Experimental investigation of the effect of entry conditions and rotation on flow resistance in circular tubes rotating about a parallel axis

A. R. Johnson and W. D. Morris*

Results are presented of an experimental investigation into the influence on flow resistance of flow conditioning prior to the entry region of a circular sectioned tube rotating about an axis parallel to its central axis of symmetry. This investigation is part of a long term study into the effect of rotation on pressure loss and heat transfer characteristics in rotating coolant channels. It is shown that for fully developed flow, rotation has little significant effect on flow resistance in the normal laminar and turbulent zones. The transition region is, however, affected; the usual 'dip' in friction factor is replaced by a smoother transition from laminar to turbulent flow. For developing flow, however, it has been shown that rotation can significantly increase the flow resistance above the normal stationary correlations. This increase can be reduced by smoothing the flow with gauzes and flow straightening honeycombs prior to the entry region of the tube

Keywords: *gas turbines, turbomachines, flow resistance*

It is well known that the fundamental laws of science suggest that the conversion of thermal energy to its useful mechanical counterpart is most efficiently accomplished if the prime movers used operate with as high a maximum cycle temperature as modern constructional materials will tolerate. At the design level this necessitates that adequate provision be made for cooling those components which are directly exposed to the hostile thermal environments required. In many instances, for example the rotor blades in a gas turbine, the components to be cooled rotate and, if internal cooling is adopted, the coolant also rotates, with the component, as it flows through the internal passages. Most of the previous theoretical and experimental studies of the extent to which rotation influences the convective mechanism inside a rotating duct have been undertaken with circular-sectioned tubes constrained to rotate about an axis which is either parallel to or orthogonal to the centre line of the tube. The typical results of these studies have been reviewed elsewhere[1].

It is important to note that in order to convey coolant to such rotating passages requires the use of a delivery and venting flow system which itself will be influenced by rotation. If some form of network analysis is used to calculate overall pressure loss characteristics for the cooling system, it is strictly necessary to be able to make suitable allowance for rotation on all the geometric features of that flow system. For example, precise information for pressure loss prediction in circuit features such as bends, enlargements, contractions, etc which allow for rotation about some axis are virtually unknown; this paper attempts to illustrate this general deficiency in the technical literature by reference to a selection of detailed experimental measurements made with one particular rotating geometric flow feature. Specifically the influence on flow resistance of flow conditioning prior to the entry region of a circular-sectioned tube constrained to rotate about an axis parallel to its central axis of symmetry has been investigated. This work is part of a continuing programme to investigate systematically the expected pressure losses in geometric features which are commonly encountered in the design of rotating cooling systems. Fig 1 illustrates the schematics and scantlings of the flow system studied. Note that ducts with parallel-mode rotation could occur in the platform regions of gas turbine rotor blades or in the rotors of

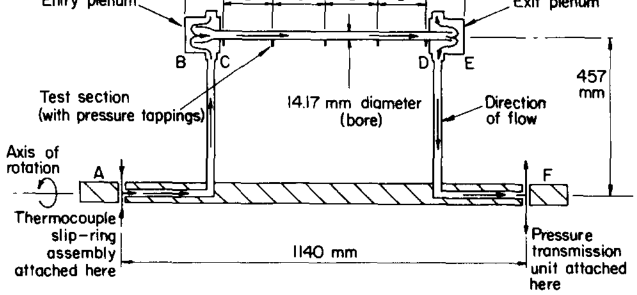

Fig 1 Schematic representation of rotating flow circuit

* Department of Engineering Design and Manufacture The University, Hull, North Humberside HU6 7RX, UK
Received on 8 July 1983 and accepted for publication on 26 October 1983

auxiliary electrical machines fitted to the turbine plant. On a larger scale, the flow geometry shown in Fig 1 is commonly encountered in the direct cooling of rotor windings for large turbine driven generators.

Formulation of the problem

Although, in general terms, the flow field in a rotating tube will be influenced by centripetal and Coriolis forces it has been demonstrated[1] that, for constant property flow, the centripetal forces are hydrostatic in nature and do not contribute to the creation of vorticity relative to the tube. Even so, with parallel-mode rotation of a tube, such as that shown as CD in Fig 1, the Coriolis forces can create secondary flows provided that streamwise gradients of velocity occur. This means, for example, that if a truly developed flow is present then the rotation will not alter the non-rotating flow field when laminar conditions prevail and no change in fully developed flow resistance will occur. Nevertheless, in the entry region the inevitable presence of axial gradients of velocity will interact with the Coriolis force field and a source for secondary flow generation will occur. Consequently, the flow resistance will be influenced as has been clearly demonstrated experimentally[2].

