Beni Cukurel Patrick B. Lawless Sanford Fleeter

Purdue University, West Lafayette, IN 47907-2040

Particle Image Velocity Investigation of a High Speed Centrifugal Compressor Diffuser: Spanwise and Loading Variations

An efficient diffuser is essential to a modern compressor stage due to its significance in stage performance, durability, and operability. To address the need for data that describe the complex, unsteady flow field in a vaned diffuser, particle image velocity is utilized to characterize the spanwise and circumferential variations in the flow features in the vaned diffuser passage of a transonic centrifugal compressor. The spanwise variation in the diffuser flow field is further investigated by the comparison of three different operating conditions representative of low, nominal, and high loading. These data demonstrate that not only the diffuser flow field is highly dependent on the operation conditions, e.g., hub-to-shroud variation increases with loading, but also the circumferential periodicity, created by the highly three dimensional impeller discharge flow, generates a larger unsteadiness toward the hub region of the vaned diffuser. [DOI: 10.1115/1.3104616]

1 Introduction

Future small gas turbines rely on higher efficiency and pressure ratio centrifugal compressors to achieve lower specific fuel consumption, higher specific power, and higher power to weight ratio. An efficient diffuser is essential to a modern compressor stage due to its significance in stage performance, durability, and operability.

Even though investigations of low speed compressor flow fields have been used to develop a knowledge base for design and generating flow models, they are not true representations of transonic centrifugal compressors. Hathaway et al. [1] conducted an investigation for identifying the feasibility of using a low speed compressor to capture the flow physics and as a starting point for experimentally validating codes that would later be used for designing high speed machines. It was concluded that the secondary flows, which are significant in centrifugal compressors, are strongly affected by the impeller speed, and for high speed applications, empirical data from nonscaled models are crucial.

Also, often data on the exit flows of impellers with vaneless diffusers have been used for design models that included vaned diffusers, i.e., making use of an isolated impeller approximation. Though this approach might be a starting point for advanced designs, it is very limited in its accuracy [2]. For example, Inoue and Cumpsty [3] reported that the presence of diffuser vanes considerably increases the pressure at the exit of the impeller, thus coupling the vane inlet and impeller exit flow fields [4,5]. Consequently, matching a vaned diffuser to an impeller is a nontrivial task due to the complicated flow mechanics involved [6].

Another approximation in the design process is that most compressors are designed for steady relative flows, but the actual flow is unsteady with a high degree of interaction between the impeller and diffuser. The effects of the diffuser geometry on the compressor stage are difficult to predict due to the existence of this coupling between the impeller and diffuser. The potential field generated by the diffuser and imposed on the impeller exit is not only driven by its geometry but also dependent on the unsteady diffuser loading. This loading is in turn a function of the rotating impeller potential field and the highly three dimensional velocity field produced by the impeller.

Unfortunately, adequate steady and unsteady data, representative of today's advanced high speed compressors, are limited in the open literature. From a high cycle fatigue (HCF) perspective, El-Aini et al. [7], in a review of the limitations in predicting and designing for HCF, outlined that the current prediction tools fall short for forced response analysis of today's machines. The necessary development areas were indicated as high Mach number unsteady flows, strong fluctuations in incidence angle, unsteady separated flows, and cases of high incidence. Specific needs were identified to be not only prediction techniques but also experimental data for validation.

HCF is a key issue especially in the impeller trailing edge region due to the unsteady pressure fluctuations caused by the diffuser potential field, which is also a function of the flow structures present at the vane throat. In radial flow turbo pumps, it has been shown that these cyclic pressure variations imposed on the impeller trailing edge can be larger than the steady pressure rise across the machine [8,9]. Less information is available on the interaction in high speed air compressors. Characterization of this type of impeller-diffuser interaction is not only important from a HCF perspective but increased coupling could lead to larger tip leakage losses characterized by considerably larger entropy production at the impeller [6], and thus affecting the stage efficiency. One of the most important parameters in unsteady impeller-diffuser interaction is the ratio of the diffuser inlet radius to impeller exit radius, i.e., the radial gap. Ziegler et al. [4,5] acquired steady and unsteady (laser-to-focus) measurements at the impeller exit and diffuser throat regions while changing this parameter. These experiments thus worked to characterize the effect of impeller-diffuser coupling on efficiency, impeller flow structures, and unsteady diffuser loading.

