OFF DESIGN FLOW MEASUREMENTS IN A CENTRIFUGAL COMPRESSOR VANELESS DIFFUSER

Ali Pinarbasi and Mark W. Johnson
Department of Mechanical Engineering
University of Liverpool
Liverpool, United Kingdom

ABSTRACT

Detailed measurements have been taken of the 3-d velocity field within the vaneless diffuser of a backswept low speed centrifugal compressor using hot wire anemometry. A 16% below and an 11% above design flow rate were used in the present study. Results at both flowrates show how the blade wake mixes out more rapidly than the passage wake. Strong secondary flows inherited from the impeller at the higher flowrate delay the mixing out of the circumferential velocity variations, but at both flowrates these circumferential variations are negligible at the last measurement station. The measured tangential/radial flow angle is used to recommend optimum values for the vaneless space and vane angle for design of a vaneless diffuser.

NOMENCLATURE

A,B,C - King's law calibration coefficients
E - Hot wire anemometer voltage
H,K - Directional coefficients for hot wire.
L - Radial distance from impeller outlet
R - Impeller outlet radius
U - Effective cooling velocity
U,U,U - Radial, Tangential and Axial velocity components
y - Tangential coordinate in measurement plane
z - Tangential distance between consecutive blade wakes
z,z - Axial coordinate in measurement plane

INTRODUCTION

The performance of centrifugal compressor diffusers is highly dependent on the flow delivered from the impeller. Measurements due to, for example, Krain(1988) and Farge and Johnson(1992) demonstrate that this flow is highly non uniform and exhibits strong secondary flows. Distinct jet and wake flow regions are observed with the wake on the shroud and the jet near the hub. A strong passage vortex exists which originates in the impeller due to secondary flows. In the diffuser the impeller blade wakes are added to this already complex 3-d flow.

The development of these complex flows in vaneless diffusers has been studied by for example, Inoue and Cumpsty (1984), Mizuki et al (1985), Senco and Ishida (1975). Theoretically the inclusion of vanes within the diffuser should vastly improve pressure recovery, but in practice a significant improvement is only achieved over a limited flow range. This is largely because of the difficulty in achieving sensible vane incidence angles over even a small flow range. The objective of the current study on a vaneless diffuser is to obtain detailed measurements of the flow from which optimum vaneless space and vane geometry can be established for the design of a vaneless diffuser. Pinarbasi and Johnson (1993) presented results for the design flow rate for the current impeller and diffuser geometry whereas the present paper is concerned with off design conditions.

EXPERIMENTAL PROCEDURE

A schematic of the low speed centrifugal compressor rig used in the study is shown in Figure 1. The impeller was a De Havilland Ghost impeller, the geometry of which is given by Johnson and Moore (1980). In order to bring the geometry of this old impeller closer to that of a modern backswept impeller for the current study, the original radial outlet section was replaced to provide a 30° backswept outlet angle as shown in Figure 2. The vaneless diffuser has straight walls and a constant cross sectional area. The geometry operating conditions and measurement stations are summarised in Table 1.

Three operating points were used in the study. The 'design' flow rate of m=0.1311 kg/s reported by Pinarbasi and Johnson (1993) corresponded to approximately zero incidence at the impeller blade leading edge. The 'below' and 'above' design flow rates of 0.1103 kg/s and 0.1450 kg/s were 16% below and 11% above the design flow rate.
flowrate. Detailed flow measurements within the impeller for similar flowrates are reported by Farge and Johnson (1990, 1992).