Strictly it may be argued that the velocity field development in the duct CD of Fig 1 depends on the flow conditions prevailing at the immediate entry plane, C, of the duct. The conditions at C, however, are also dependent on the upstream ducting, and on the manner in which rotation is affecting this upstream region. This means that any attempt to quantify pressure losses along CD cannot be uncoupled from the specification of upstream geometry. In an attempt to assess the likely severity of the influence of entry plane flow conditioning on subsequent pressure loss for this parallel-mode rotation of a circular-sectioned tube, a series of experiments were planned and executed with the tube fitted with a bell-mouthed entry section into which could be fitted a range of flow straightening devices. The detailed nature of these tests and the conclusions drawn now follow as an exemplification of the general problems of assessing the pressure loss characteristics of rotating cooling circuits.

Experimental apparatus

A rotor system to support the test section to be used was available as a result of previous investigations into the effect of rotation on heat transfer in cooling channels in turbo-generator rotor windings. This rotor system has been described in detail elsewhere[3,4]. Fig 1 shows schematically the rotating flow circuit section of the rig. Air was used as working fluid and enters the rotor via a sealing chamber at A, from a regulated compressed air supply. The air enters the circular test section CD via an inlet plenum chamber B which is described in detail later. The air exhausts from the test section at D into the exit plenum chamber F, and leaves the rotor via another sealing chamber at F.

The air flow was measured after leaving the sealing chamber F using Fisher 2001 rotameter flow meters before venting to atmosphere. To ensure accuracy, four flow meters were used to cover the flow range covered (2–500 litres/min), pressure and temperature being measured at each flow meter to permit corrections to be made for departures from calibration conditions. Further, during the experiments tests were repeated using different flow meters, where the ranges overlapped, to ensure repeatability.

The circular drawn brass test section, nominally 600 mm long by 14.17 mm bore diameter, was well supported by spacers inside a rigid aluminium tube to prevent any distortion during running. The test section had five equispaced pressure tappings along its length, with great care being taken to ensure that the tappings were burr and distortion free. This permitted pressure drops for tubes having four length/diameter ratios to be measured from the same experiment, thus permitting an assessment of entry length sensitivity. Further the final quarter span section of the test section gave the 'best approach' to fully developed conditions. The pressure signals were transmitted from the rotor via a five channel rotary seal unit which was attached to the main rotor by a flexible coupling. Rotary sealing was achieved by using a system of permanent magnets and a magnetic fluid. Full details of this leakproof method of transmitting the rotary pressure signals to stationary measuring equipment have been given elsewhere[5].

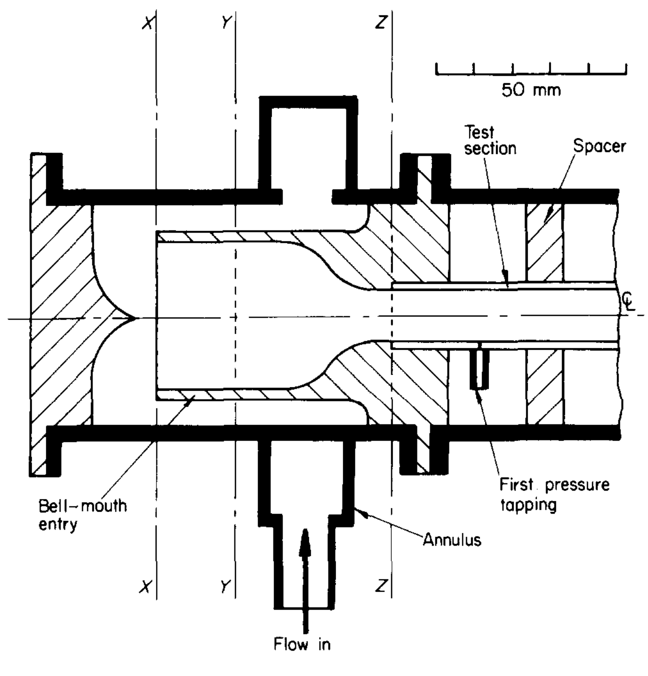

Fig 2 Test section inlet plenum chamber

Nomenclature

C_f	Blasius friction factor
d	Test section diameter
H	Test section eccentricity
J	Rotational Reynolds number
L	Test section length
Re	Pipe flow Reynolds number
ν	Kinematic viscosity
Ω	Angular velocity

Pressure measurements were made with Furness micromanometers or 'U' tube mercury manometers as appropriate to the pressure level being measured. To ensure accuracy, regular cross-checking of the pressure measurements were made between the various manometers when range overlapping occurred.