In an impeller passage there are high and low momentum regions that are often referred to as the jet and wake, respectively. The intensity of these zones not only varies in the circumferential direction but also along the span [10]. As the flow emerges from the impeller, the blade forces are lost and the jet and wake undergo a rapid mixing process in the vaneless space. Even though some early models have assumed uniform flow due to this mixing in the circumferential directions downstream of the vaneless

Contributed by the International Gas Turbine Institute of ASME for publication in the JOURNAL OF TURBOMACHINERY. Manuscript received October 1, 2008; final manuscript received November 25, 2008; published online January 12, 2010. Review conducted by David Wisler. Paper presented at the ASME Turbo Expo 2008: Land, Sea and Air (GT2008), Berlin, Germany, June 9–13, 2008.

space, it has been shown by Gallier et al. [11] and Gallier [12] that the mixing process is not sufficient to produce a homogeneous flow region. On the contrary, the flow imposed on the vaned diffuser inlet is highly irregular and three dimensional. There are other results supporting this observed flow complexity in the vaneless space. Krain [13,14] used laser-to-focus measurements to show large unsteady vane inlet flow angle variations in both the spanwise and circumferential directions.

This diffuser inlet flow unsteadiness was shown to propagate far into the diffuser passage. The delayed mixing in the spanwise direction and the circumferential propagation of low momentum fluid from the impeller resulted in a highly three dimensional flow field throughout the diffuser [15]. The vaned diffusers are highly sensitive to the mean and unsteady incidence. The single most important parameter governing the channel diffuser recovery is the boundary layer blockage at the throat [16], and this is known to be a nearly linear function of the vane leading edge incidence [17].

From an operability aspect, often the flow range of a centrifugal compressor is limited by stall or choke of the vaned diffuser, and the most important portion of the diffuser is the semivaneless space between the leading edge and the throat of the diffuser vanes [18]. The blockage factor at the throat, which is based on the diffusion from the leading edge of the blade to the throat and the inlet condition of the diffuser channel, is determined by this part of the diffuser. The complexity of the inlet flow, specifically the periodic variations in the flow angle, has adverse affects on the performance and operability, if not managed well. It is also known that the mean and unsteady incidence relative to the vane suction surface is the significant parameter for diffuser performance and stall [18]. If the flow field in the diffuser is well understood, this creates potential for radial machines to have higher efficiencies and wider operation ranges.

Overall, the flow field through the impeller exit and vaned diffuser is three dimensional, coupled, and characterized by high levels of deterministic unsteadiness. The vane flow features include shocks [19], boundary layer/shock interactions [18], partial separation zones (concentrated in the hub leading edge) [15], varying inlet flow momentum, and incidence regions imposed on the diffuser vanes [13–15]. In addition, there is a high degree of interaction between the impeller and diffuser that prevents accurate analysis as isolated components [3,5]. To address the need for experimental characterization of this complex flow field, presented herein are the results from the particle image velocimetry (PIV) measurements in the diffuser passage of the Purdue high speed centrifugal compressor. This high-efficiency compressor features an impeller that produces a diffuser entry flow field typical of modern transonic compressors. The flow characteristics are analyzed from hub-to-shroud at several relative impeller-diffuser positions for operating conditions at low (on the choke line), nominal, and prestall loading.

2 Technical Approach

2.1 Experimental Facility. The Purdue high speed centrifugal compressor facility consists of an Allison 250-C30G turboshaft engine that drives the research compressor through a slave gearbox. The centrifugal test compressor includes an advanced design 50 deg back-sweep impeller that consists of 15 full and splitter blade pairs upstream of 22 wedge-type diffuser vanes. The ratio of the diffuser inlet radius to the impeller exit radius is 1.094. The nominal operating speed of the compressor is 48,450 rpm. The design and nominal performance parameters of the research compressor are noted in Table 1.

