The Influence of Flow Rate on the Wake in a Centrifugal Impeller

Three-dimensional flows and their influence on the stagnation pressure losses in a centrifugal compressor impeller have been studied. All 3 mutually perpendicular components of relative velocity and stagnation pressure on 5 cross-sectional planes, between the inlet and outlet of a 1 m dia shrouded impeller running at 500 rpm were measured. Comparisons were made between results for a flow rate corresponding to nearly zero incidence angle and two other flows, with increased and reduced flow rates. These detailed measurements show how the position of separation of the shroud boundary layer moved downstream and the wake's size decreased, as the flow rate was increased. The wake's location, at the outlet of the impeller, was also observed to move from the suction surface at the lowest flow rate, to the shroud at higher flow rates.

M. W. Johnson
Department of Mechanical Engineering,
The University of Liverpool,
Liverpool, U.K.

J. Moore
Department of Mechanical Engineering,
Virginia Polytechnic Institute,
Blacksburg, VA

NOMENCLATURE

\( p \) static pressure

\( p_r \) reduced static pressure, \( p_r = p - \frac{1}{2} \rho \omega^2 r^2 \)

\( p^* \) rotary stagnation pressure, \( p^* = p + \frac{1}{2} \rho \omega^2 - \frac{1}{2} \rho c_w^2 r^2 \)

\( \rho \) fluid density

\( R_n \) radius of curvature of bend

\( W \) relative velocity

\( x/x_o \) meridional inlet-to-outlet co-ordinate measured along the shroud

\( y/y_o \) pressure-side to suction-side co-ordinate

\( z/z_o \) hub-to-shroud co-ordinate

\( \omega \) angular velocity

SUBSCRIPTS

\( \text{min} \) minimum value

\( \text{max} \) maximum value

INTRODUCTION

A 'jet-wake' flow pattern has been observed in the discharge flow of a centrifugal compressor impeller by many researchers (e.g., 1,2,3). However, the wake has been detected at different positions in the outlet plane. For example, Fowler (4) has located a wake on the suction surface at the outlet of his shrouded impeller (running with zero tip clearance), whereas Eckardt (5) observed his wake on the shroud. The location of the wake is known to be influenced by flow rate and running speed as well as impeller geometry, but even today design techniques cannot predict accurately either the wake's size or its location. This shortcoming must not only influence the resulting impeller performance, but also the efficiency of the diffuser which must accept the impeller discharge flow.

Objective

In this paper, results of measurements made in a de Havilland Ghost centrifugal impeller running at low speed (500 rpm) are presented. The authors' aim was to determine the main flow phenomena influencing the formation and development of the wake and how these phenomena are modified by changes in the flow rate. Accordingly, in the present study, all three mutually perpendicular components of relative velocity and the rotary stagnation pressure were measured on five cross-sectional planes between the inlet and the outlet of the impeller for three different flow rates.

Secondary Flow

In all types of turbo-machine secondary flows develop in the boundary layers and it is therefore fruitful to consider the secondary flow pattern we might expect to develop in a centrifugal impeller. Johnson and Moore (6) discussed this in some detail, and here we note their salient points.

1. The centrifugal and Coriolis forces associated with the impeller passage curvature and rotation induce gradients of reduced static pressure \( (p_r = p - \frac{1}{2} \rho \omega^2 r^2) \) across the impeller passage. Frequently in turbo-machines pressure gradients due to both curvature and rotation occur simultaneously and here the Rossby Number \( Ro = \frac{W}{\omega R_n} \).
2. Fluid with a low rotary stagnation pressure \((p^* = p + \frac{1}{2}u^2+\frac{1}{2}W^2-\frac{1}{2}pr^2)\) is found in the boundary layers. This fluid tends to migrate under the influence of secondary flows to regions of low reduced static pressure. These regions have been termed 'stable locations' by Johnson (7). Johnson also discussed how low \(p^*\) fluid may not necessary accumulate at its stable location, as it may be carried beyond this point by the inertia of the secondary flows. Nevertheless, the stable location still gives a good indication of where a wake might be expected to form.

3. The centrifugal compressor impeller has two bends, the inducer bend and the axial-to-radial bend and these will both induce secondary flows. Rotation will modify these secondary flows wherever the flow has a non-axial component of velocity. This occurs in the early part of the inducer, in the later part of the axial-to-radial bend and in the radial outlet section. Johnson and Moore therefore concluded that the stable location for low \(p^*\) fluid would move from the suction-side/shroud corner in the inducer to the shroud in the early part of the axial-to-radial bend and then pass back through the corner region to a point on the suction surface at the outlet.

