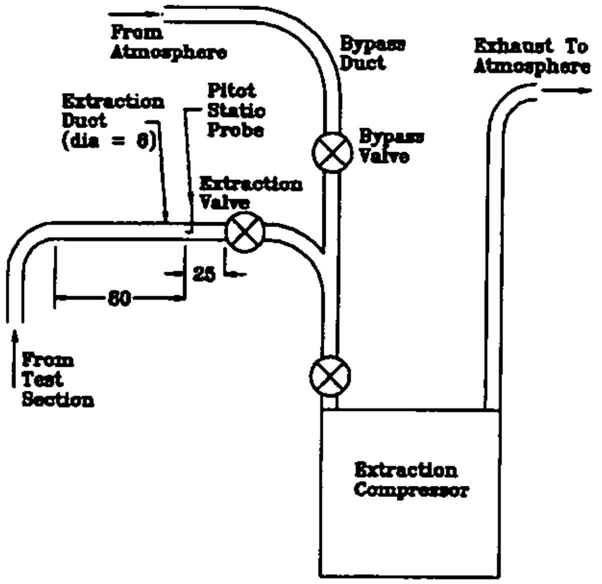
During the main experiments with 5% or 20% extraction, one of the three extraction ports, X1, X2, and X3, was connected to the extraction duct. First, the main flow through the test section was adjusted with the extraction valve closed. Then, the extraction flow rate was obtained by adjusting the extraction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

After the velocity profiles were measured by traversing the five-hole probe at different locations (as described in the following section), the whole process was repeated for each of the other two extraction ports.



NOTE:
1. In this view, flow is into the paper at the Pre-Diffuser Inlet.
2. Plane 1 is the mid-plane between combustors 1 and 2 or between combustors 15 and 16.
3. Plane 2 is the mid-plane of combustor 2 or 15.
4. Plane 3 is the mid-plane of a rotor cooling pipe and in between two combustors.
5. X1, X2 and X3 are three extraction sites; only one is used at a time.

Figure 3. Circumferential layout of measurements



(All dimensions are in inches)

Figure 4. Air extraction ducting and flow control

INSTRUMENTATION AND MEASUREMENTS

In the developing section, a pitot-static probe was placed 6 D_h upstream of the inlet to the annular pre-diffuser at four circumferential locations, 90° apart. By moving the probe radially at each of these four locations, the radial profiles of the axial velocity and the total pressure were obtained.

An angle-type five-hole probe was traversed at fifteen different locations or ports to measure three-dimensional velocity profiles at those locations. These fifteen measurement ports were located on three longitudinal planes; the circumferential locations of these planes, numbered 1 through 3, are shown in Figure 3. The ports were located at six different streamwise locations along the flow path. The streamwise locations, named B through G, are shown in Figure 2. Out of the possible total of eighteen ports, B1 through G3, no measurement was made at C2, D3, or E2 because of geometric obstructions. In total, 49 traverses were performed to measure the 3-D velocity profiles in the dump diffuser.

The calibration indicated that the five-hole probe used has an acceptance angle of ±30° in the pitch plane and an acceptance angle of ±35° in the yaw plane. Outside these acceptance angles, flow separation occurred for the air velocities used, and the five hole probe became ineffective. If the pressure coefficients calculated from the measured data were outside the range of the pressure coefficients obtained during the calibration process, the corresponding measurement point was discarded. This could happen for one of two reasons: (a) either the air was stagnant at the point of measurement or (b) the local velocity vector was outside the acceptance angle of the probe. After the experiments were completed, it was found that the 3-D flow was very complex, so only the results from the B, C, and E ports have been used as representatives for discussion in this paper.

The pressure readings of the five-hole probe were measured by a scanning module from Scanivalve. The transducer used in the experiments has a range of ±2.5, with a calibrated precision of ±0.007 psi (at 95% confidence). The detailed calibration process is described in Kapat et al. (1996).

The overall mass flow rate entering each combustor was obtained by integrating the velocity distributions measured by a commercial tungsten hot wire 2.5 μm in diameter.