The facility is instrumented with various steady temperature, pressure, and optical probes to measure rotational speed, mass flow rate, pressure ratio, and efficiencies. To change the speed of the test compressor, the C-30 engine output shaft speed is changed. The compressor is throttled with a butterfly valve at the exit of the outflow duct. The mass flow rate is calculated from the

Table 1 Research compressor parameters

	Impeller		
Tip diameter	21.65 cm		
Inlet diameter	14.2 cm		
No. of blades	15 full and splitter pairs		
Backsweep angle	50 deg		
Design speed	48,450 rpm (CW seen from inlet)		
	Diffuser		
Inlet diameter	23.67 cm		
Exit diameter	34.50 cm		
Axial passage width	1.384 cm		
No. of vanes	22		
Radial gap	1.094		
Diffuser inlet vane angle	79.4 deg		
Wedge diffuser opening angle	7.85 deg		

total and static pressures and the inlet total temperature measured with two rakes upstream of the test section. The pressure ratio (PR) is determined by the ratio of the mass averaged inlet total pressure and the mass averaged exit total pressure calculated from the measurements of four three-headed total pressure rakes distributed in four separate diffuser passages. Also, the exit gas temperature is measured at the exit plenum. From these measurements, the compressor's flow-pressure characteristic can be defined by the corrected speed and corrected mass flow rate.

2.2 PIV Velocity Measurements. PIV is an optical imaging technique that allows velocities in a flow field to be measured. The flow is seeded with particles that track the fluid, and a planar laser light sheet is pulsed to illuminate these particles. An image of the particles is captured by a charge-coupled device (CCD) camera perpendicular to the plane of the light sheet. A second laser pulse and exposure is made after a short time delay to extract a second image of the flow field. During analysis, both of these images are then divided into smaller sectors, called interrogation regions, and an average velocity within each interrogation region is determined by advanced cross correlation methods.

The PIV configuration for these experiments consists of a Solo PIV Nd:YAG laser of 532 nm wavelength, a Hi-sense MKII CCD camera, and Nikon Nikkor 35 mm focal length camera lens. To synchronize the laser and the camera, Dantec DYNAMICS FLOW MANAGER, Version 4.71, software was used. This software also provided the necessary routines to process the PIV images. A BNC Model 555 pulse delay generator is used to generate a phase lag to a once-per-revolution trigger signal digitally generated by TTI LT-850 laser tachometer. This allows data acquisition of different relative impeller-diffuser positions.

The seeding was introduced by a Topas Model ATM 210/H aerosol generator using diethyl hexyl sebacate (DEHS) seeding fluid. The DEHS seeding fluid particles have a mean diameter of 0.25 μ m. A detailed analysis by Gallier [12] showed that they are small enough to track flow features as small as 0.5 mm with error limited to 1.09% of the true velocity in regions bounded by the sonic velocity in the diffuser region. Clearly, for shock structures where the length scale is small (a few mean free paths), the error is considerably larger. Based on the above analysis, the "smearing" of the shock interface to a more finite length scale in the range of 10^{-4} m is thus expected.

An iterative multigrid cross correlation routine with window offsetting was used to extract the velocity information. In the current analysis, raw images are subjected to an adaptive correlation routine with initial and final interrogation areas of 128 \times 128 pixels and 16 \times 16 pixels, respectively, with a 50% overlap applied among different regions at each four refinement steps.

Assuming an adiabatic process from the diffuser to the plenum, the total temperature throughout the diffuser is taken as that measured in the discharge plenum, and this with the measured velocity determines the local static temperature, local acoustic speed, and

021010-2 / Vol. 132, APRIL 2010

Transactions of the ASME



Fig. 1 Diffuser access window

thus the Mach number, M. The flow angles presented are measured from the centerline of the diffuser passage and taken positive in the clockwise direction.

2.3 Error Analysis. The PIV measured velocity $V = S\Delta p / \Delta t$, where Δp , S, and Δt are particle pixel displacement, optical magnification, and the time delay between the two consecutive images, respectively. Westerweel et al. [20] reported that the average measurement error for an interrogation analysis through cross correlation with window offset was approximately 0.04 pixels. Assuming the maximum particle displacement of 4 pixels (onefourth of the diameter of a 16×16 pixel interrogation window), this would imply a relative measurement error of $\delta(\Delta p)/\Delta p$ =1%. In this setup, the scale factor, S, is 12.108 pixels/mm. It is established by comparing the images of a high precision Max Levy DA039 line grid ruler with the distances indicated by the grid. Considering the magnification factor, the grid lines appear 0.92 pixels wide through the CCD camera; thus the scale is known to be within a pixel. Using this information, the uncertainty due to optical magnification is $\delta(S)/S=0.41\%$. In this experiment, the time delay between the two consecutive images of the CCD camera is the minimum allowed by the hardware limitations, Δt $=10^{-6}$ s. The architecture of integrated circuits, such as those found on a computer board, may introduce fluctuations in the time delay, on the order of 1 ns. Thus the uncertainty associated with the time delay can be calculated as $\delta(\Delta t)/\Delta t = 0.1\%$. For the above considerations, the uncertainty in the velocity measurements based on the experimental setup is given by