Turbulent Mixing

Curvature and rotation modify the turbulent mixing which occurs in the boundary layers and in other regions of high shear. Johnston (8) and Bradshaw (9) have studied these modifications extensively, but here it is only necessary to note a simple rule for determining how mixing is modified. If in a region of high shear the cross stream gradients of rotary stagnation pressure \(p^*\) and reduced static pressure \(p_r\) are in the same direction, turbulent mixing will be suppressed. However, if the gradients are in opposite directions, mixing will be enhanced. In general, this means that mixing is suppressed significantly in the boundary layer close to the stable location for low \(p^*\) fluid.

**THE TEST RIG**

The test rig, which is described by Johnson (10) and Johnson and Moore (6), is shown in a schematic diagram (Fig. 1). The impeller, which rotates at 500 rpm, is from a de Havilland Ghost engine. A shroud is attached to, and therefore rotates with, the impeller. Measurements were made on five cross-sectional planes (see Fig. 2), using probes which rotate with the impeller. A 5-hole probe was used to measure all three mutually perpendicular components of relative velocity with an estimated accuracy of \(\pm 1 \text{ m/s}\) and \(\pm 5 \text{ degrees}\) for a typical reading of 10 m/s. A Kiel probe was used to measure the rotary stagnation pressure relative to atmospheric pressure with an estimated accuracy of \(\pm 10 \text{ N/m}^2\).

**EXPERIMENTAL RESULTS**

The results presented are for three flow rates. The first corresponds to approximately zero incidence angle at the blade leading edge and is therefore referred to as the 'design' flow rate. The other two flows are 'below design' and 'above design' and are for flow rates of 85% and 121% of the 'design' flow rate respectively. The mass flow rates for the three flows are given in Table 1.

**Presentation of Results**

Rotary Stagnation Pressures. The limiting values of rotary stagnation pressure \((p^*_{\text{max}}\text{ and } p^*_{\text{min}})\) are determined by consideration of the impeller flow. At each flow rate, the approximately uniform axial inlet flow gave a uniform value of \(p^*\) at the impeller inlet. This was the highest value of \(p^*\) measured anywhere in the impeller flow and is therefore referred to as \(p^*_{\text{max}}\). The lowest values of \(p^*\) in the impeller flow occur where fluid with low velocity is found in regions of low static pressure. \(p^*_{\text{min}}\) was therefore estimated from the measurements made in the boundary layer near the shroud surface in the inducer. The values of \(p^*_{\text{min}}\) and \(p^*_{\text{max}}\) for each flow rate are listed in Table 1.
Table 1. Impeller Flow Rates and Limiting Values of $p^*$

<table>
<thead>
<tr>
<th></th>
<th>Below Design</th>
<th>Design</th>
<th>Above Design</th>
</tr>
</thead>
<tbody>
<tr>
<td>Measured mass flow rate</td>
<td>0.121 (±.007)</td>
<td>0.142 (±.009)</td>
<td>0.172 (±.008)</td>
</tr>
<tr>
<td>(kg/s) in one impeller</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>passage</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Equivalent total mass</td>
<td>2.30 (±.13)</td>
<td>2.71 (±.17)</td>
<td>3.27 (±.15)</td>
</tr>
<tr>
<td>flow rate (kg/s)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>through impeller</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$p^*$ max (N/m$^2$ gauge)</td>
<td>-275 (±20)</td>
<td>-270 (±20)</td>
<td>-250 (±20)</td>
</tr>
<tr>
<td>$p^*$ min (N/m$^2$ gauge)</td>
<td>-565 (±30)</td>
<td>-600 (±30)</td>
<td>-650 (±30)</td>
</tr>
<tr>
<td>Mean incidence angle at</td>
<td>7$^\circ$ (±10$^\circ$)</td>
<td>3$^\circ$ (±10$^\circ$)</td>
<td>-2$^\circ$ (±10$^\circ$)</td>
</tr>
<tr>
<td>blade leading edge</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The pressure difference $p^*$ max $p^*$ min is used to define a dimensionless rotary stagnation pressure $P^*$ such that

$$P^* = \frac{p^* - p^*_min}{p^*max - p^*_min}.$$  \(1\)

Clearly, $P^*$ can only vary between zero and one in the impeller and is equal to one anywhere within the inviscid potential flow region.

A dimensionless reduced static pressure $P_r$ is also defined such that

$$P_r = \frac{P_r - p^* min}{p^* max - p^*_min}.$$  \(2\)

Relative Velocities. A 'passage direction' is defined such that in this direction the ratio of the distances from each pair of walls remains constant. The relative velocities are then resolved into two components; a component in the passage direction and a component perpendicular to this, that is, in the cross-passage direction. In the diagrams, the velocity component in the passage direction is represented by contours and the cross-passage velocity component by an arrow at each measurement point. These arrows will therefore give an indication of the combined effects of flow diffusion and secondary flow.

