Bayley and Conway's Paper on Turbomachinery Internal Air Systems

J. W. Chew

**Faculty of Engineering and Physical Sciences,
University of Surrey,
Guildford, GU2 7XH, UK**

ABSTRACT

Bayley and Conway's 1964 paper [1], motivated by shortcomings in industrial design methods and understanding, was one of the first investigations of flow and heat transfer in turbomachinery rotating disc cavities. At the time of the study, a theoretical or numerical treatment was considered intractable and so experiments were undertaken. These paved the way for a large body of further research in this area. Today, use of computational fluid dynamics (CFD) in industry for internal air flow prediction is now commonplace. In this review of Bayley and Conway's paper it is shown that the unshrouded disc cavity flow considered is still challenging for modern CFD methods, and so the experimental data remains of interest to researchers in the field.

1. INTRODUCTION

The past fifty years have seen spectacular developments in turbomachinery technology for aerospace propulsion, power generation and other applications. This is illustrated in the relatively recent paper by Ruffles [2] which considers developments in aircraft propulsion and gives examples of engine developments at Rolls-Royce. Figure 1, taken from this paper, shows how engine specific fuel consumption (sfc) has improved between 1958 and 2000. According to Ruffles, aircraft fuel burn reduced by around 60% in this period relative to the first commercial jet aircraft, with 45% due to engine sfc improvements and 15% as the result of aircraft improvements. Not surprisingly, the most rapid sfc improvements occurred in the earlier years, but it may also be noted that figure 1 shows continuing performance improvements.

Such advances in engine technology at Rolls-Royce and other companies represent significant achievements by the Engineering profession as a whole, and many individuals. It would perhaps be invidious to single out specific areas of the technology or individual contributions in such a large field, and that is not the motivation here. Rather the opportunity is taken to reflect on one particular aspect of turbomachinery technology that is of particular interest to the writer. This serves to illustrate how technology has advanced in the last 50 years and how research work published in this (and other) journals can have impact and be of lasting value.

The research discussed here is related to the fluid dynamics of turbomachinery internal air systems. Bayley and Conway's paper [1] published in early 1964 by the Journal of Mechanical Engineering Science is one of the earliest examples of research in this area. Conway was supported by the Institution of Mechanical Engineering through a James Clayton Fellowship, and this paper was a precursor to much further research in this area. A brief introduction to turbomachinery internal air systems is given below in section 2. Bayley and Conway's work and other developments are discussed further in section 3, and then modern predictive methods are put to the test

in section 4 through comparison with Bayley and Conway's experimental data. Closing remarks and conclusions are given in section 5.

Figure 1. Aeroengine performance trends (Ruffles [2]).

2. ROLE OF THE INTERNAL AIR SYSTEM

The internal air system plays a vital role in modern gas turbines. It serves to dissipate heat generated outside the main gas path due to work done by the rotors and heat conducted to the discs, deliver air for turbine blade cooling, maintain the required axial pressure loading on the bearings, prevent hot gas ingestion from the main annulus overheating the turbine discs, and isolate the oil system. A section of the internal air system showing the high and intermediate pressure sections for a three shaft engine is shown in figure 2. The later stages of the high pressure (HP) compressor, the combustor, the HP turbine and the intermediate pressure (IP) turbine are shown in white. The colour coding for the internal air flows relates to the compressor stage from which the cooling air originates. For example, the red areas are fed from the HP compressor delivery. Green areas denote the presence of oil in the bearing chambers. With up to 25% of compressor air being used for cooling and sealing, the design of the air system has a significant impact on engine efficiency. Considerable attention is given to the internal air system and its interaction with the main annulus flow during engine design.

A number of different flow areas are labelled on figure 2. Flow in rotor/stator disc cavities and suppression of hot mainstream gas ingestion into those cavities are of particular interest in axial flow turbines. The geometry of such cavities and cavity rim sealing arrangements can vary widely with different turbine designs and for different application areas.

Figure 2. Section of the internal air system, showing the HP and IP sections for a three shaft engine (courtesy Rolls-Royce plc).

3. BAYLEY AND CONWAY'S PAPER

Bayley and Conway's [1] objectives were to provide experimental data for use in design calculations and to investigate the flow field in a narrow gap between rotating and stationary discs. In discussing the motivation for the work they referred to turbomachinery failures due to miscalculating the radial pressure distributions in disc cavities. They also initially intended to use measured velocity profiles to help develop Pohlhausen-type solutions of the equations of motion, but found that a theoretical solution to the problem was intractable. As will be discussed further below, the experimental data generated is still of interest. However, the most important aspect of this work is perhaps the recognition of the importance and difficulty of the problem and the need for further research.

