LDA INVESTIGATION OF THE FLOW DEVELOPMENT THROUGH ROTATING U-DUCTS

S. C. Cheah, H. Iacovides, D. C. Jackson, H. Ji, and B. E. Launder
Department of Mechanical Engineering
UMIST
Manchester, United Kingdom

ABSTRACT

This paper reports results from the use of laser Doppler anemometry (LDA) to measure the mean and the fluctuating flow field in a U-bend of strong curvature, $R_c/D = 0.65$, that is either stationary or rotating in orthogonal mode (the axis of rotation being parallel to the axis of curvature). The data acquisition system enables a stationary optical fibre probe to collect flow data from a rotating U-bend sweeping past it. Three cases have been examined all concerning a flow Reynolds number of 100,000; a stationary case, a case of positive rotation (the pressure side of the duct coincides with the outer side of the U-bend) at a Rotational number ($4D/U_m$) of 0.2 and a case of negative rotation at a Rotational number of -0.2. Measurements have been obtained along the symmetry plane of the duct and also along a plane near top wall. The most important influence on the development of the mean and the turbulence flow fields is exerted by the streamwise pressure gradients that occur over the entry and exit regions of the U-bend. In the stationary case a 3-dimensional separation bubble is formed along the inner wall at the 90° location and it extends to about 2 diameters downstream of the bend causing the generation of high turbulence levels. Along the outer side, opposite the separation bubble, turbulence levels are suppressed due to streamwise flow acceleration. For the Rotation numbers examined, the Coriolis force also has a significant effect on the flow development. Positive rotation doubles the length of the separation bubble and generally suppresses turbulence levels. Negative rotation causes an extra separation bubble at the bend entry, raises turbulence levels within and downstream of the bend, increases velocity fluctuations in the cross-duct direction within the bend and generates strong secondary motion after the bend exit. It is hoped that the detailed information produced in this study will assist in the development of turbulence models suitable for the numerical computation of flow and heat transfer inside blade-cooling passages.

NOMENCLATURE

$D$ duct hydraulic diameter
$R_c$ radius of curvature of U-bend
$Re$ flow Reynolds number ($= U_m D/v$)
$Ro$ rotation number ($= 4D/U_m$)
$U_m$ bulk velocity
$U_x$ mean velocity component in the cross-duct direction
$u_x$ fluctuating velocity in the cross-duct direction
$U_z$ mean velocity component in the streamwise direction
$u_z$ fluctuating velocity in the streamwise direction
$x$ cross-duct direction
$y$ direction normal to duct symmetry plane
$z$ streamwise direction
$v$ fluid kinematic viscosity
$\Omega$ angular kinematic viscosity

1 INTRODUCTION

In modern gas turbines, internal cooling plays an important role in maintaining the operating temperature of turbine blades down to safe levels. Relatively cool air, extracted from the compressor stages of the engine, is circulated through internal cooling passages inside the turbine rotor blades. Flow and heat transfer inside the cooling channels of turbine rotor blades are affected by the presence of sharp U-bends, the use of heat transfer enhancing ribs, and also by Coriolis and rotational buoyancy forces generated by the rotation of the blades.
The engine designer needs to be able to develop efficient cooling systems that can produce the required cooling rates with the minimum of amount of coolant flow. In order to do so, the designer needs to have a clear understanding of the main features of the flow and thermal behaviour inside the cooling passages and also access to reliable numerical procedures that can produce quantitatively accurate flow and heat transfer predictions. Progress in either of these areas depends on the availability detailed flow and heat transfer measurements that show the effects of strong curvature, Coriolis forces, rotational buoyancy and roughness on the hydrodynamic and thermal development.

The effects of Coriolis forces and curvature on the development of two-dimensional boundary layers has been known for many years, Bradshaw (1973) Johnston et al (1972), the extra strain associated with either producing an effect on the turbulent stresses at least an order of magnitude greater than the modification to the strain field itself. As remarked by Bradshaw, this feature effectively means that eddy viscosity turbulence models (in which the percentage change in stress equals that in strain) cannot capture the effects of either Coriolis force or streamline curvature adequately without introducing flow-specific modifications to the modelling.