The air temperature in both the plenum chambers was monitored using chromel/alumel thermocouples; signals were transmitted to a computer logger via a silver/silver graphite slipring assembly attached by a flexible coupling to the rotor. The pressure signals and thermocouple outputs were recorded on a Solatron system 35 computer data logger which was also subsequently used for analysing the data.

As described above, the purpose of these experiments was to demonstrate the sensitivity to upstream flow conditions of subsequent pressure loss in a typical rotating channel. This was achieved by using a fixed geometry bell-mouthed entry section prior to the test section itself and fitting the bell mouth with a range of straightening devices. Fig 2 shows a detailed view of the inlet plenum chamber. The air enters the annulus and then enters the main plenum chamber via holes around the circumference of the annulus, before entering the test section via the bell mouth entrance. A flush brass ring (not shown) enabled a woven brass wire gauze screen to be positioned at the entrance section XX of the bell mouth, and similar screens could also be trapped between the inlet of the test section and the end of the bell mouth at section ZZ. A flow straightening honeycomb made out of very thin plastic straws could also be inserted between sections XX and YY. During manufacture care was taken to ensure that the bell mouth and the junction with the test section was smooth and step free. Further, to ensure air tight joints 'O' ring seals (not shown) were used at all joints.

By placing screen gauzes of varying mesh geometries at the locations XX and ZZ in Fig 2 and using the small diameter honeycomb straightener between XX and YY, it was possible to conduct flow resistance measurements over a range of flows and rotational speeds with a variety of inlet configurations. The full range of inlet geometry configurations studied is given in Table 1.

Table 1 Geometric description of test section and inlet configurations.

Test section details

Bore diameter = 14.17 mm
Length = 600 mm
Eccentricity = 457 mm
Length/diameter ratios
$\frac{1}{4}$ span = 10.6,
$\frac{1}{2}$ span = 21.2
$\frac{3}{4}$ span = 31.8,
full span = 42.3
Eccentricity ratio = 32.3

Inlet configuration details

	Inlet Type			
	A	B	C	D
Gauze P at section XX	No	Yes	Yes	Yes
Flow straightener honeycomb between sections XX and YY	No	Yes	Yes	Yes
Gauze Q at section ZZ	No	No	Yes	No
Gauze R at section ZZ	No	No	No	Yes

Gauze details

Gauze	Wire diameter, mm	Wire pitch, mm	Flow area, %
P	0.23	0.64	42
Q	0.17	0.42	35
R	0.13	0.32	34

Honeycomb Flow Straightener Details

Tube outside diameter = 4.25 mm
Tube thickness = 0.15 mm
Tube length = 18 mm
Tube configuration—touching with centres on a square grid.

Results and discussions

The flow system may be described geometrically for a specified inlet configuration in terms of the test section length, L, bore diameter, d, and the centre line eccentricity, H. By considering the basic momentum conservation equations it has been demonstrated[2] that the combined effect of the velocity field and rotation on flow resistance may be functionally related to the properties of the fluid concerned and geometry by the non-dimensionalised equations:

$$C_f = \phi[Re, J, L/d, H/d] \tag{1}$$

where C_f is the usual Blasius pipe flow friction factor, Re is the pipe flow Reynolds number and J is the rotational Reynolds number defined as:

$$J = \frac{\Omega d^2}{\nu}$$

where Ω is the angular velocity of the test section and ν is the kinematic viscosity of the fluid. The experiments were conducted with the range of non-dimensional parameters given in Table 2.

Tests were initially undertaken at zero rotational speed and a range of pipe flow Reynolds numbers to commission the instrumentation and data processing software and to furnish a reference data base with which subsequent experiments with rotation could be compared and contrasted. The results of these reference tests are shown in Fig 3 for each of the inlet configurations described in Table 1. When interpreting these results it should be noted that the

Table 2 Range of non-dimensional parameters covered

Centre line eccentricity/diameter ratio	32.3
Length/diameter ratio	10.6, 21.2, 31.8 and 42.3
Pipe flow Reynolds number	250–50 000
Rotational Reynolds number	0–1300

length/diameter ratios quoted are referred to the location of the first pressure tapping. This means that the inlet configurations cited include a short axial length (1.6 diameters) of tube between the true physical end and the location of the first pressure tap.