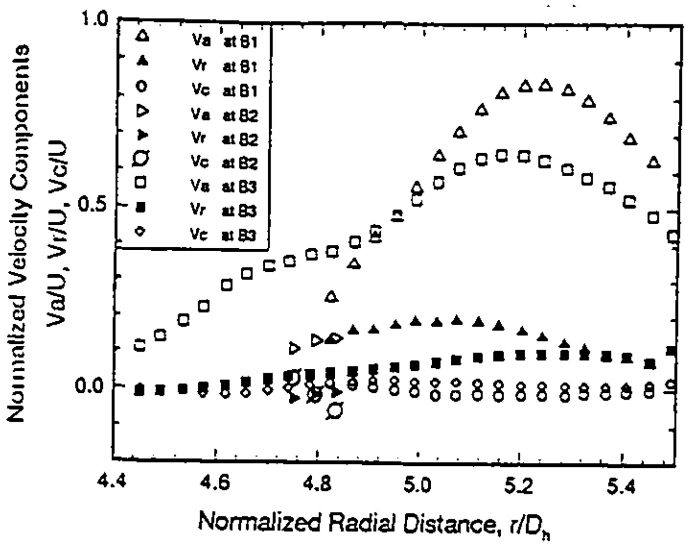
RESULTS AND DISCUSSION

The results are presented in nondimensional form. Radial distances were measured from the centerline unless otherwise specified and were normalized by the hydraulic diameter (D_h) of the annular inlet to the test section. All axial distances were measured from the annular pre-diffuser inlet (locatinn A in Figure 2) and were normalized by D_h. All velocities were normalized with U, the average (axial) velocity at the annular pre-diffuser inlet. All of the experiments were performed for a fixed flow rate at the inlet to the pre-diffuser. This flow rate was monitored by the pitot-static probe in the development section.

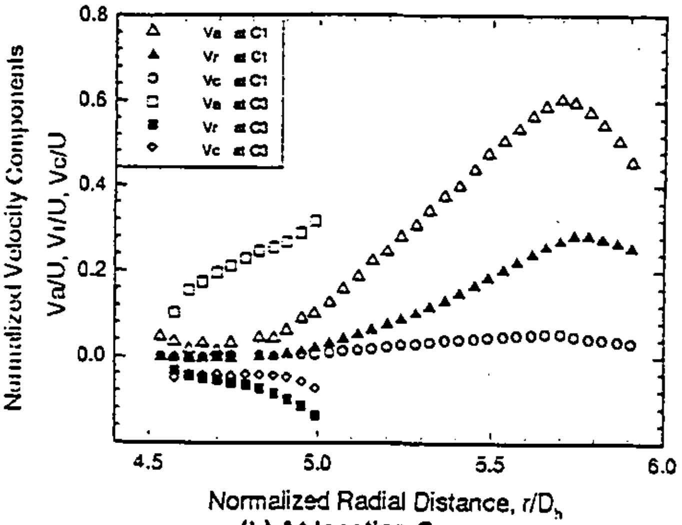
The flow field for the baseline (no extraction) has been discussed in detail by Kapat et al. (1996), and hence the corresponding velocity profiles are presented only for reference and for comparison with the velocity profiles for the extraction cases. The velocity results at three locations (B, C, and E, see Figure 2) are grouped for each port (X1, X2, or X3, see Figure 3) for a specific extraction rate at 5% or 20%. The figure for each location includes three planes: Plane 1 cuts through the midplane between two can-type combustors, Plane 2 cuts through the centerplane of the combustor, and Plane 3 cuts through the midplane between two combustors and the centerplane of a cooling pipe (Figure 3). There were sixteen combustors and only four cooling pipes, so while Plane 3 cuts through a cooling pipe, Plane 1 does not. The results for the X1 port indicate the maximum effects of nearby extraction. The X2 port is located 90° away from the measurement planes, and the X3 port is located diametrically from the X1 port as well as from Plane 1. Hence, the results of the X3 port extraction should indicate the effects from the air extraction directly opposite from the other side of the dump diffuser. The following discussion of the results is conducted by fixing a specific location, say the B location first, and then comparing the results from port to port and between two extraction rates.

Figure 5 shows the results of the velocity profiles at locations B, C, and E for the baseline case, i.e. without air extraction. For 5% air extraction, only the results for extraction at the X1 port are shown (see Figure 6). Comparison of the velocity profiles between Figures 5 and 6 at B1 (location B and Plane 1), which is at the pre-diffuser exit, shows that there was no qualitative change in the shape of the axial velocity profiles due to 5% extraction at X1. Plane 1 is located between the combustors with no flow obstructions between the combustors. Increasing the extraction to 20% did not change the trend of the axial velocity profiles very much at B1 (Va in Figures 7a, 8a, and 9a), except that the profile became broader with a slightly reduced axial velocity. A greater reduction of the axial velocity occurred for extraction at X2 and X3 (Va in Figures 8a and 9a) than at X1 (Va in Figure 7a). This is because both X2 and X3 are located at different quadrants; X2 is 90° away and X3 is diametrically across the dump diffuser, respectively. Extraction at X3 increased the radial momentum on Plane B1 toward the other side of the dump diffuser where X3 is located (negative Vr) and made the flow exiting the pre-diffuser bend less radially toward the outer dump diffuser of the same side (positive Vr). Extraction at X2 would have increased the circumferential velocity (Vc) as well as the negative radial velocity. Indeed, the results of the radial and circumferential velocity components (Vr and Vc) at B1 in Figures 7a, 8a, and 9a support these facts. As can be seen in these figures, the radial component for 20% extraction actually became negative for a small section of the profile at B1 for all ports (Vr in Figures 7a, 8a, and 9a), and more regions of negative values occurred at port X3. The increase in the circumferential velocity components (Vc) can be also seen in these figures. The increased values of both Vr and Vc are thought to contribute to the reduction of the axial velocity components (Va).