Of course, in the practical flows of interest here, the duct is not two-dimensional and the confining end walls (enforcing gradients in the third direction) induce secondary motions either in a straight rotating duct, (eg Moon (1964), Moore (1967)) or in a curved passage, (Chang et al (1983), Taylor et al (1982)). Paradoxically, this greatly increased flow complexity can sometimes simplify the task of turbulence modelling since these are effects directly on the mean velocity components whereas the influences discussed in the preceding paragraph act only on the turbulence. Now, the types of flow arising in blade-cooling systems entail the combined complexities of curved and rotating ducts. Moreover, space constraints dictate that the U-bend must be so tight that flow separation occurs, certainly downstream of the bend and possibly at entry also. These flows are, therefore, very much more complex than the unseparated flows in ducts with either curvature or Coriolis effects present that have been the subject of previous hydrodynamic studies.

An impression of the flow complexity can be inferred from the wall heat transfer data obtained by Guidez (1988) and, more recently and extensively by Wagner et al (1989, 1991). These investigations were of major significance, since for the first time they provided heat transfer data for rotating passages that were directly of relevance to the engine designer and which could also be used for judging whether the turbulence models employed in numerical computation of these flows produced realistic predictions or not. Nevertheless both for design and also for turbulence modelling purposes, there remains a strong need to advance our knowledge of these flows to a more fundamental level, at which the detailed flow and thermal processes that determine the overall heat transfer behaviour are clearly understood. In order to develop such a comprehensive understanding of these flows it is essential to have detailed measurements of the mean and the fluctuating flow fields to complement and provide a basis for understanding mapped distributions of local heat-transfer coefficient. This data the present study attempts to provide.

Here we report the results of a mapping of the mean and the fluctuating motions, using laser Doppler anemometry (LDA), in flows through a smooth U-bend of strong curvature that rotates in an orthogonal mode. The objective is to reveal how the combination of the Coriolis force and strong curvature influence the development of the mean and of the turbulent flow field inside blade cooling passages.

2 EXPERIMENTAL APPARATUS AND PROCEDURE

2.1 The Test Facility

In order to be able to match engine levels of Reynolds and Rotation numbers at large scale with a rig rotating at a sufficiently modest rate not to cause stress problems, it became clear that water was the desired working fluid. Moreover, using water removed several of the LDA problems that would have been present if air had been used.

The test rig consists essentially of a motor-driven turntable mounted in a 1.22 m diameter water tank. A U-bend of square cross-section 0.05 by 0.05 m, and curvature ratio $\text{Rc/D} = 0.65$ shown in Figures 1a and 1b, is mounted on a turntable, so that the curvature axis of the duct is parallel to the axis of rotation. Most turbine coolant passages have their axis of rotation normal to their curvature axis. The present mode of rotation was however chosen at this initial stage of our investigation because, with this arrangement, the geometrical plane of symmetry of the U-bend still remains a hydrodynamic symmetry plane. Numerical computations, aiming to assess how well turbulence models reproduce the combined effects of Coriolis and strong curvature, can consequently consider only one half of this 3-dimensional flow domain, thereby making it easier to produce solutions that are free of numerical error. The axial throughflow to the test rig is fed into the square duct through a passage that is built into the rotating turntable and the outlet stream exhausted into the open water tank. A combination of fine wire-meshes and a honeycomb section are located downstream of the flow entrance to the duct to ensure a uniform and symmetric flow into the U-bend section. Figure 1(c) shows velocity profiles along the horizontal and vertical duct centre lines at a distance of 3 diameters upstream of the bend entry, which for the stationary case demonstrate the degree of uniformity and symmetry produced by the entry arrangements. The test section is made of 0.01 m thick perspex for optical access by LDA laser beams.