**Impeller Geometry and Operating Condition:**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet blade radius at the hub</td>
<td>R_h = 88.75 mm</td>
</tr>
<tr>
<td>Inlet blade radius at the shroud</td>
<td>R_s = 283.75 mm</td>
</tr>
<tr>
<td>Outlet radius</td>
<td>R_o = 454.6 mm</td>
</tr>
<tr>
<td>Backswept blade angle</td>
<td>β = 30°</td>
</tr>
<tr>
<td>Number of the blades</td>
<td>N = 19</td>
</tr>
<tr>
<td>Outlet blade span</td>
<td>b = 72.3 mm</td>
</tr>
<tr>
<td>Rotating speed</td>
<td>n = 500 rpm</td>
</tr>
</tbody>
</table>

**Measurement Locations:**

<table>
<thead>
<tr>
<th>Sta.</th>
<th>L/R_o</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.02</td>
</tr>
<tr>
<td>2</td>
<td>0.08</td>
</tr>
<tr>
<td>3</td>
<td>0.15</td>
</tr>
<tr>
<td>4</td>
<td>0.21</td>
</tr>
<tr>
<td>5</td>
<td>0.27</td>
</tr>
<tr>
<td>6</td>
<td>0.33</td>
</tr>
<tr>
<td>7</td>
<td>0.39</td>
</tr>
<tr>
<td>8</td>
<td>0.45</td>
</tr>
</tbody>
</table>

Table 1. Geometry, Operating condition and measurement locations

Figure 1. Schematic of Centrifugal Compressor Test Rig

Figure 2. Backswept Impeller Blade Geometry

**Instrumentation and Measurement Technique**

A triple hot wire probe was used to measure the velocities within the diffuser. A single wire (Dantec 55P11) was aligned circumferentially with a double wire (Dantec 55P61) arranged with each wire in the axial radial plane and at 45° to both the radial and axial directions. This mutually perpendicular arrangement of wires was therefore capable of resolving the axial, circumferential and radial velocity components and the directional sign of the axial component. It was assumed that the radial and circumferential components remained positive throughout the flow. There was no evidence in the results of either of these components reducing to zero and hence this assumption was justified.

The hot wires were connected to three constant temperature anemometer bridges. The wires were then calibrated in two stages in a wind tunnel following the procedure of Jongensen (1971). In the first stage, the velocity/voltage relationship was established with the wire perpendicular to the flow direction. King's law

\[ E^2 = A + B \cdot U_x^2 \]  

was then fitted to the data for each wire using a least r.m.s. error technique to establish the calibration coefficients A, B and C. The second stage of calibration was achieved by varying the wire orientation at fixed wind tunnel velocity to establish the directional coefficients K and H where

\[ U_x^2 = U_x^2 + K \cdot U_y^2 + H \cdot U_z^2 \]  

was fitted to the data for each wire using a least r.m.s. error technique to establish the directional coefficients K and H.

An optical shaft encoder provided a pulse for every 1/3° of impeller rotation. This was used to trigger the simultaneous sampling of the three anemometer voltages through a Microlink data acquisition unit. Readings from 57 measurement points spanning one of the 19 impeller passages were logged on each of 230 consecutive impeller revolutions. The triple wire sensor was traversed in the axial direction in order to provide a mesh of data points for each measurement plane.

**Analysis of Results**

The 230 readings obtained at each measurement point for each of the three anemometers were used to compute the mean tangential, radial and axial velocity components using the calibration coefficients. The flow rate was also computed by numerical integration of the radial velocity component over each of the measurement planes. The maximum deviation of this flow rate from the mean for all stations was 6.5% which gives an indication of the overall experimental accuracy.

**MEAN VELOCITY RESULTS**

The mean velocities on the eight measurement planes in the diffuser are presented in Figures 3-14. The radial velocity component is presented as a contour and the remaining velocity component in the measurement plane is shown as an arrow.

**Station 1**

The velocity distributions measured at the above and below design flow rates are shown in Figures 3 and 4. These flow patterns are very similar to those discussed by Farge and Johnson (1992) at the impeller exit. At the above design flowrate, a very distinct passage wake is observed on the shroud at y∗ = 0.3. A very strong velocity shear gradient separates this region from the rest of the flow. The blade wake is observed on the suction side between y∗ = 0.95 and 1.
Strong secondary velocities sweep high velocity fluid from the hub to the shroud near the suction side and low energy fluid from the blade and passage wakes in the shroud to hub direction. The large difference in flow angle between the blade wake and adjacent passage flow led to rapid mixing out of the blade wake at the design flow rate. (Pinarbasi and Johnson (1993)).