DISCUSSION

The Inducer Flow

At Station 2, which is close to the exit of the inducer, there are substantial differences between the $P^*$ contours for the three flow rates (Figs. 3, 4 and 5). The loss close to the pressure-side/shroud corner at the 'above design' flow rate (see Fig. 5) is probably due to a local separation induced by the high incidence angles at the blade leading edge (up to 20$^\circ$). However, in order to understand how much larger losses occurred at the lower flow rates, the flow in the suction-side/shroud corner region must be studied.

At each station, the lowest measured value of $P^*$, which always occurred near the suction-side/shroud corner, together with the reduced static pressure at the same measurement point are plotted in Fig. 6. This diagram then shows that an adverse pressure gradient in the streamwise direction occurs in this region at some point in the impeller for each of the three flow rates. This adverse pressure gradient will induce the separation of the boundary layer in this region. Let us define the probable location of this separation as the point where flow with a relative velocity less than 20% of the maximum relative velocity in the inlet plane was first observed (that is, where $P^* - P_r < 0.04$). Figure 6 then indicates that separation occurs near Station 2 in the 'below design' flow, near Station 3 in the 'design' flow and near Station 4 in the 'above design' flow.

The $P^*$ contours at Station 1 in the 'below design' flow (Fig. 7) and at Station 1 (Fig. 8) and Station 2 (Fig. 4) in the 'design' flow show that substantial $P^*$ losses occur upstream of the point of separation. The loss mechanism responsible can perhaps be best understood by considering the plight of low $P^*$ fluid in the shroud boundary layer at Station 1. Regions with $P^*$ as low as zero are present here, but they do not appear in the figures because the measurement probes could not be brought close.
enough to the shroud wall. Such low energy fluid must mix with fluid with a higher $p^*$ value in order to avoid being brought to rest by the adverse pressure gradient. The $P^*$ plot in Fig. 6 suggests that fluid particles in the suction-side/shroud corner region mix with lower $p^*$ fluid until their momentum is depleted at which point they start mixing with higher $p^*$ fluid, thus maintaining a small, but positive, velocity in the streamwise direction.

It is perhaps surprising that large mixing losses occur in the inducer as turbulent mixing is suppressed in the suction-side/shroud corner region by both Coriolis and centrifugal forces. However, at Station 2 in both the 'design' and 'below design' flows, the measured pressures in this corner region were found to be very unsteady. The authors therefore believe that the separation of the shroud boundary layer is probably an unsteady flow phenomenon and that periodic reversed flow may occur in this region.

The Axial-to-Radial Bend

Station 3 is about halfway through the impeller and the axial-to-radial bend has turned the flow through about 58 degrees. The velocity diagram for
similar to that observed in a stationary bend by Hawthorne (11) and therefore the curvature of the axial-to-radial bend can be considered to dominate the secondary flow pattern. However, close to the shroud the affects of Coriolis acceleration are noticeable. A secondary flow which transports low $p^*$ fluid from the shroud boundary layer towards the suction-side corner region has developed. This secondary flow is of similar strength to that on the suction surface as might be expected as the Rossby Number for the axial-to-radial bend is approximately one.

At all three flow rates an accumulation of low $p^*$ fluid occurs in the suction-side/shroud corner region. The size of this wake region is larger in the 'below design' case (Fig. 10) and smaller in the 'above design' case (Fig. 12) than for the 'design' flow rate (Fig. 11) as might be predicted from the differing losses which occurred in the inducer.
The passage is almost radial at Station 4 and therefore the stable location for the wake fluid will be on the suction surface. It is perhaps surprising therefore to find in the 'design' flow that the wake has moved the other way to a position on the shroud! (See Fig. 13.) The migration is due to the inertia of the suction-side passage vortex which developed in the axial-to-radial bend. Again, the flow pattern is similar in the 'above design' flow, but in the 'below design' flow the secondary flows generated by the curvature of the axial-to-radial bend are weaker (Rossby Number smaller) and so the wake remains in the corner region (see Fig. 14).