Bayley continued the rotating disc cavity flow research theme following his work with Conway and his move (with the experimental rig) to the University of Sussex. This is illustrated in the paper by Bayley and Owen [3], which considers a shrouded disc system with a radial outflow, as opposed to the unshrouded system with inflow previously investigated. The correlation given by Bayley and Owen for the minimum flow rate required to prevent ingestion of external flow into the disc cavity has been widely used. The theme was continued by Owen and others with extension to corotating disc cavities and other configurations. Much of this work is described in the two part book by Owen and Rogers [4, 5]. Bayley and Conway's paper can therefore be said to have paved the way for and encouraged a strong UK research activity in this area. This continues to the present day with sustained internal air systems research at a number of UK Universities, often involving international collaboration.

Other early contributions to engineering research in this field, in addition to those cited by Bayley and Conway, include those of Dorfman [6] and Daily and Nece

[7]. Dorfman provided empirical correlations and theory which have been frequently used in engineering calculations. Daily and Nece reported measurements and correlations for rotor moment and flow in a sealed shrouded disc system which have also been used frequently in engineering calculations, and for evaluation of computational fluid dynamics (CFD) models. More recent publications are far too numerous to give here. Interest and activity in the area has increased significantly over the last two decades with, for example, the annual ASME Gas Turbine Conference running regular specialist internal air systems sessions.

The unshrouded, narrow gap flow problem considered by Bayley and Conway can now be recognised as potentially more challenging for predictive methods than shrouded disc cavity flow. The relatively very recent study by Cao et al [8] bears some resemblance to that of Bayley and Conway. Cao et al considered a narrow gap turbine rotor/stator cavity with weak net radial outflow. The investigation was motivated by concerns regarding engine performance and failure of earlier computational fluid dynamics (CFD) models to agree with the cavity pressure distributions measured on a test turbine using conventional pressure tapings. Unsteady CFD modelling indicated that, even with fully axisymmetric and steady geometry and boundary conditions, the flow is inherently unsteady. Large scale unsteady flow structures were predicted by the CFD, and confirmed by subsequent measurements with fast response pressure transducers. The cavity pressure distribution given by the unsteady CFD was in reasonable agreement with the pressure tapping measurements.

Bayley and Conway used wool tufts to examine the flow patterns at the periphery of their system, but do not report whether or not these indicated flow unsteadiness. They did not observe any unsteadiness in their cavity velocity measurements. It is of interest to compare Bayley and Conway's experiment with that of Cao et al. The main differences are that the more recent investigations are at slightly higher axial gap to radius ratio and have a highly swirling main annulus flow external to the cavity. Looking at the trends observed by Cao et al with decreasing gap ratio, it might be expected that, the flow in the outer part Bayley and Conway's cavity could become 3 dimensional and unsteady, at least for their higher gap ratios and lower flow rates. The negative radial gradient of swirl velocity at the cavity/external flow interface in Bayley and Conway's experiment might be expected to make an axisymmetric steady flow solution less stable than for Cao et al's conditions.

4. CURRENT CAPABILITY

A considerable body of research, since 1964, has been devoted to improving the predictive methods available for turbomachinery disc cavity flows. This has included very useful developments of momentum integral methods for shrouded disc cavities, but current attention focuses on CFD. As illustrated, for example, by Dixon et al. [9], use of CFD in industry for internal air flow prediction is now widespread and continues to expand. It is of interest to see how predictions from such methods compare with Bayley and Conway's measurements.

A preliminary CFD study has been conducted for Bayley and Conway's experimental configuration at their zero net throughflow condition. This was conducted by a research student using the commercial CFD code FLUENT [10]. Two

different representations of the experimental rig were used as shown in Figure 3. The first shows an axisymmetric model extending beyond the cavity to avoid the problematic imposition of boundary conditions in the important area of the cavity rim. The second model extends the first to a 90° sector with circumferential periodicity and treats the flow as unsteady. In all cases the k-ε turbulence model with FLUENT's "enhanced wall treatment" has been used. This blends a k-l model near the rotor and stator disc faces to the k-ε model in the freestream. Other aspects of the CFD method are similar to those used by Cao et al.

Figure 3. The CFD models considered (dimensions in inches).

Bayley and Conway's measurements and the present CFD predictions for the moment on the stator with no net throughflow are shown in Figure 4. Here the moment coefficient, defined as $C_m^* = 2M/\rho\Omega^2 r_o^5$, follows recent convention and differs from that used by Bayley and Owen. The figure plots $C_m^* Re^{0.2}$ against G , where Re and G are the rotational Reynolds number and gap ratio as defined by Bayley and Conway. It may first be noted that this choice of ordinate has some success in collapsing the experimental data. CFD results from the axisymmetric model cover a similar range of Reynolds numbers to those covered experimentally. The 3D CFD results are for $Re = 3 \times 10^6$.

At the lowest gap ratio considered by Bayley and Conway, $G \approx 0.004$, the axisymmetric, steady CFD model shown in Figure 3a gives reasonable agreement with the measurements. This condition may be associated with simple shear of the fluid between the rotating and stationary discs dominating. At higher values of G the CFD models give higher moments than measured. As shown by results for $G = 0.016$ and 0.031 , the three dimensional, unsteady CFD solutions produce very similar moments to the simpler model. Possible reasons for overprediction of C_m^* include errors or approximations in the boundary conditions, turbulence modelling assumptions and flow features that cannot be captured under the restrictions applied to the models. It is of interest to examine the unsteady CFD solution further.