For the below design flow rate (Figure 4), the passage wake, jet and blade wake are less distinct. The location of the passage wake is similar, but is larger and is spread over a much larger proportion of the shroud. The secondary flows are also weaker, particularly the flow up the suction side, however there is still evidence of opposed secondary flows either side of the blade wake.

Station 2

Figure 5 shows that the secondary flows are still strong for the above design flow rate at station 2. The rapid variation in flow angle across the blade wake has indeed largely resulted in the mixing out of the blade wake which is observed at y/\(\gamma_o\)=0.1. This is in fact the blade wake from the neighbouring passage which has been convected to its current location by the tangential velocity component. The passage wake has moved more rapidly in this direction, because of its lower radial velocity and hence larger flow angle. The wake has altered shape and has spread over the shroud. It has altered little in size and hence the mixing mechanisms within this region are clearly much weaker than those in the blade wake region.

At the below design flow rate (Figure 6), the blade wake has been convected further than for the higher flowrate to y/\(\gamma_o\)=0.15, but again has substantially mixed out. The passage wake has been convected to y/\(\gamma_o\)=0.4, but has altered little in size. There are now only small variations of secondary velocity in the circumferential direction at this flowrate.

Station 3

The passage wake has continued to move and spread across the shroud between stations 2 and 3 for the above design flow rate (Figure 7). It has however continued to traverse faster than the blade wake which is still just discernable near the hub. Relatively strong secondary flows are still observed in Figure 7 with marked variations in the circumferential direction. However the variations in radial velocity in this direction have now decreased significantly from Station 1 with the velocity contours aligned parallel with the diffuser walls.

Mixing out in the circumferential direction is almost complete for the below design flow (Figure 8) and there is also some evidence of thickening of the hub boundary layer as the peak velocity in the jet region has been displaced towards the centre of the passage. The passage wake has now spread almost evenly along the shroud wall. The main variation in radial velocity in the circumferential direction is associated with the blade wake which is skewed across the passage. This skewing is due to the more rapid translation in the circumferential direction of the low energy fluid near the hub and shroud walls as compared with the high energy fluid in the jet.

Station 4

At station 4 for the above design flowrate (see Figure 9) the passage wake occupies most of the shroud surface. The blade wake is still detected near the hub. There is now some evidence of
thickening of the hub boundary layer from the displacement of the jet peak velocity region from the hub wall to \( z/z_o = 0.1 \). The strong axial velocity component near the hub walls is also an indication of diffusion as the jet slows from 22 m/s at station 1 to 17 m/s at the current station.

The below design flow (Figure 10) now closely resembles a Couette flow between the diffuser walls, although the hub boundary layer is still much thinner than that on the shroud, which is reflected in the off centre position of the peak velocity at \( z/z_o = 0.3 \).

Figure 6. Below Design Flow, Velocities at Station 2 (L/R_o = 0.08)

Figure 7. Above Design Flow, Velocities at Station 3 (L/R_o = 0.15)

Figure 8. Below Design Flow, Velocities at Station 3 (L/R_o = 0.15)

Figure 9. Above Design Flow, Velocities at Station 4 (L/R_o = 0.21)
Station 5 to 8

As the flow progresses through the diffuser, it continues to tend towards a Couette flow. This process is more advanced in the case of the below design flow as shown by Figures 12 (Station 6) and 14 (Station 8) than for the above design flow case at the same stations (see Figures 11 and 13). Figure 14 shows that at the last station for the below design flow, the peak velocity region is at z/z_r = 0.35, and the hub boundary layer is still 2 to 3 times thinner than its shroud counterpart. In contrast, for the above design flowrate (Figure 13), the peak velocity region is still very close to the hub wall (z/z_r = 0.1) and the hub boundary layer thickness is perhaps only 10% of that on the shroud. The reason for this difference is believed to be the stronger secondary flows observed at the higher flow rate up to station 3, which continue to extract low energy fluid from the hub boundary layer and hence reenergise it.