Fig 3(a) compares the friction factor–Reynolds number relationships obtained with each entry configuration for the 'best approach' to fully developed flow. Also shown are the theoretical relationship for laminar flow and the generally accepted correlation for turbulent flow with smooth tubes. The following points should be noted. For Reynolds number values up to approximately 1500, data for each entry configuration was encompassed, without any systematic individual deviation, by the dark bandwidth shown. Although a mean smoothing line in the notionally laminar flow region tended to be approximately 10% higher than the predicted value, the slope was in good agreement. With Reynolds number values greater than about 9000, data for all entry configurations also tended to fall into a fairly tight bandwidth with mean values about 6% greater than the accepted correlation. This figure is well within the accepted accuracy of the correlation. In the Reynolds number range 1500–9000, transition from notionally laminar to turbulent flow occurred along individual lines for each inlet configuration as shown. These results were checked to assure reproducibility and illustrate that even at the 'best approach' to fully developed flow with 31.8 equivalent diameters of calming length prior to the measured pressure drop, the entry configurations have individual signatures in the transitional region.

It is interesting to note that the change from an empty bell-mouthed entry to one with a frontal screen followed by the straightening honeycomb (ie the change from inlet configuration A to B) delayed transition dramatically. Further, the subsequent placing of the screens at section ZZ brought about progressively earlier transition as the coarseness of that screen was refined (ie the change from screen type Q to R at the location ZZ in Fig 2).

Figs 3(b)–(d) show the corresponding experimental results for length/diameter ratios of 10.6, 21.2 and 42.3 respectively. In the notionally laminar region the friction factor is significantly higher than the fully developed values although, once more, the bandwidths shown include all inlet configurations. For length/diameter ratios of 10.6 and 21.2 inlets A, C and D tended to give a smooth transition from laminar-like to turbulent-like. Only entry section B produced its own unique signature in these instances. The full span resistance characteristics, which contain in principle the greatest proportions of developed-like flow, tended to have similar trends to Fig 3(a) for the 'best approach' to fully developed flow.

Turning attention to the combined influence of rotation and inlet configuration on the 'best approach' to fully developed flow, it has been argued previously[6], from an examination of the momentum conservation equations, that the influence of rotation on flow resistance is likely to diminish as axial velocity gradients become increasingly small and developed flow is approached. This trend has been confirmed experimentally[2] and the data shown in Fig 4 has been constructed from this earlier work. The figure shows