The Radial Section

At the outlet of the impeller, the passage is radial and therefore the stable location for low \( p^* \) fluid will be on the suction surface. However, the wake is only found here at the 'below design' flow rate (see Figs. 15 and 16). The wake's position is found to be strongly influenced by any increase in flow rate and hence Rossby Number (see Figs. 17 and 18). At these higher flow rates, the inertia of the vortex on the suction side of the passage prevents the wake from moving to its stable location. This vortex can still be seen at Station 5 in the relative velocity diagram at the 'above design' flow rate (Fig. 19, \( y/y_0 = 0.6 \) to 1.0, \( z/z_0 = 0.6 \) to 1.0). The overall cross flow from the suction to pressure side is due to slip, as Station 5 is only about 5 mm from the impeller outlet.

Turbulent mixing is suppressed on the suction surface by Coriolis forces, in the radial section of the passage. Therefore it might be expected that
that there are two sections of the impeller where major losses occur. The first is in the inducer where the shroud boundary layer separated at the two lower flow rates. The severity of this separation and hence the loss increases with a reduction in the flow rate. The second region of loss occurs in the radial section between Stations 4 and 5. As already discussed, the upstream influence of rapid mixing at the impeller exit is believed to be responsible for the mixing losses at the 'below design' flow rate. The smaller losses at the two higher flow rates are attributed to mixing between the jet and the wake which is not suppressed near the shroud in these flows. It should be noted that downstream of the impeller the discharge flow must eventually mix out to a uniform flow. Large mixing losses are therefore likely to result in the diffuser at the higher flow rates where the impeller outlet flow is very non-uniform.

The reduced static pressure curves (Fig. 20) show where diffusion took place in the impeller. It is clear therefore that most diffusion occurred in the impeller at the 'design' flow rate. The diffusion is reduced in the 'below design' flow by the high mixing losses in the inducer and is smaller in the 'above design' flow because of the highly non-uniform flow in the axial-to-radial bend and outlet section of the impeller.

CONCLUSIONS

1. Significant total pressure losses occur in two regions of the impeller:

(a) In the suction-side/shroud corner region in the outlet. This is likely to occur between the wake fluid on the suction side of one passage with the fluid on the pressure side of the neighbouring passage in the 'below design' flow. Such rapid mixing will be less significant at the two higher flow rates where the rotary stagnation pressures near the suction and pressure surfaces are more nearly equal.

Mass-Averaged Pressures

Mass-averaged reduced static pressures and rotary stagnation pressures for each of the three flow rates are presented in Fig. 20. The $p^*$ curves show more mixing will occur in the radial outlet section between the wake and the jet at the two higher flow rates.

The total pressure plots (Figs. 15, 17 and 18) show that the size of the wake increases with a decrease in flow rate. The jet and the wake are quite distinct at the two higher flow rates, but in the 'below design' flow the two regions are less well defined. This merging of the jet and wake may be due to the upstream influence of rapid mixing at the
inducer at both the 'below design' and 'design' flow rates, and

(b) in the final radial section of the impeller at all three flow rates.

2. The losses in the inducer are the result of an adverse pressure gradient in the streamwise direction in the suction-side/shroud corner region, which causes the separation of the boundary layer. The separation point moves downstream as the flow rate is increased until, at the 'above design' flow rate, separation losses are entirely avoided in the inducer.

3. Secondary flows develop in the axial-to-radial bend of the impeller and these transport low p* fluid from the boundary layers towards the stable location near the shroud/suction-side corner. A wake forms in this region at all three flow rates. Turbulent mixing is suppressed close to the stable location and normally therefore little mixing between the wake and jet fluid occurs.

4. The losses which result in the final radial section of the impeller occur for two different reasons. In the 'below design' flow rapid mixing is believed to take place downstream of the outlet between the wake from one passage and the jet from the neighbouring passage. This mixing may then influence the flow in the radial section of the impeller and induce the stagnation pressure losses observed in the results. At higher flow rates, losses are induced by the inertia of the secondary flows developed in the axial-to-radial bend. The secondary flow up the suction surface directs the wake away from its stable location on the suction-side to a position on the shroud. Here, turbulent mixing between the jet and wake is not suppressed and therefore some mixing losses occur. At the below design flow rate, this secondary flow is much weaker and the wake remains close to its stable location.

5. At the outlet, the wake is located on the suction surface in the 'below design' flow, near the suction-surface/shroud corner in the 'design' flow and on the shroud in the 'above design' flow. It is concluded that the relative magnitudes of the secondary flows due to curvature and due to rotation generated in the axial-to-radial bend are responsible for the wake's position. The Rossby Number associated with this bend, therefore gives a useful indication of where the wake is likely to reside in the discharge flow.

ACKNOWLEDGMENTS

The authors wish to thank Rolls-Royce Limited for their financial support of this project and especially Mr. P. H. Timmis of the Rolls-Royce Helicopter Engine Group and Mr. P. Clark of Rolls-Royce, Bristol, for their encouragement.

REFERENCES