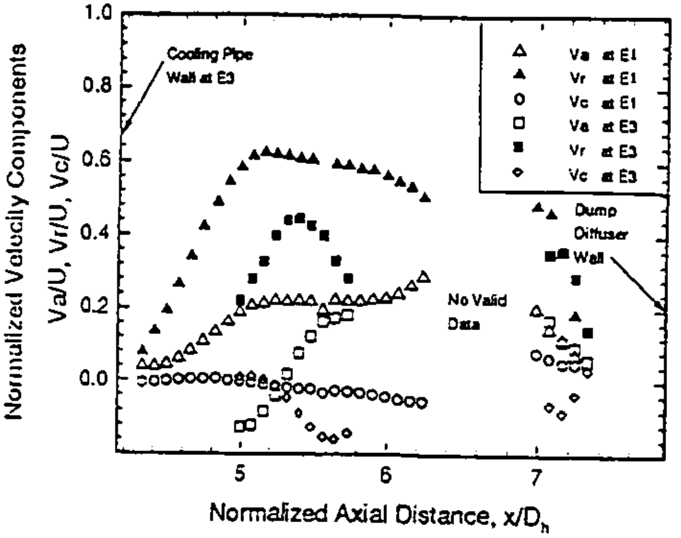
Plane 3 is similar to Plane 1 in that both cut through the midplane between two combustors; however, a rotor cooling air supply pipe is located on Plane 3. This rotor cooling air pipe forced the flow to become more uniform at the annular pre-diffuser exit (see Va at B3 in Figure 5a) and reduced the radial momentum (see Vr at B3 in Figure 5a). Extractions did not strongly affect the velocity profiles at B3 at any of the three ports, except that the circumferential component increases in magnitude (Vc at B3 in Figures 4a - 9a) and some of the radial velocities become negative at X3 (Vr at B3 in Figure 9a). This happens because extraction at any of these ports disturbs the



(a) At location B (exit of the pre-diffuser, see Figure 2)



(b) At location C



(c) At location E

Figure 5. Velocity profiles without air extraction (baseline)

circumferential uniformity of the flow field around B3.

Just downstream of Plane B2, the flow was obstructed by a support strut (Figure 2). As a result, there was not much flow through this section for the baseline case (B2 in Figure 5), and measurements were difficult. Five percent extraction did not strongly affect the velocity profile; however, 20% extraction significantly changed the flow pattern in all aspects on Plane 2. The axial velocity at B2, which was low and had only a few points within the measuring range of the five-hole probe, increased considerably in the presence of extraction at all extraction ports (Va at B2 in Figures 7a, 8a, and 9a). The circumferential and radial velocity components, small as they were, were measurable by the five-hole probe. As on Plane B1, negative Vr appeared at B2 (Figure 8a and 9a) as the result of extraction. These increases in Va and Vc were probably related to the increase in the circumferential component at B1 or B3.