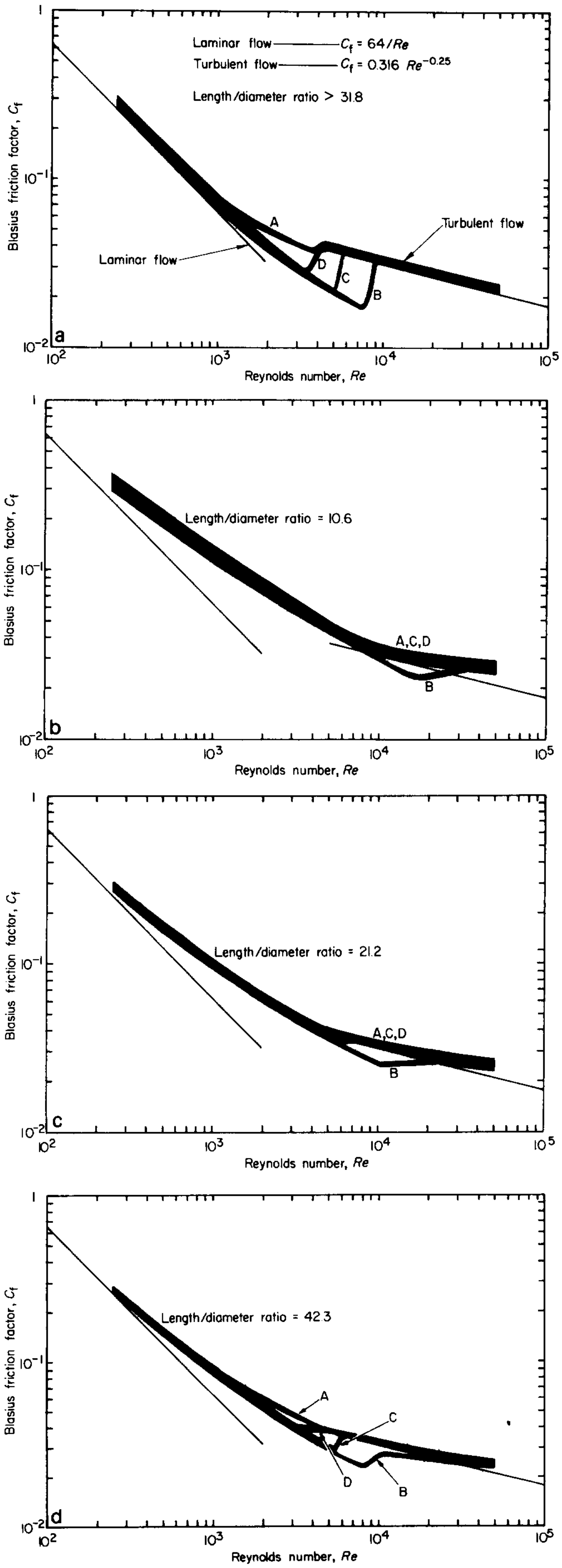

Fig 3 Influence of inlet configuration and length/diameter ratio on stationary friction factor against Reynolds number relationships

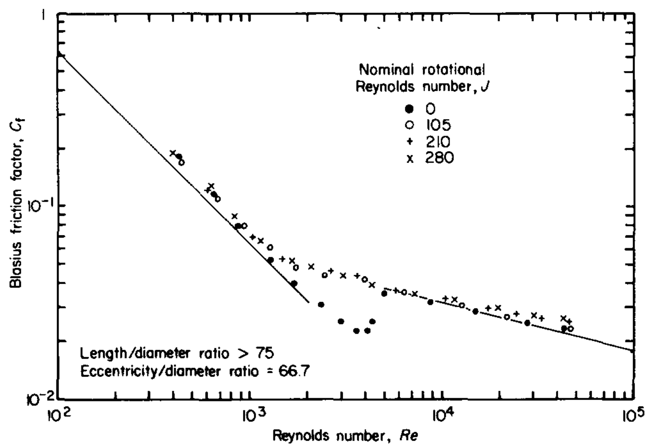

Fig 4 Influence of rotation on fully developed friction factors[2]

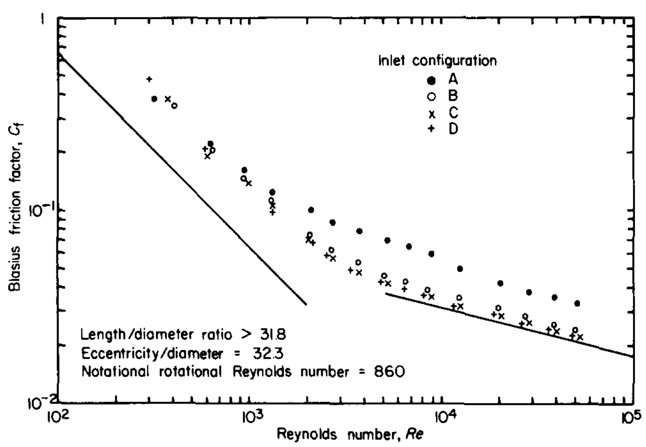

Fig 5 Influence of inlet configuration and rotation on best approach to fully developed friction factors

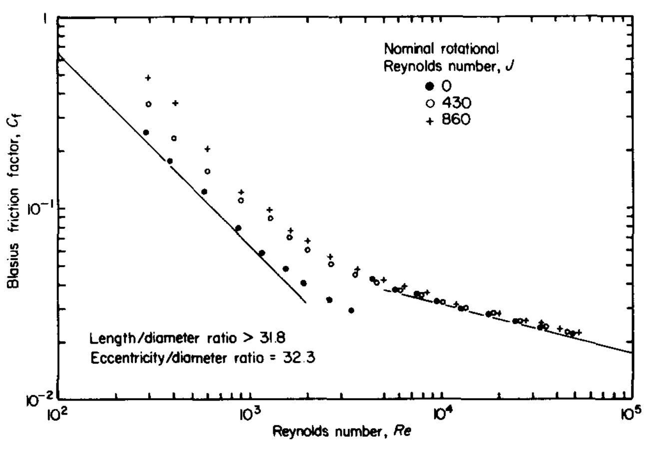

Fig 6 Influence of rotation on best approach to fully developed friction factor with inlet configuration D