The rotor/turntable can be driven at any required speed up to 250 rpm in both directions. A feedback control circuit utilises the input signals from a shaft rotary encoder to control precisely the rotational speed.

2.2 LDA Measurements

The LDA employed was a TSI two-channel, four beam fibre-optic system with frequency shifting on both the blue and the green channels. A 4-watt Argon-ion Laser was used to power the system and two counter processors (TSI 1980B) were used for signal validation. Subsequent data processing
LDA measurements were collected along two planes normal to the axis of rotation; the symmetry plane and a plane at a distance of 0.125 D away from the duct top wall. The fibre-optic probe, of focal length 135 mm, was mounted in a stationary position above the rotating turntable as seen in Figure 1b. The laser beams entered the top wall of the duct, with the LDA measurements being taken as the rotating duct swept across the beams. Simultaneous two-component velocity measurements were taken, with a coincidence window of 20 µs, at several selected radial locations. The position of each data point in the flow was indexed by an incremental rotary encoder mounted on the rotating drive-shaft. Measurements from each discrete angular location were then stored in separate data bins. The stationary probe was thus able to collect data along an entire circumferential traverse line of the rotating duct, as shown in Figure 2. Sufficiently detailed flow mapping of the duct flow was obtained by employing a 1/8-degree resolution of 2880 data bins. With artificial seeding of the flow, a total 60,000 - 80,000 data points was gathered in each traverse over a period of 3-4 hours. The maximum circumferential length of a single data bin, along the circumferential traverse line of the largest radius, was 0.6 mm. Within the straight sections, where the radius of the traverse lines was smaller, several adjacent bins of the data were combined to obtain the ensemble average results of a sample of up to 1,000 data points for each angular position. The total circumferential length of the combined data bins did not exceed 0.8 mm. To obtain the velocity distribution along a straight traverse line across the rotating duct, 3 circumferential traverses were
performed along 3 slightly different radii so as to envelope the required cross-duct plane (Figure 2a). The corresponding cross-duct distribution was subsequently obtained by interpolating the data of the enveloping circumferential traverses. Measurements within the U-bend were obtained along eleven equispaced radial traverses as shown in Figure 2b. Bi-linear interpolation was then used to construct profiles along radial lines.

Similar two-component LDA measurements of the same duct flow without rotation were also taken at various locations to provide a reference set for comparison. For all the tests the fluid temperature was maintained at a steady level within 0.2°C by cold water injection into the flow. The flow through the duct was monitored by an orifice run preceding the U-bend. The bulk velocity \( U_m \) was measured by a U-tube waster manometer to an uncertainty of less than \( \pm 0.25\% \).

The overall estimated uncertainties for the non-dimensionalised mean velocities in the streamwise and cross-duct directions \( U_z \) and \( U_x \) respectively, and of the corresponding turbulence quantities are:

\[
U_z \leq 0.02 U_m \quad \sqrt{U_z^2} \leq 0.02 U_m \\
U_x \leq 0.03 U_m \quad \sqrt{U_x^2} \leq 0.09 U_m^2
\]

In arriving at the above figures several effects were considered; the uncertainty of the rotor speed, phase uncertainty of the Doppler signal, uncertainty on wavelength and beam angle, velocity gradient broadening, velocity bias and flow-rate uncertainty associated with the orifice plate.

3 PRESENTATION AND DISCUSSION OF RESULTS

Measurements have been obtained along the symmetry plane \((y/D = 0)\) and along a plane near the top wall \((2y/D = 0.75)\), with \( Y \) denoting the vertical distance from the duct symmetry plane. A number of cross-duct traverses have been obtained covering a region from 3 hydraulic diameters \((D)\) upstream of the bend entry to 8\( D \) downstream of its exit. LDA traverses yielded measurements of the mean velocity components in the streamwise and cross-duct directions \( z \) and \( x \) respectively, measurements of the two turbulence intensities along these two directions and also of the related turbulent shear stress \( U_z U_x \). The four triple correlations associated with these directions have also been obtained.