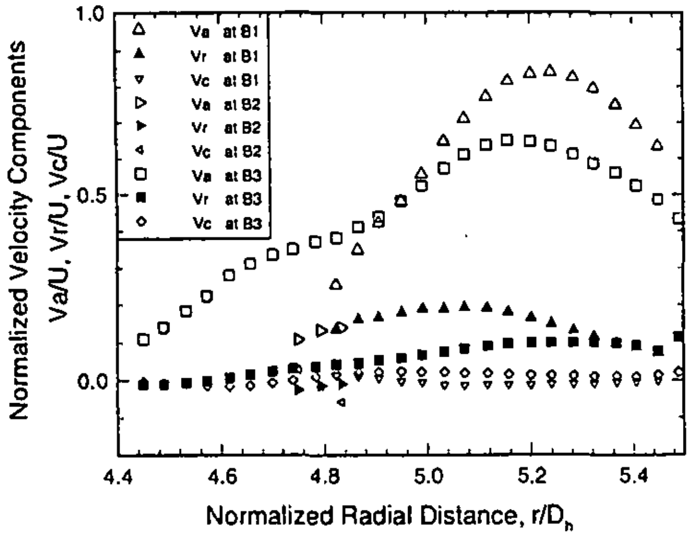
Location C is situated downstream of location B. Plane 2 at location C (or C2) is blocked by the support strut, so no flow was available for C2. Only C1 and C2 are shown in Figure 5b for the baseline cases. The nonsymmetric, nonuniform velocity profile at B1 continued at C1 (see Va at C1 in Figure 5b) as the air flowed through the space between two combustors towards the outer dump diffuser. Because of the cooling pipe, the air at C3 had a radially inward motion (with negative Vr values at C1 in Figure 5b), possibly caused by impingement on the cooling pipe. This radially inward motion suggests the existence of a recirculation zone in the region between the cooling pipe and the inner surface of the dump diffuser (region 4 in Figure 2). It is hypothesized that air from the annular pre-diffuser which flowed toward the rotor cooling pipe was deflected to either side of the pipe, flowed around the pipe, ultimately joined together on the other side of the pipe (region 2 in Figure 2), and then flowed outward to the outer dump diffuser. This presumed flow pattern would have created wakes and recirculation eddies on the downstream side (region 2 in Figure 2) of the cooling pipes. Further testing is required to verify this presumption; however, its existence is supported by the Vr profile at E3 in Figure 5c, as to be discussed later. Extraction seems to have a negligible effect on the above mentioned flow pattern for the baseline cases except that the circumferential component at C1 had a noticeable increase with 20% extraction at X3 (see Vc at C1 in Figure 9b). Otherwise, extraction at this port or at the other two ports caused only a minor increase in the axial velocity near the outer wall (see Va at C1 in part b of Figures 5 - 9). At plane C3, extraction seems to have reduced the negative circumferential velocity for all ports (see Vc at C3 in part b of Figures 5-9).

Of all the profiles studied here, the profiles at the E locations are most affected by extraction. The velocity profiles at the E planes indicate the major flow activity in the dump diffuser, the flow field mostly consisted of radial flow moving from the inner dump diffuser to the outer dump diffuser. Due to the complex three-dimensional flow in various planes at location E and the strong effects of extraction on the flow pattern, it is very difficult to piece together the limited information available from the data. For the baseline cases, the air at plane E1 moved outward and forward without any obstruction with dominant radial velocities, as indicated by the Va profiles in Figure 5c. With 20% extraction, the qualitative nature of the profiles of all three components of velocity at E1 did not seem to be affected by extraction at X1 or X2, although the flow did increase with more uniform distribution of the radial components (Vr), as can be seen at E1 in Figures 7(c) and 8(c).

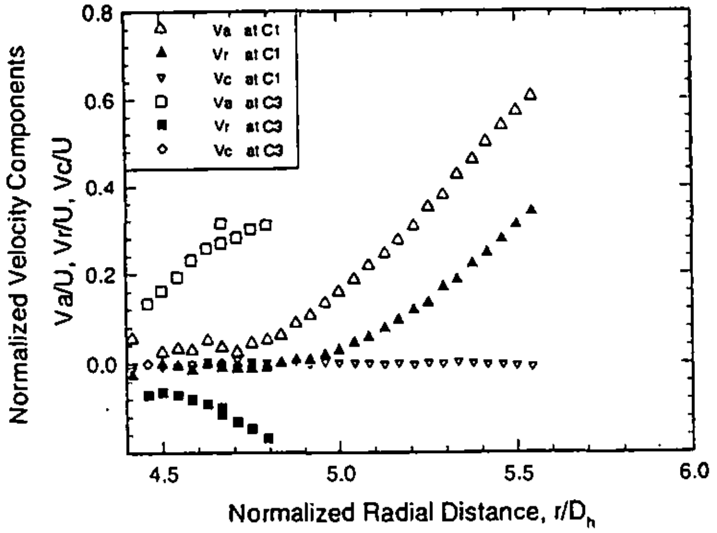
For the baseline cases, the axial velocity profile at E3 in Figure 5c indicates that Va became negative at around x/D_h= 5.3 and that the air started reversing its axial motion to flow towards the combustor inlets. The radial velocity at E3 flowed radially outward only over a limited region. Between this region and the cooling pipe, no valid data was available, as marked in Figure 5c. One possible reason for this lack of valid data was the existence of a recirculation zone(s) next to the cooling pipe. The recirculation flow was obviously outside the effective measuring range of the five-hole probe. The existence of this recirculation zone also supports the previous speculation that circulation eddies may have occurred behind the cooling pipes. It should be noted that the center of the extraction port is at x/D_h = 5.2.