the 'best approach' to developed friction factors for a circular tube having a nominal length/diameter ratio of 75.0, prior to the pressure loss measurement an eccentricity/diameter ratio of 66.7 and rotational Reynolds numbers in the range 0–280. This data was obtained with an entry configuration comprised of a straight tube having the same bore as the actual test section instead of the bell mouth used in the experiments reported in this paper. In the notionally laminar and turbulent regimes we note that rotation has little influence on the zero speed friction factor. However the transitional range of Reynolds number (say $2000 < Re < 6000$) was markedly influenced with a significant increase in friction factor. Note also the gradual change from laminar to turbulent flow which was apparent. Fig 5 shows how rotation was found to affect the 'best approach' to developed friction factors with the currently reported investigation for each of the four inlet configurations studied. It is clear that the influence of the entry plane velocity field has not yet decayed sufficiently with each inlet configuration for the claim to be made that rotation has no effect on developed flow resistance. It should be noted that the overall length/diameter ratio of the test section used was 42.3 whereas the equivalent figure for the earlier work reviewed by Fig 4 was almost double this. This is the most likely reason that rotation is still demonstrating a significant influence particularly with the inlets designated A (ie the empty bell mouth) and B (ie bell mouth with inlet screen P and straightening honeycomb). However in the range $6000 < Re < 50\,000$ the data gradually tended to approach the individual zero speed data with inlets C and D. These are the inlets fitted with screen gauzes at the immediate entry plane to the test section as well as the straightening honeycomb. Nevertheless, for notionally laminar flow ($Re < 2000$) there is a significant effect of rotation on friction factor with a tendency for all inlet data to merge into a faily tight bandwidth for Reynolds number values up to approximately 1000. The tendency for a relatively smooth transition from the laminar-like to turbulent-like behaviour noted in the reported data of Johnson and Morris[2] was still strongly in evidence.

Fig 6 presents additional typifying details of the effect of rotation on the 'best approach' to fully developed flow. In this figure inlet configuration D (ie the bell mouth with frontal screen P, straightening honeycomb and entry plane screen, R) has been selected. Rotation is still seen to influence results in the lower range of Reynolds number values studied with the same gradual blending into a turbulent-like behaviour which shows a small increase relative to the zero speed condition. Despite the fact that the 'best approach' to fully developed flow data available here is still suffering entry effects with some of the inlets studied, it is nevertheless evident, at least with Reynolds number values in excess of about 10 000, that with truly developed flow, rotation is not likely to have a significant effect on the zero speed friction factors provided care is taken to pre-condition the entry plane flow. In the range of Reynolds numbers usually associated with transition, however, significant impediment to flow resistance can be expected. Insufficient data is, as yet, available with which to make confident design-type proposals for this region.

It is with relatively short aspect ratio tubes that the combined effect of rotation and inlet configuration produces the greatest effect on friction factor. In the axial regions immediately downstream of the entry plane, the departure from a fully developed velocity profile is most marked and hence axial gradients of velocity are relatively high. This is precisely the condition which gives strong Coriolis action as dictated by the conservation of momentum principle[1]. It is in these pipe locations that strong Coriolis-induced secondary flows enhance mixing with a consequential increase in wall skin friction. This is shown in Fig 7 where data for a range of length/diameter ratios is presented for each inlet.

Fig 7(a) illustrates the friction factors measured with the shortest aspect ratio test section (ie a length/diameter ratio of 10.6). If it is argued that inlet configuration A is likely to produce the most disturbed entry plane velocity profile because no attempt is being made to smooth out the velocity field resulting from the re-alignment of the flow through the entry plenum, then it might well be expected that this will produce the most significant effect on friction factor when the test section is rotating. This is precisely what may be seen in Fig 7(a), where the increases in friction factor compared to the corresponding zero speed condition is severe. For this length/diameter ratio, progressive modification of the inlet by the inclusion of screens and the straightening honeycomb tends to supress the overall increase in friction factor when rotating. Beyond a Reynolds number of about 18 000 data for all inlets having some element of flow smoothing tended to gradually approach its own zero speed characteristic and became virtually coincident at the maximum Reynolds number tested.

Figs 7(b)–(d) show similar measurements made with test sections with length/diameter ratios of 21.2, 31.8 and 42.3 respectively. Similar trends to those described above for the shortest aspect ratio tube were evident.

Finally Fig 8 attempts to overview the data trends obtained with inlet D, the most smoothed entry plane flow studied. In this figure the friction factor has been plotted against the rotational Reynolds number for the entire range of tube aspect ratios and Reynolds numbers studied. The same qualitative trends were in evidence for each aspect ratio. Thus at the lower Reynolds number the increasing rotational Reynolds number produces a steady increase in flow resistance at all aspect ratios. As the Reynolds number increases, however, the influence of rotation becomes smaller until, at a Reynolds number of approximately 50 000 with the present range of variables, little increase is observed.