$$\delta(V)/V = \pm \left[(\delta(S)/S)^2 + (\delta(\Delta p)/\Delta p)^2 + (\delta(\Delta t)/\Delta t))^2 \right]^{1/2}$$
(1)

using the formula outlined by Kline and McClintock [21] for a three-parameter model, which in this case is equal to 1.08%.

2.4 Experiment Conditions. In these experiments, the diffuser flow field in the Purdue centrifugal compressor is investigated utilizing PIV. Experiments are conducted in the vaned diffuser at three spanwise locations: 25%, 50%, and 75% spans, referred to as the hub, mid, and shroud planes. At all spanwise locations, five relative diffuser-impeller locations are considered, with the impeller full-splitter blade passage pair divided into five equally spaced phase delays, referred to as Delay 0, Delay 1, Delay 2, Delay 3, and Delay 4. The data presented are ensemble averaged with 200 images with a local minimum of 25 valid time instances for each vector flow field map. The window over the diffuser section where data are acquired is shown in Fig. 1.

Experiments are conducted at three operating conditions representative of low, nominal, and prestall loading conditions [22]. The corresponding corrected speeds ($N_{\rm cor}$), corrected mass flow rates ($m_{\rm cor}$), and total-to-total stage PRs are given in Table 2.

Table 2 Testing conditions

Loading	$N_{ m cor}$	$m_{\rm cor}$ (kg/s)	PR
Low	$101.87\% \pm 0.06\%$	2.326 ± 0.009	4.0452 ± 0.004
Prestall	$101.23\% \pm 0.06\%$ $101.08\% \pm 0.06\%$	2.296 ± 0.009 2.023 ± 0.008	4.2414 ± 0.004 4.5556 ± 0.004

3 Results

3.1 Vaneless Space: Diffuser Inlet Flow. Prior to analyzing the diffuser results from this study, it is useful to characterize the relationship between structures in the impeller exit flow and those in the diffuser flow field. The impeller discharge flow, as described by Dean [16], consists of two main regions characterized by high and low relative momentum fluids referred to as the jet and wake. Although later authors have refined the details of this model, this basic description of the exit flow of the impeller is both accurate and useful. The wake region, accumulating at the corner formed by the suction surface and the shroud, is a low velocity region in the impeller frame of reference. Due to the high tangential component created by the wheel speed in the fixed frame of reference, the wake is identified as a high velocity region with a flow angle mainly in the tangential direction. The jet region is a high velocity region in the relative frame but similarly is observed as a low momentum region in the fixed reference frame.

PIV experiments conducted by Gallier [12] for the 90% speed line describe the flow in the vaneless space of the Purdue Centrifugal Research Facility. Gallier's results show a jet and wake structure persisting from the hub to the shroud at the impeller exit, with the extent of the wake fluid increasing toward the shroud, with the mean incidence on the vanes varied by 11.1 deg from hub to shroud. There were also significant circumferential variations in velocity and flow angle due to the jet/wake passing.

To visualize the relationship between the flow in the vaneless space and that measured in the diffuser, Mach contours reported by Gallier are matched with exemplary diffuser data gathered in this investigation at 100% speed for a single relative impellerdiffuser position (Fig. 2). Even though Gallier's investigation was conducted at lower speed, this illustration provides a qualitative interpretation of the flow features imposed on the diffuser. For illustrative purposes, local flow vectors, shown as black arrows superimposed on the contour plot, are drawn to be representative of the flow direction and are scaled to the local Mach numbers. The suction and pressure surfaces of the vanes and blades are indicated with an s or p, respectively.