Extraction significantly changed the flow pattern at E3. The region of flow reversal, indicated by the change in sign of Va at E3 in Figure 5, became all negative with 20% suction at X1 and X3 (Figure 7c and 9c). It is not clear what the actual mechanism was that caused the negative Va. One possible explanation is that the suction on the wall at X/D_h = 5.2 generated a low pressure center at the suction ports, which reduced the region of flow moving forward (positive Va) and outward (positive Vr); hence the portion of reversed flow which moved toward the combustor inlet expanded, and Va became all negative. This trend did not occur with suction at X2, so Va at E3 in Figure 8c still shows evidence of changing sign. The circumferential velocity components (Vc) became all positive with extraction at X1 and X3 (Figures 7 and 9), which indicates flow toward the suction port. Extraction at X3 caused complex flow patterns around E1 or E3, as can be seen by the corresponding profiles in Figure 9(c). It is interesting to see that Vr at E1 was measurable in the region with X/D_h < 4.0 (Figure 9c), where no data was available for the baseline and all other cases.

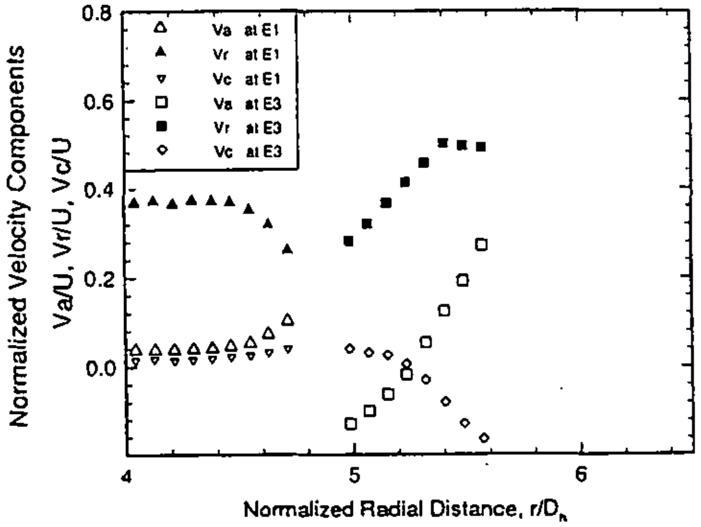
The total pressure losses at 20% suction are compared to the baseline case in Table 1. Near extraction port 3, the effect of suction on the total pressure losses is negligible, whereas the total pressure losses increased 7.6% near port 1. The velocity distributions at every 45° interval surrounding the inlet of the combustor for various conditions were measured by a hot wire sensor and are shown in Figure 10. Generally, this figure shows that the flow rate near the region between 130° and 180° (the inner side of the ombustor in Figure 2) is about 20% less than the flow rate at the outer part of the combustor. This is consistent with the CFD prediction conducted by Zhou et. al. (1996). The differences between the two baseline cases for combustor 1 indicate a maximum measurement uncertainty of approximately ±5%. The flow rate in combustor 2, which was located closer to the rotor cooling pipe (see Figure 3), was greater than the flow rate in combustor 1. Figure 10 also shows that air extraction did not cause the flow distribution of the baseline case near port 1 to vary, although the flow rate did decrease. Near port 3, 20% air extraction significantly reduced the flow rate about 20% on top of the combustor (near 0°) and about 50% on the bottom of the combustor (near 180°). The difference between port 1 and port 3 implies that the global flow distribution was approximately 35% nonuniform diametrically across from the dump diffuser.



(a) At location B (exit of the pre-diffuser, see Figure 2)