Three cases have been investigated, all at a flow Reynolds number of \((Re = U_m D/\nu)\) of 100,000; a stationary U-bend case, a case with the U-bend rotating positively at a Rotation number \((Ro = \Omega D/U_m)\) of 0.2 and a case with negative rotation at \( Ro = -0.2 \). A positive rotation direction, as shown in Figure 2, is defined as one in which the duct trailing (pressure) side coincides with the outer side of the U-bend. Due to space limitations only a selection of the available measurements can be included.

The mean velocity measurements are summarized in the vector plots of Figure 3 which provide a picture of the overall flow development through the U-bend for the stationary and also for the two rotating conditions. The stationary U-bend results, Figure 3a, indicate that in the upstream tangent, a reasonably symmetric developing flow approaches the bend entry. At the bend entry there is a strong flow acceleration along the inner wall and flow...
deceleration along the outer wall which becomes stronger near the corner between the top and outer walls. As the flow progresses through the bend, the fluid begins to move faster over the outer wall. Along the inner wall, by the 90° location the flow has separated near the symmetry plane while, close to the top wall the flow is still attached. This development can be more clearly observed in Figure 4 which shows the measured profiles of the streamwise velocity. The two sets of velocity vectors at the 90° location indicate that, because of the development of secondary motion the flow has become strongly 3-dimensional. The secondary motion transports fluid from the outer to the inner side along the top wall (whereas along the symmetry plane the secondary motion has the opposite direction) and consequently separation is delayed near the top wall. By the 180° location, (the bend exit), the separation region along the symmetry plane has grown to 35% of the duct diameter, while near the top wall the reverse motion is weaker and confined to a narrower region. Both sets of measurements show that the fluid undergoes strong acceleration along the outer wall at the bend exit. Over the downstream region, the stationary U-bend measurements show that even though the reverse motion is stronger near the symmetry plane than near the top wall, reattachment occurs earlier along the symmetry plane. Recovery downstream of the reattachment point is also faster along the symmetry plane than along the top wall. At a distance of 6 diameters downstream of the bend, the velocity distribution on the symmetry plane is virtually symmetric while near the top wall the velocity is still greater near (what was) the outer wall.

A recent numerical investigation of flow through an identical stationary U-bend (Bo and Iacovides 1993) has shown that the fast recovery along the symmetry plane is caused by the strong secondary motion that persists over the first three downstream diameters.

The effects of positive rotation (Ro = 0.2) on the mean flow development are shown in Figure 3(b). In positive-mode of rotation the Coriolis force acts in the same direction as that due to bend curvature. Positive rotation (at Ro = 0.2) does not appear to affect the mean flow development significantly upstream of and within the first half of the bend. Downstream of the bend, because the Coriolis-induced secondary motion opposes the return of the high momentum fluid to the inner side of the duct, the separation region is twice as long as that of the stationary case (Fig 3b). Thus flow acceleration along the outer wall is stronger. In contrast to the stationary case, the reverse motion near the top wall is almost as strong as it is along the symmetry plane. However (again contrary to the non-rotating case), the separated flow region in the straight duct tangent is shorter near the upper wall than on the symmetry plane. Further downstream, the data indicates that the flow starts to approach fully developed flow conditions for a rotating duct. In agreement with earlier numerical studies (Iacovides and Launder 1991) of flow in rotating straight ducts, the high momentum fluid accumulates along the trailing side of the rotating duct which in this case coincides with (what was) the bend outer side.
The mean flow development under negative rotation (Ro = -0.2) is shown in Figure 3c. Under negative rotation, the Coriolis-induced secondary motion transports the low momentum fluid to the outer side of the duct. As a result, at the bend entry the flow along outer wall is considerably slower than in the stationary case. Indeed the results indicate that a small region of flow reversal develops at the symmetry plane. Again, this flow feature is more easily visible in the relevant velocity profile of Figure 4(a) at θ = 0°. Along the symmetry plane, the low momentum region along the outer wall extends over the first half of the U-bend. Near the top wall, the outer side fluid accelerates more rapidly over the first half of the bend. In the downstream region the negative Coriolis force leads to the development of a secondary motion which is opposite to that caused by the bend curvature. The symmetry plane fluid consequently begins to move across the duct towards the inner side causing reattachment to occur at around 2 diameters downstream of the exit. Along the top wall, in contrast to the stationary case, the reverse flow is as strong as it is on the symmetry plane. Immediately after reattachment (3D), the reversal of the secondary flow leads to the accumulation of low momentum fluid along the inner side near the top wall. By the 6-diameter location the flow again begins to approach fully developed conditions for a rotating duct, in which the high momentum fluid now moves to what was the inner side of the duct. The axial velocity profiles of Figure 4 basically present the same mean velocity information albeit in profile form which allows a larger scale to be adopted.