With all data in the fixed frame, Vector A indicates the impeller jet flow imposed on the diffuser suction side wall, with less swirl and lower momentum than the wake fluid. Vector B shows the approach of the impeller wake toward the diffuser suction side wall. For the given impeller-diffuser geometry, with 30 full and splitter blades and 22 vanes, there are 1.35 blade passages imposed on the diffuser inlet at any given time. For the delay shown, there is impingement of yet another impeller jet on the semivaneless space region just before the diffuser throat, Vector D. In the prethroat region E, the flow coming from the upstream portion of the vane with higher momentum, Vectors B and C, interacts with the low momentum region of the impeller jet (Region D). As the fluid enters the throat, it accelerates due to area reduction. Downstream of the throat, Vector F, the flow transitions supersonic to subsonic. Further downstream, there is a more uniform diffusion region, represented by Vector G.

3.2 Diffuser Flow Field-Nominal Loading. Figures 3 and 4 present the diffuser flow field at the hub measurement plane for five time delays at nominal loading. Figure 3 is the phase locked ensemble averaged Mach number, and Fig. 4 is the corresponding flow angle; the flow angles presented are measured from the cen-

Journal of Turbomachinery

APRIL 2010, Vol. 132 / 021010-3



Fig. 2 Impeller flow interaction with the diffuser, Mach number distribution, and typical delay

terline of the diffuser passage and taken positive in the clockwise direction (i.e., with impeller rotation). In this coordinate system, increasing flow angles (more positive) indicate flow in a more tangential direction.

There is a Mach number increase just downstream of the leading edge of the vane suction surface, in what is termed the semivaneless space. The semivaneless space consists of the triangular region restricted by the suction side leading edge, pressure side leading edge, and the diffuser throat. The Mach number is approximately 0.9 in this region, consistent from the design criteria outlined by Pampreen [23] that suggest that designers avoid supersonic flows in this region, thereby avoiding prethroat shock structures and their adverse effects on boundary layer growth. This region is consistent with a blockage-driven acceleration of the flow about the suction side of the diffuser and is seen in all time instances. As shown in Fig. 4, circumferential flow angle variation across the entry to the semivaneless space is on the order of 12 deg.

In Fig. 4, the more negative (i.e., more radial) flow angle data reflect the passage of the jet flow structure into the diffuser. As explained in Fig. 2, due to the blade-to-vane ratio of 1.35, two impeller jet regions are seen to be imposed on the semivaneless space, most visible in Delay 3. Immediately after the acceleration at the start of the semivaneless space, the flow decelerates again, most clearly seen at Delays 0 and 3 in Fig. 3 where the Mach number changes from 0.8 to 0.7 before the throat is reached. The



021010-4 / Vol. 132, APRIL 2010

Transactions of the ASME



Fig. 7 Nominal loading, shroud plane, Mach number

Journal of Turbomachinery

APRIL 2010, Vol. 132 / 021010-5

Fig. 8 Nominal loading, shroud plane, flow angle α°



Fig. 9 Choke loading, mean Mach number

Fig. 10 Choke loading, mean flow angle α°

tions (Figs. 4 and 6). This is mainly due to the fact that at this

spanwise plane the effects of the jet flow are minimized.

behavior of the flow in this region is partly a function of the area increase with increased radius, as it is seen at all time instances to some extent. But the main cause is believed to be due to the jet flow, characterized by a large negative flow angle region by the diffuser throat (Fig. 4) interacting with the suction side acceleration region, identified by a locally higher Mach number in the semivaneless space (Fig. 3). This interaction creates a zone of rapid adjustment, where these two flow fields interact, identified by the locally lower Mach number region upstream of the throat.

The flow downstream of the throat seems to be fairly unaffected by the circumferential variations after M=0.8 is reached (Fig. 3). In contrast, near the throat, the flow is significantly affected by the impeller position. The flow reaches a supersonic Mach number, then shocks down to a subsonic flow, with the location of this shock transition and the maximum Mach number a function of the vane-impeller relative position.

Figures 5 and 6 present the flow Mach number and angles for the midplane region at nominal loading. Similar features to that of the hub plane are observed in the flow field. In Fig. 6, at the start of the semivaneless space the minimum flow angle has become more tangential, resulting in a maximum circumferential flow angle variation of approximately 8 deg. Also, comparing Figs. 3 and 5, the Mach numbers in the semivaneless space and the throat are elevated, consistent with the higher momentum levels expected at midspan. These results are consistent with an increasing fraction of the flow being wake fluid compared with the hub plane. The Mach numbers for the region downstream of the throat at the midspan are also elevated, approximately by 0.25 compared with the near hub measurement plane, expected by the higher inlet momentum flow. In the throat region, the Mach number variations with delay time are also larger when compared with the hub measurement location. At the hub plane, the flow downstream of the throat seems to be fairly unaffected by the circumferential variations after M=0.8 is reached.