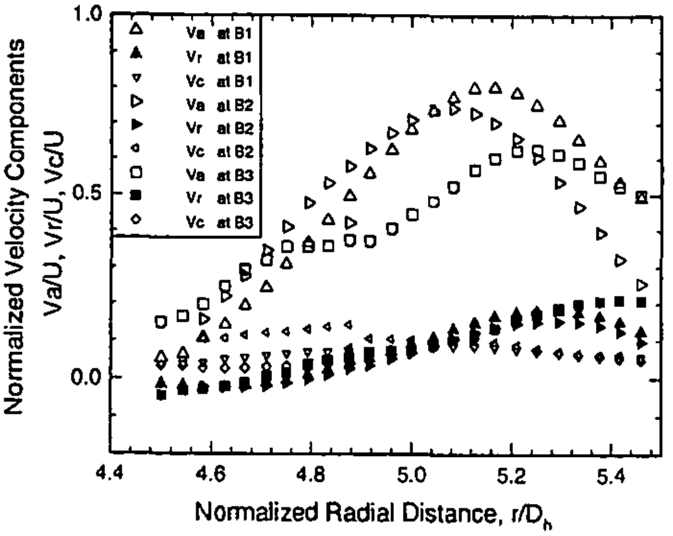


(b) At location C

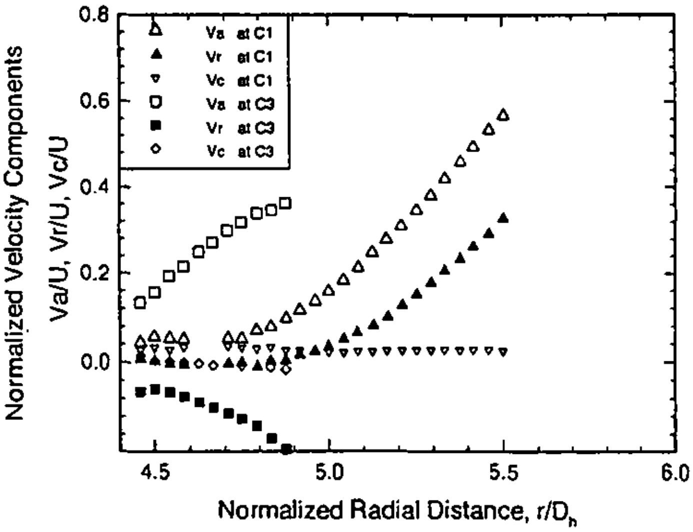


(c) At location E

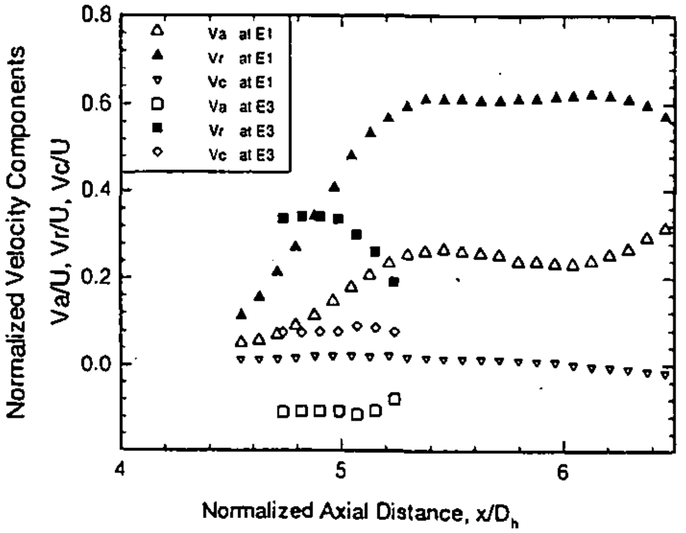
Figure 6. Velocity profiles with 5% air extraction at X1



(a) At location B (exit of the pre-diffuser, see Figure 2)