Figure 5 shows profiles of the streamwise turbulence intensity. In the first half of the bend, low inlet turbulence levels begin to rise near the outer side where the flow is decelerated. Along the symmetry plane, this initial increase in turbulence levels is stronger in the negative rotation case in which there is a small region of flow separation at the bend entry. At the 90° location, a thin region of high turbulence is formed along the inner side, caused by flow separation. As the separation region grows over the second half of the U-bend, the region of high turbulence expands from the inner side towards the exit centre. The outer side turbulence levels are suppressed over the second half of the bend probably due to the strong flow acceleration. The one exception to this development is the behaviour of the turbulence levels for the negative rotation case along the symmetry plane where the flow acceleration is not as strong. As the flow develops downstream, the measurements show a gradual return to a more uniform distribution of the streamwise turbulence intensities over a length of about six diameters, through an increase in the turbulence levels along the outer side and a corresponding reduction along the inner side. For the positive rotation case however, the outer side turbulence levels remain suppressed over a longer distance possibly because positive rotation, (as shown in Figures 3 and 4), causes a stronger flow acceleration along the outer side. Negative rotation leads to a more rapid reduction in the velocity of the outer side fluid downstream of the bend exit, this strong deceleration causes the turbulence levels there to recover more rapidly in the case of negative rotation. For the
The corresponding development of the turbulence intensities in the cross-duct direction is shown in Figure 6. Along the symmetry plane, the distribution of \( \sqrt{\frac{u'^2}{U_m}} \) is largely similar to the corresponding distribution of streamwise intensities but the levels of the cross-stream intensities are generally lower. In the case of negative rotation however, over the second half of the U-bend and up to one diameter downstream, the levels of the cross-duct turbulence intensity within the outer half of the duct are similar to or higher than those of the streamwise intensities at the corresponding locations. Near the top wall, the results show that, over the second half of the U-bend, the cross-duct turbulence intensities along the outer side are higher than the corresponding streamwise intensities for all three cases. The strong secondary velocity that develops along the top surface is probably responsible.

The distribution of the turbulent shear stress component within the cross-duct plane \( \overline{u_z u_x} \) is shown through the profiles of Figure 7. Since most of the significant shear stress variation occurs downstream of the bend, the profiles in Figure 7 begin from the \( z/D = 1.0 \) location. These measurements indicate that within the first three downstream diameters, turbulence shear stress levels rise significantly along the symmetry plane, especially for the case of negative rotation. The measurements also indicate that the shear stress distribution is significantly affected by rotation.
1D location, negative rotation increases the symmetry plane shear stresses while positive rotation has the opposite effect. Further downstream, the symmetry plane shear stresses are reduced with the reduction occurring more rapidly for the stationary U-bend case. Near the top wall, shear stress levels are generally higher. In the downstream region, the shear stresses near the top wall are higher for the case of negative rotation and considerably lower for the case of positive rotation. In contrast to the symmetry plane development, the decay rate of the shear stresses for the stationary duct case near the top wall is similar to the decay rate for the two rotating cases.