Focusing on the near-shroud plane at nominal loading (Fig. 7), the trend of an increase in the overall Mach numbers as the shroud is approached continues. This is especially noticeable in the extension of the semivaneless space acceleration region as well as downstream of the throat. This feature is consistent with an increasing fraction of wake fluid. Comparing the flow angle data (Fig. 8) with other spanwise locations (Figs. 4 and 6), the flow is more in the tangential direction, and the circumferential variation across the entry to the semivaneless space has been further reduced to 4 deg. Also, the flow downstream of the throat appears to be much more uniform when compared with other spanwise loca-

3.3 Diffuser Flow Field-Loading Effects. Overall at nominal loading, there is a large inhomogeneity in the diffuser flow field dependent on not only spanwise measurement location but also on the impeller-vane relative position. To compare the effects of loading change on the spanwise variation in the diffuser flow, mean flow Mach number and mean flow angles are reported for loading conditions representing low loading on the choke line and the flow characteristics of near stall loading conditions. The mean reported data are acquired by the averaging of phase-resolved PIV data, and thus contain only the delay-independent information.

When the lower loading condition along the 100% speed line is considered (Figs. 9 and 10), higher velocities are observed when compared with the nominal loading case as expected with the increased mass flow rate. The velocity increase seems mainly in the hub and midplane, reducing the overall variation that is seen from the hub to the tip in the nominal loading case. Downstream, it can be seen that the role of the upper and lower surfaces of the vanes as suction and pressure side has reversed with the changing incidence on the vane (Fig. 10). Figure 10 also shows that the flow angles entering the semivaneless space are similar to those seen at nominal loading expected from the relatively small excursion in mean impeller exit flow angle (\sim 1 deg) expected in this case.

At nominal loading, maximum average Mach numbers in the throat region were M=1.2 at the shroud plane, with M=1.1 typical of the mid and hub measurement planes. This region is followed by a mild supersonic deceleration and finally by a normal shock that brings the flow down to subsonic velocities (Fig. 11). At choke loading, Mach numbers greater than 1.2 are now observed at all spanwise planes, with the highest throat Mach numbers now found in the hub plane. There is a clear supersonic acceleration region associated with the expansion fan that originates from the upper vane leading edge. The turning in this region is also indicative of these principle flow features (Fig. 12). Downstream of the expansion fan, the flow is decelerated down to a lower Mach number by an oblique shock and through a normal shock terminates to subsonic flow.

Figures 13 and 14 present the mean Mach number and flow angle data for the case where the compressor is throttled to a point just prior to stall/surge. At this condition, the flow in the throat is now subsonic at all spanwise planes. Starting with the hub flow, the maximum Mach number observed at the hub is only approximately 0.7 and that is located in the semivaneless space acceleration region (Fig. 13). Focusing on the hub plane in Fig. 14, when



Fig. 11 Nominal loading, midplane, typical delay, throat shock structures, Mach number (top), and flow angle α° (bottom)

compared with the previous loading conditions studied, in the aft sections of the diffuser passage more spatial flow angle variations are observed. At midplane, the trend of higher velocities toward the shroud seen at nominal loading is again observed. Also, for the midplane and shroud plane spanwise locations, the variation in flow angle downstream of the throat observed for the hub plane is significantly reduced, indicating less random unsteadiness. Comparing the choke, nominal, and prestall loading cases at the hub, the increasing nondeterministic unsteadiness is likely indicative of a hub flow breakdown as the stall is approached. From the velocity data at nominal and prestall loading, there is also evidence for a hub-pressure side unsteady separation, taking the form of a separation bubble at the leading edge similar to that described by Justen et al. [19] and Dawes [2]. Although the separation zone was not resolved sufficiently to directly characterize it from velocity measurements in the core, the flow in the leading edge region still provides evidence of its existence. It was shown by Dawes that an unsteady separation bubble at the pressure-side hub



Fig. 12 Choke loading, midplane, typical delay, throat shock structures, Mach number (top), and flow angle α° (bottom)

region is not necessarily phase locked to the impeller. Thus, a separation of this type would result in a higher variance calculated from the ensemble averaged vector maps.