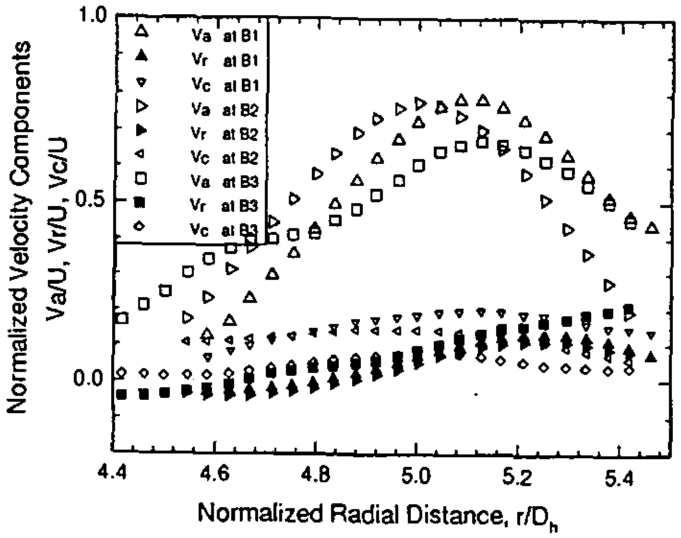


(b) At location C

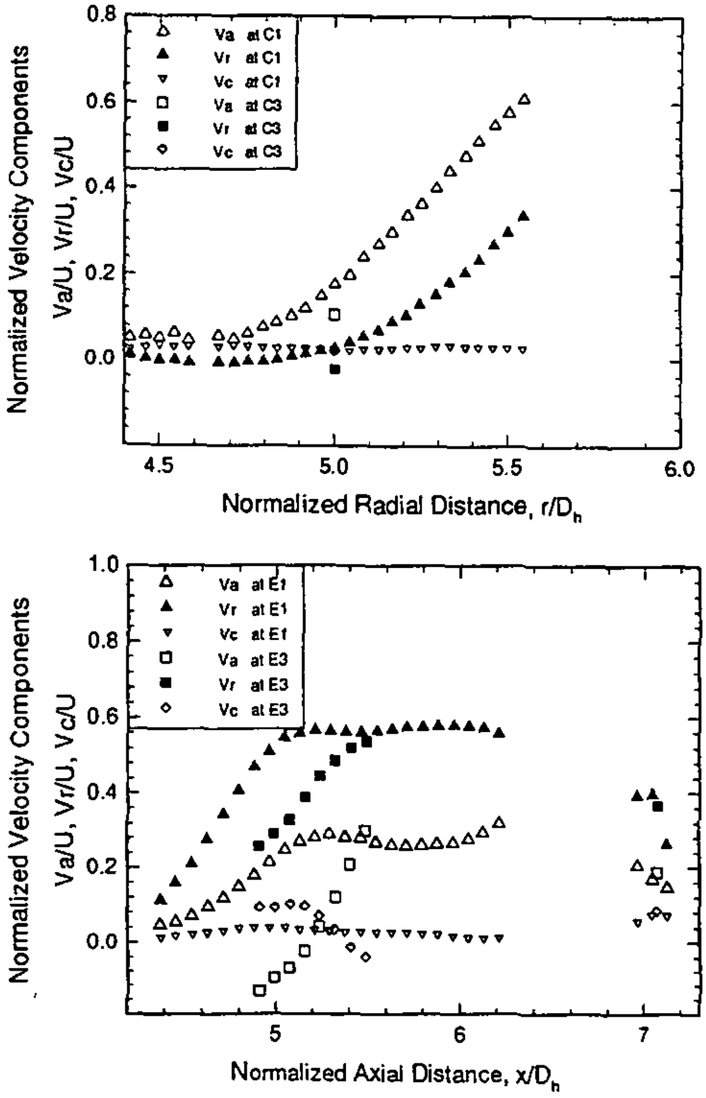


(c) At location E

Figure 7. Velocity profiles with 20% air extraction at X1

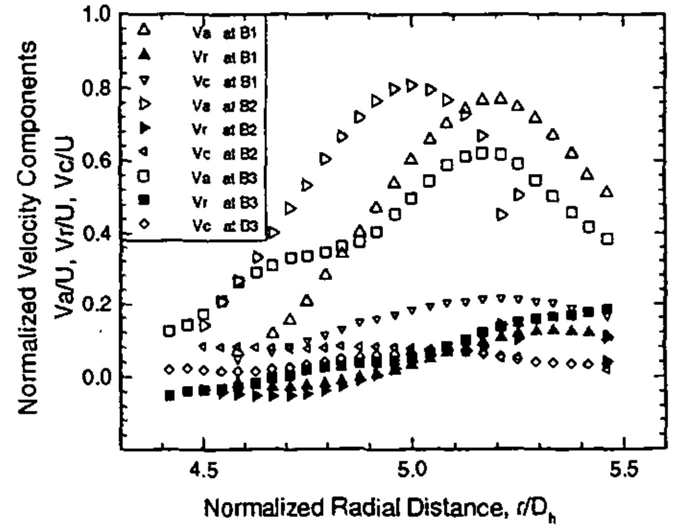


(a) At location B (exit of the pre-diffuser, see Figure 2)