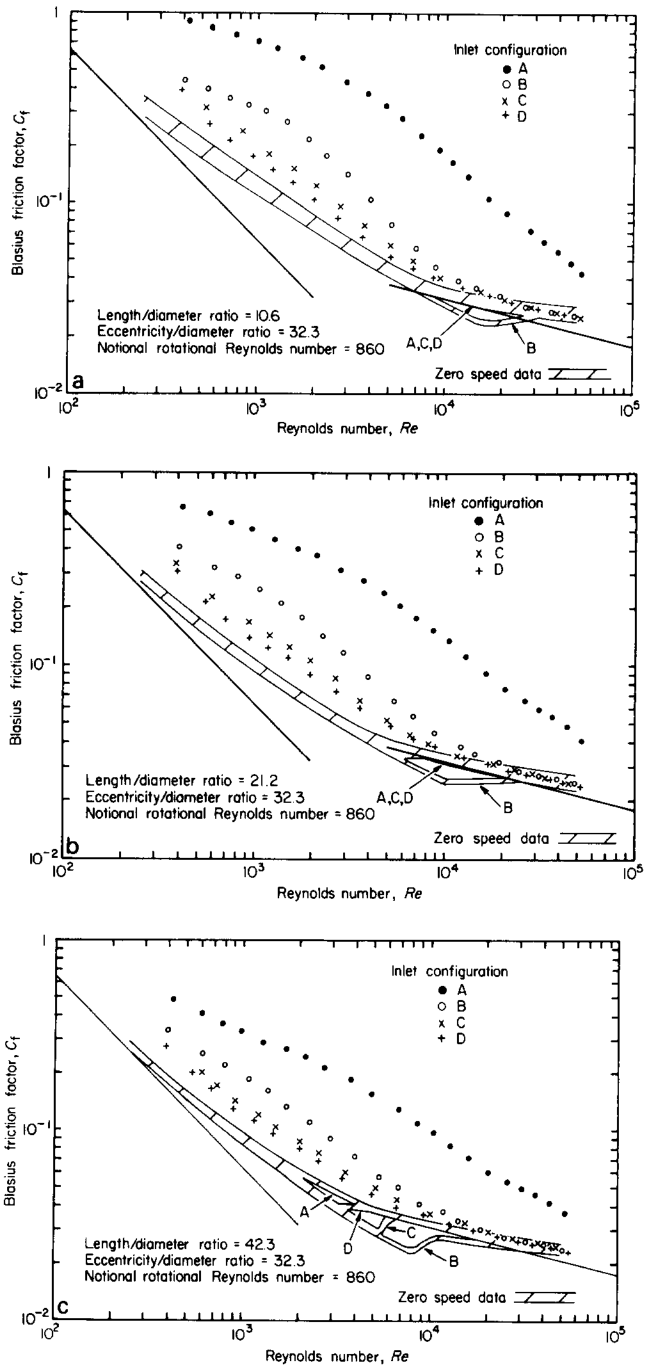

Fig 7 Influence of rotation, inlet configuration and length/diameter on friction factor against Reynolds number relationships

Closure

It has been demonstrated experimentally that the influence of rotation on flow resistance in a circular tube rotating in the parallel-mode can result in significant increases in pressure loss for a given flow rate over a range of operating conditions. Although this work is still in its preliminary stages and higher rotational speeds are to be studied in the future, there are a number of tentative conclusions which may be drawn at this stage.

If the tubes are long enough to establish a truly fully developed flow then rotation is not likely to have a significant influence on fully developed pressure loss in the range of Reynolds numbers commonly associated with laminar flow (ie up to 2000). This is also true for the notional turbulent range with Reynolds numbers in excess of about 15 000. However rotation does appear to give a more gradual transition