For the nominal loading case, the variance in the ensemble about the throat region for all three spanwise locations can be seen in Fig. 15. The variance is calculated separately for the radial component, U, and transverse component, V, of the velocities and summed linearly. Notice the high variance zone at the hub leading edge around the pressure side. This occurs in a region of the flow field otherwise characterized by a low value of variance, and likely is a result of the mean flow deflecting due to a transient leading edge bubble. A similar region can also be seen for the midplane with reduced magnitude, which supports the idea that the separation region at the vane leading edge is more concentrated at the hub. Focusing on the shroud plane, the high variance region as seen in the other spanwise locations does not exist, indicating that the unsteadiness in this plane is mostly impeller



Journal of Turbomachinery

APRIL 2010, Vol. 132 / 021010-7



Fig. 15 Nominal loading, sum of variances (m^2/s^2)

phase locked.

When a similar variance analysis is conducted for the prestall operating point, the shroud plane variance is similar to the prior lower loading condition (Fig. 16). In the hub region, there is a large high variance zone supportive of the hub flow breakdown previously suggested in Figs. 13 and 14. Interestingly, in the midplane the variance near the leading edge is reduced, possibly due to some incidence alleviation from the hub flow blockage. However, the ensemble average velocity data in this region do not provide sufficient resolution to confirm this.

4 Summary and Conclusions

This study has provided detailed phase-resolved velocity information in the vaned diffuser of the Purdue centrifugal compressor through a PIV technique. The data at all loading conditions demonstrated that the flow field in the diffuser is, as expected, characterized by a much more complicated structure than that which would be associated with steady, uniform diffusion. Although mixing clearly occurs in the vaneless space, these data demonstrate that strong momentum variations still exist in both the spanwise and the circumferential directions in the diffuser.

The nominal loading case indicated that the circumferential variations, driven by the jet/wake flow structure exiting the impeller, are highest for the hub plane. At the midspan, the flow features are similar to the hub plane, but the variation is reduced. In the shroud region, the circumferential variations are of lesser magnitude, dominated by the large extent of the impeller wake in that region. In all cases, circumferential variations are dominant in the prethroat region, and much reduced after the throat. Hub to shroud variations in the diffuser passage velocities are increased as the



Fig. 16 Prestall loading, sum of variances (m^2/s^2)

compressor is throttled back from the choke line, with the hub flow gradually being starved for flow until, just prior to compressor stall, the hub flow begins to break down.

Diffuser throat structures were shown to vary significantly with loading at all spanwise locations. Near the choke line, the throat Mach numbers are supersonic at all planes. The flow then adjusts to the downstream subsonic flow through an oblique and then a normal shock. Further increase in loading results in a more mildly supersonic condition and an adjustment to subsonic flow through a normal shock. As the mass flow is further decreased, the entire diffuser flow field becomes subsonic.

Acknowledgment

This research was funded in part by the Rolls-Royce Corporation. This support is most gratefully acknowledged.

Nomenclature

- α = flow angle measured from diffuser passage centerline (+CW)
- δ () = absolute uncertainty
- Δp = particle pixel displacement
- Δt = time lag between consecutive images
- M = Mach number
- $m_{\rm cor}$ = corrected flow rate
- $N_{\rm cor}$ = corrected speed
 - S = optical magnification factor
 - V = local velocity

References

[1] Hathaway, M. D., Chriss, R. M., Wood, J. R., and Stazisar, A. J., 1993, "Ex-

021010-8 / Vol. 132, APRIL 2010

Transactions of the ASME

perimental and Computational Investigation of the NASA Low-Speed Centrifugal Compressor Flow Field," ASME J. Turbomach., **115**, pp. 527–542.