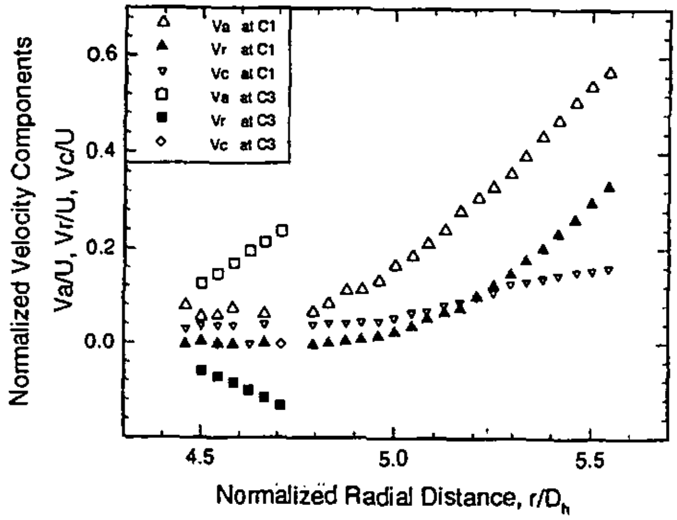


(c) At location E

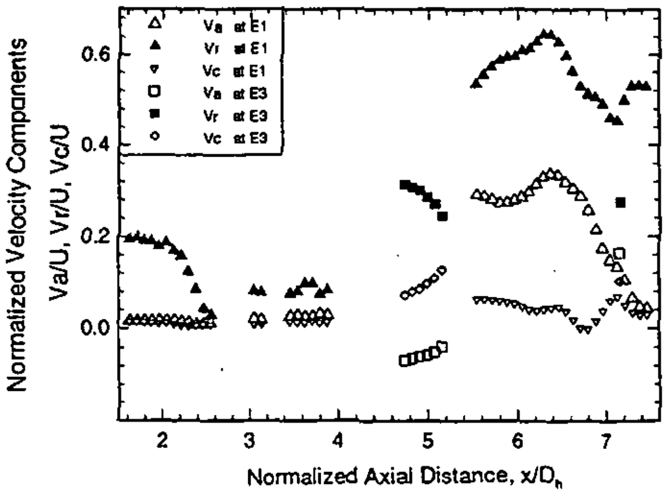
Figure 8 Velocity profiles with 20% air extraction at X2



(a) At location B (exit of the pre-diffuser, see Figure 2)



(b) At location C



(c) At location E

Figure 9 Velocity Profiles with 20% air extraction at X3

Table 1. Total pressure losses (normalized by the velocity head at the the inlet of pre-diffuser) at 20% suction from location A to the top hat inlet (or dump diffuser outlet)

20% Suction	Average of 3 Planes	Difference %
Baseline	0.58	
Port 1	0.624	7.6 %
Port 3	0.581	0 %

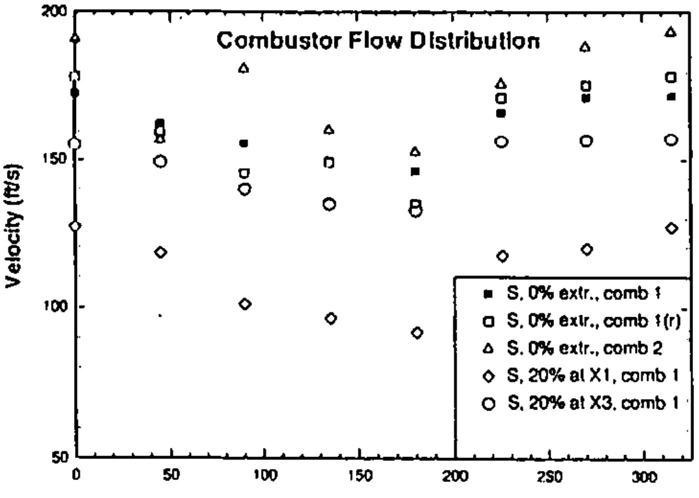


Figure 10. Combustion flow distributions for baseline and extraction cases

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 the effects of various air extractions for cooling (5%) and gasification (20%) on flow fields in the dump diffuser. Detailed wall pressures and three-dimensional flow fields were measured. The results for 5% air extraction did not show a significant effect on the flow fields in the dump diffuser. However, the results for 20% air extraction from a single port on the shell indicated that strong circumferential motion, up to 10% of the mean flow velocity, was induced and significant distortion of the flow fields was produced in various parts of the dump diffuser. The distortion of the flow field adversely affected the uniformity of the air flow to the combustor, and hence the combustor performance. The global flow distribution was approximately 35% nonuniform diametrically across from the dump diffuser. The circumferential motion through the dump diffuser resulted in an approximate 7.6% increase of the total pressure losses near the suction port in the dump diffuser. This test demonstrated the adverse effects of single-port air extraction on total pressure losses and on the uniformity of the flow distribution to each combustor. The results of this experiment strongly suggest that single-port air extraction should be avoided and that axisymmetric multi-port extraction should be considered.

ACKNOWLEDGMENT

The authors wish to thank Mr. Michael Thom for his help in fabricating 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

DOE/Clemson Workshop, 1991, Proceedings, Workshop to Define Gas Turbine System Research Needs, Part I, April 8-10.

DOE/Clemson Workshop, 1992, Proceedings, Workshop to Define Gas Turbine System Research Needs, Part II, Jan. 7-8.

DOE, 1993, Comprehensive Program Plan for Advanced Turbine Systems, Congressional Report, DOE/FE-0279.

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., 1996, "Cold Flow Experiments in a Sub-Scale Model of the Diffuser Combustor Section of an Industrial Gas Turbine," ASME paper 96-G 7-518.

Lefebvre, A.H., 1983, Gas Turbine Combustion, Hemisphere Pub. Corp., Washington, DC.

Lyons, 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-Nulled Mode," AIAA/ASME 4th Fluid Mechanics, Plasma Dynamics and Lasers Conference, Atlanta, Georgia, 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 Turbo Expo '96, Birmingham, UK.