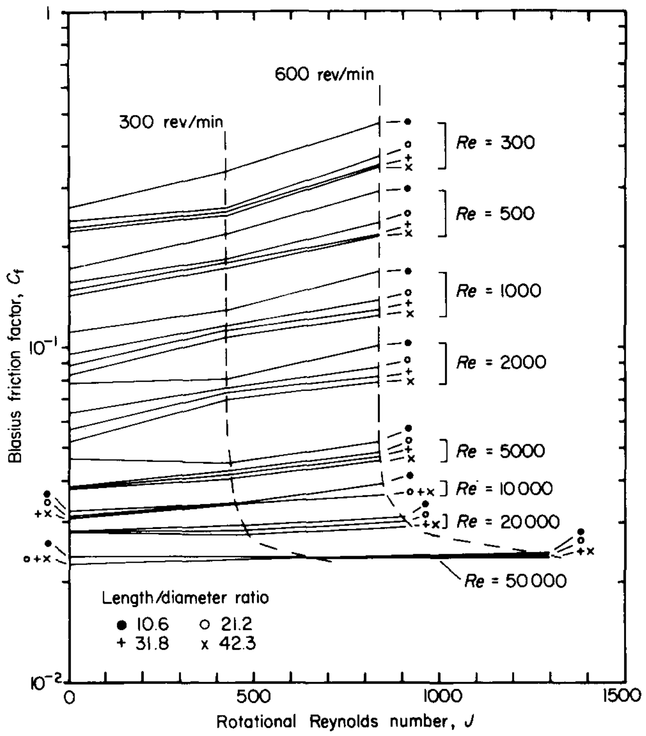

Fig 8 Influence of rotational Reynolds number, and length/diameter ratios, on friction factors for various Reynolds numbers with inlet configuration D

from laminar to turbulent flow in the Reynolds number range $2000 < Re < 15\,000$. In this range of Reynolds numbers a severe 'dip' in friction factor can occur with stationary tubes, so friction factors with rotation can be increased significantly in relative terms.

These comments are mainly applicable to situations where the deliberate attempt has been made to smooth upstream flow irregularities at the entry plane of the tube concerned. If this is not done (see data for inlet configuration A), then particularly large tube lengths in terms of equivalent diameters will be necessary before developed flow is produced; thus there will be severe increases in flow resistance for practical tube aspect ratios.

With relatively short aspect ratio tubes, the combined effect of entry plane velocity condition and Coriolis forces, as the manifestation of rotation, can cause noticeable increases in friction factor with this effect becoming progressively suppressed as the tube aspect ratio increases, ie as greater proportions of the tube length are dominated by fully developed flow.

No attempt is made at this stage to produce correlation-type equations because the work is currently being extended to cover a more extensive range of variables. This report is, consequently, intended to inject a note of caution when the flow and associated heat transfer characteristics of rotating cooling systems are being considered.

Acknowledgements

The authors wish to express their appreciation to the Central Research Laboratories of the Central Electricity Generating Board and the Science and Engineering Research Council for financial support for conducting this programme of work.

References

1. **Morris W D.** Heat Transfer and Fluid Flow in Rotating Channels. *John Wiley & Sons Ltd., Chichester, England. (Research Studies Press). (1982)*
2. **Johnson A. R. and Morris W. D.** Pressure Loss Measurements in Circular Ducts Which Rotate About a Parallel Axis. *Presented at the XIV ICHMT Symposium on Heat and Mass Transfer in Rotating Machinery, Dubrovnik, Yugoslavia, September* 1982
3. **Morris W. D. and Woods J. L.** An investigation of Laminar Flow in the Rotor Windings of Directly-Cooled Electrical Machines. *J. Mech Eng. Sci. (1974)* **16**, *408*
4. **Morris W. D. and Dias F. M.** Experimental Observations on the Thermal Performance of a Rotating Coolant Circuit with Reference to the Design of Electrical Machine Rotors. *Proc Inst Mech Eng (1976)* **190**, *46/76, 561*
5. **Morris W. D.** A Pressure Transmission System for Flow Resistance Measurements in a Rotating Tube. *J Phys Sci Instrum (1981)* **14**, *208*
6. **Morris W. D.** The Influence of Rotation on Flow in a Tube Rotating About a Parallel Axis with Uniform Angular Velocity. *J. Roy Aero Soc. (1965)* **69**, *201*

CALENDAR

International Symposium on Two-Phase Annular and Dispersed Flows	24–29 June 1984 Pisa, Italy	Paolo Andreussi, Dipartimento di Ingegneria Chimica, Universita di Pisa, Via Diotislavi 2, 56100 Pisa, Italy
2nd International Symposium on Applications of Laser Anemometry to Fluid Mechanics	2–4 July 1984 Lisbon, Portugal	Professor D. G. F. Durao, Department of Mechanical Engineering, Instituto Superior Tecnico, Avenida Rovisco Pais, 1096 Lisbon, Portugal
1st UK National Heat Transfer Conference	3–5 July 1984 Leeds, UK	Dr P. J. Heggs, Chemical Engineering Department, Leeds University, Leeds LS2 9JT, UK
NATO Advanced Study Institute: Natural Convection—Fundamentals and Applications	16–27 July 1984 Izmir, Turkey	Dr S. Kakac, Department of Mechanical Engineering, University of Miami, Coral Gables, FL 33124, USA