- [2] Dawes, W. N., 1994, "A Simulation of the Unsteady Interaction of a Centrifugal Impeller With Its Vaned Diffuser," ASME Paper No. 94-GT-105.
- [3] Inoue, M., and Cumpsty, N. A., 1984, "Experimental Study of Centrifugal Impeller Discharge Flow in Vaneless and Vaned Diffusers," ASME J. Eng. Gas Turbines Power, 106, pp. 455–467.
- [4] Ziegler, K. U., Gallus, H. E., and Niehuis, R., 2003, "A Study on Impeller-Diffuser Interaction—Part I: Influence on the Performance," ASME J. Turbomach., 125(1), pp. 173–182.
- [5] Ziegler, K. U., Gallus, H. E., and Niehuis, R., 2003, "A Study on Impeller-Diffuser Interaction—Part II: Detailed Flow Analysis," ASME J. Turbomach., 125(1), pp. 183–192.
- [6] Shum, Y. K. P., Tan, C. S., and Cumpsty, N. A., 2000, "Impeller Diffuser Interaction in a Centrifugal Compressor," ASME J. Turbomach., 122, pp. 777–786.
- [7] El-Aini, Y., deLaneuville, R., Stoner, A., and Capece, V., 1997, "High Cycle Fatigue of Turbomachinery Components—Industry Perspective," AIAA Paper No. 97-3365.
- [8] Arndt, N., Acosta, A. J., Brennen, C. E., and Caughey, T. K., 1990, "Experimental Investigation of Rotor-Stator Interaction in a Centrifugal Pump With Several Vaned Diffusers," ASME J. Turbomach., 112(1), pp. 98–108.
- [9] Arndt, N., Acosta, A. J., Brennen, C. E., and Caughey, T. K., 1989, "Rotor-Stator Interaction in a Diffuser Pump," ASME J. Turbomach., 111(3), pp. 213–221.
- [10] Chen, Y. N., Seidel, U., Haupt, U., and Rautenberg, M., 1996, "Jet, Wake and Intrinsic Motion in Impellers of Centrifugal Compressors," ASME Paper No. 96-GT-261.
- [11] Gallier, K., Lawless, P. B., and Fleeter, S., 2007, "PIV Characterization of High Speed Centrifugal Compressor Impeller-Diffuser Interaction," 43rd

AIAA/ASME/SAE/ASEE Joint Propulsion Conference, pp. 153-160.

- [12] Gallier, K., 2005, "Experimental Characterization of High Speed Centrifugal Compressor Aerodynamic Forcing Functions," Ph.D. thesis, Purdue University, West Lafayette, IN.
- [13] Krain, H., 1999, "High Pressure Ratio Centrifugal Compressor With Transonic Flow," ASME Paper No. FEDSM99-7801.
- [14] Krain, H., 2002, "Unsteady Diffuser Flow in a Transonic Centrifugal Compressor," Int. J. Rotating Mach., 8(3), pp. 223–231.
- [15] Ibaraki, S., Matsuo, T., and Yokoyama, T., 2007, "Investigation of Unsteady Flow in a Vaned Diffuser of a Transonic Centrifugal Compressor," ASME J. Turbomach., 129, pp. 686–693.
- [16] Dean, R. C., 1974, "The Fluid Dynamic Design of Advanced Centrifugal Compressors," VKI Lecture Series 66.
- [17] Connor, W. A., 1987, "Design and Off-Design Performance Prediction of High Pressure Ratio Centrifugal Compressors," VKI Lecture Series 1987-01.
- [18] Senoo, Y., 1984, "Vaned Diffusers," *Flow in Centrifugal Compressors*, Von Karman Institute for Fluid Dynamics.
- [19] Justen, F., Ziegler, K. U., and Gallus, H. E., 1999, "Experimental Investigation of Unsteady Flow Phenomena in a Centrifugal Compressor Vaned Diffuser of Variable Geometry," ASME J. Turbomach., **121**, pp. 763–771.
 [20] Westerweel, J., Dabiri, D., and Gharib, M., 1997, "The Effect of a Discrete
- [20] Westerweel, J., Dabiri, D., and Gharib, M., 1997, "The Effect of a Discrete Window Offset on Accuracy of Cross-Correlation Analysis of Digital PIV Recordings," Exp. Fluids, 23, pp. 20–28.
- [21] Kline, S. J., and McClintock, F. A., 1953, "Describing Uncertainties in Single-Sample Experiments," Mech. Eng., 75(1), pp. 3–8.
 [22] Carnell, W. F., 2002, "Aerodynamic Performance and Forcing Function Mea-
- [22] Carnell, W. F., 2002, "Aerodynamic Performance and Forcing Function Measurements in a High-Speed Centrifugal Compressor," MSME thesis, Purdue University, West Lafayette, IN.
- [23] Pampreen, R. C., 1972, "The Use of Cascade Technology in Centrifugal Compressor Vaned Diffuser Design," ASME Paper No. 72-GT-39.