Figure 10. Below Design Flow, Velocities at Station 4 (L/R_e=0.21)

Figure 11. Above Design Flow, Velocities at Station 6 (L/R_e=0.33)

Figure 12. Below Design Flow, Velocities at Station 6 (L/R_e=0.33)

Figure 13. Above Design Flow, Velocities at Station 8 (L/R_e=0.45)
FLOW ANGLE RESULTS

Station 1

Figures 15 and 16 show the differences between the flow angle 
(Tan^(-1)(U/\mu)) for the two flow rates at the diffuser inlet.

At the above design flowrate (Figure 15) the flow angle variation 
is extreme with angles as high as 75° in the slow moving wake and 
as low as 10° in the rapidly moving jet. For the designer of a vaned 
diffuser the axial variation in flow angle can be accommodated by the 
use of a twisted blade. Large variations in the circumferential 
direction however, will inevitably lead to significant incidence angles. 
It seems likely that in the current unsteady flow higher transient 
incidence angles may be tolerated without separation than would be 
the case in a steady flow. However the magnitude of the incidence 
angles will still be a good guide to the likelihood of incurring 
significant separation losses. Figure 15 shows that the circumferential 
variation in flow angle is 30° to 35° which implies for a correctly set 
blade incidence angles of at least ±15° will result.

For the below design flowrate (Figure 16), the flow angle 
variations are more modest from 50° to 80° with only a 15° 
circumferential variation in flow angle. A diffuser vane twisted from 
around 70° at the shroud to around 55° at the hub would accept this 
flow with a maximum incidence variation of ±8° where it would be 
expected that significant separation losses would be avoided.

Station 2

The variation in flow angle for the above design flow rate has 
moderated considerably at station 2 (Figure 17) with a circumferential 
variation of around 20°. This would still lead to vane incidence 
angles of around ±10° which could induce significant separation losses.

At the below design flow rate (Figure 18), the circumferential 
variation near the shroud is still about 15°, but is less than 10° closer 
to the hub.

Station 3

At station 3 for the above design flow rate (Figure 19), the
circumferential variation in flow angle of less than 15° is sufficiently low to prevent significant separation losses from a diffuser vane. The circumferential flow angle variation for the below design flowrate (Figure 20) is now only 5° to 10°.

Figure 19. Above Design Flow, U0/U, Flow Angle Station 3

Figure 20. Below Design Flow, U0/U, Flow Angle Station 3

VANED DIFFUSER DESIGN

The results of this study suggest that when designing a vaned diffuser a vaneless space extending to $L/R_e=0.02$ would give good performance at below design flowrates with a blade twisted from about 70° at the shroud to 55° at the hub. However a much longer vaneless space is required at above design flowrates where $L/R_e=0.15$ is more appropriate and the blade should be twisted from around 60° near the shroud to 30° near the hub.

If a wide range of operation is required between the flow rates considered in the current study and fixed vanes are used in the diffuser, separation losses are inevitable because of the dependence of flow angle, particularly in the jet region, on flow rate as shown by Figures 19 and 20. However if a vaneless space extending to station 3 ($L/R_e=0.15$) is used a blade twisted from around 65° at the shroud to about 45° at the hub should limit the incidence angles to ±15° and minimise the separation losses.

CONCLUSIONS

1. The blade wake mixes out very much more rapidly than the passage wake within the vaneless diffuser.
2. At the above design flowrate the shear gradients associated with the blade and passage wakes and the secondary flows are much stronger.
3. Circumferential variations in velocity are prevented from mixing out rapidly at the higher flowrate due to the presence of the strong secondary flows.
4. The hub boundary layer is thinner than that on the shroud at the last measurement station. The ratio of the boundary layer thicknesses is around 2:1 for the below and 10:1 for the above design flowrate.
5. The optimum vaneless space for a vaned diffuser increases with flowrate, because of the slower mixing out of circumferential variations. Twisted vanes will also minimise incidence losses.
6. It is impossible to avoid significant incidence losses with fixed vane diffusers if a wide flow range is required, because of the inherent dependence of flow angle on flowrate.

REFERENCES


Jørgensen, F.E., 1971, "Directional sensitivity of wire and fiber film probes" DISA information No. 11, pp. 31-37.