Finally the contour plots of Figure 8 present the variation of the triple moment, \( \overline{u^3} \), downstream of the 90° location. This component is typical of the triple-moment data which we plan to report more fully in a separate publication. The main feature of interest would appear to be the very strong rise in the level of this quantity downstream of the bend and the considerably differences in the profiles across the duct depending on the sense of rotation; in particular at 1D downstream negative rotation produces values of \( \overline{u^3} \) an order of magnitude greater than for positive rotation with the stationary results lying in between. A negative value of \( \frac{\partial u^3}{\partial x} \) implies a local diffusive gain of fluctuating energy: thus regions close to the inner and outer walls are gaining energy at the expense of the highly turbulent fluid in the core. The difference in the magnitude of the slopes near the wall helps explain why, for negative rotation, fluctuating velocity levels are higher and, for positive rotation, lower than in the stationary case. The decay of the triple product with distance downstream also seems to be consistent with the normal stresses and mean velocity becoming more uniform across the duct.

4 SUMMARY OF RESULTS

The present measurements reveal the combined effects of strong curvature and orthogonal rotation (with the axis of rotation parallel to the axis of curvature) on the core flow of a square duct. The stationary U-bend flow is dominated by the entry and exit plane streamwise pressure gradients. At the exit, separation raises the inner side turbulence levels and streamwise acceleration suppresses the outer wall turbulence. The reverse flow is stronger along the symmetry plane than near the top wall due to the effects of secondary motion. Reattachment occurs within the first 2 downstream diameters. Along the symmetry plane the mean velocity distribution rapidly becomes symmetric after reattachment, whereas near the top wall recovery proceeds more slowly. Moreover strong gradients in the variation of triple moments in the downstream region are expected to contribute towards the development of high turbulence by diffusing turbulence energy from the core to the near-wall regions.

Rotation (at \( \text{Ro} = 0.2 \)) is found to exert a strong influence in the development of the mean and also of the fluctuating motion. Positive rotation almost doubles the length.
of the separation bubble and also strengthens the reverse motion near the top wall. The laminarizing effect of flow acceleration along the outer side at the bend exit is prolonged by positive rotation, remaining influential over the first 3 downstream diameters. The triple moments in the downstream region also appeared to be dumped by positive rotation. Turbulence levels downstream of the exit plane are consequently lower for this mode of rotation. Negative rotation produces flow separation at the bend entry which raises the core turbulence levels within the U-bend. Negative rotation also increases the cross-duct turbulence intensity within the U-bend core relative to the streamwise turbulence intensity. Downstream of the bend exit, this sense of rotation causes the development of strong secondary motion that has the opposite direction to that of the secondary motion induced by the bend curvature. Along the symmetry plane, the high momentum fluid along the outer side is convected across the duct to the inner side thus confining the recirculation bubble to a length of 2 downstream diameters. Because negative rotation causes a strong deceleration of the downstream flow along the outer side and also increases the triple moments, turbulence levels downstream of the bend exit are generally higher in the presence of negative rotation.

The measurements obtained in this study thus show that at rotational Reynolds numbers relevant to blade cooling applications, rotation can substantially alter the development of the mean and also the fluctuating motion within and downstream of sharp U-bends. This conclusion is consistent with the findings of earlier heat transfer investigations of similar cases. As well as revealing the details of the flow development, the present measurements also provide test data for the assessment of turbulent models used in the computation of blade cooling flows.

5 ACKNOWLEDGEMENTS

The research has been supported by the SERC through grant GR/F89039. Outstanding technical assistance has been provided by Mr J Hosker and Mr D Cooper. Authors' names appear alphabetically.

REFERENCES

Bo T and Iacovides H, "The discretization of the turbulence transport equations in the prediction of flow and heat transfer in a rotating U-bend", 8th Int Conf on Num Meth in Lam and Turb Flow, Swansea, July 1993

Bradshaw P, "The Effect of Streamline Curvature on Turbulent Flow", AGARD ograph, No 169, 1973