Figure 4. Measured and calculated moment coefficients for the stator for $C_w=0$.

Figure 5 shows contours of the radial velocity component from the 3D, unsteady CFD model on the mid-axial plane in the disc cavity between the rotating and stationary discs. This clearly shows non-axisymmetry of the flow, with similar features to those identified in CFD and experiment by Cao et al. Further examination of the solution shows that the calculated flow is unsteady as well as three dimensional. While these predictions are plausible and consistent with previous work they should be treated with some caution. Considering the similarities between the unsteady flow features in figure 5 and those identified by Cao et al, it is surprising that the moment coefficients from the unsteady model agree so well with the axisymmetric CFD model. Cao et al found very significant effects of flow unsteadiness on mean flow features. This is clearly of interest for further investigation. These CFD solutions are likely to be sensitive to the angular extent of sector modelled and turbulence modelling assumptions. Numerical modelling assumptions such as mesh size, discretisation scheme and solution method might also affect the results. The influence of boundary conditions, rig geometry and both positive and negative net radial throughflow are also of practical interest.

The computing time required for the axisymmetric CFD was modest. Using 2 cores in a computing cluster the convergence could be achieved in around 2 hours. For the unsteady solutions, for which the mesh size was about 2.15 million cells, the solution was run for a simulation time corresponding to about 21 disc revolutions. This took about 180 hours running in parallel on 16 cores of the cluster which had fast interconnect.

Figure 5. Predicted radial velocity contours (m/s) on a plane midway between the rotating and stationary discs for $G=0.03$, $Re=3 \times 10^6$, $C_w=0$.

5. CONCLUSIONS

There have been considerable advances in understanding and predictive capability for turbomachinery disc cavity flows in the 5 decades since Bayley and Conway's research. Modern CFD methods represent a considerable advance in predictive capability for such flows. However, the particular flow problem Bayley and Conway considered still offers some challenges today, and cannot be said to be fully understood. Their experimental data is therefore still of interest to current researchers. Current CFD methods are able to give reasonable estimates of parameters such as moment coefficients and insight into the complex, unsteady flow that occurs.

Awareness and demonstration of the natural unsteadiness of interaction between disc cavity and main gas path flows has grown in recent years, and is receiving significant attention in the research community. Understanding and predictive capability for such effects can be expected to improve with consequent improvements in design methods. This is just one of many aspects of turbomachinery design where research in Mechanical Engineering Science continues to improve Engineering practice.

ACKNOWLEDGEMENTS

The CFD results presented here were kindly provided by Umesh Javiya who holds an EPSRC/Rolls-Royce Dorothy Hodgkin Postgraduate Award at the University of Surrey.

REFERENCES

1. Bayley, F.J. and Conway, L., 1964. Fluid friction and leakage between a stationary and a rotating disc. J. Mech Eng. Sci., vol. 6. pp.164-172.

2. Ruffles, P.C., 2000. The future of aircraft propulsion. Proc. Instn Mech. Engrs Vol. 214 Part C, J. Mechanical Engineering Science, pp. 289-305.
3. Bayley and Owen, 1970. The fluid dynamics of a shrouded disc system with a radial outflow of coolant. J. Eng. Power (1970), pp. 335–341.
4. Owen, J.M, and Rogers, R.H., 1989. Flow and Heat Transfer in Rotating Disc Systems: Rotor-Stator Systems (Vol. 1), Research Studies Press, Taunton (John Wiley, New York).
5. Owen, J.M, and Rogers, R.H., 1995. Flow and Heat Transfer in Rotating-Disc Systems : Rotating Cavities (Vol. 2). Research Studies Press, Taunton (John Wiley, New York).
6. Dorfman, L. A., 1963. Hydrodynamic Resistance and the Heat Loss of Rotating Solids. Oliver and Boyd, London.
7. Daily J.W. and Nece, R., 1960. Chamber dimension effects on induced flow and frictional resistance of enclosed rotating discs. ASME Journal of Basic Engineering, 82, 217-232.
8. Cao, C., Chew, J.W., Millington, P.R. and Hogg, S., 2004. Interaction of rim seal and annulus flows in an axial flow turbine. ASME J. Eng. for Gas Turbines and Power, pp. 126, 786-793.
9. Dixon, J.A, Verdicchio, J.A., Benito, D., Karl, A. and Tham, K.M., 2004. Recent developments in gas turbine component temperature prediction methods, using computational fluid dynamics and optimization tools, in conjunction with more conventional finite element analysis techniques. Proc. Instn Mech. Engrs Vol. 218 Part A: J. Power and Energy, pp. 241-155.
10. Fluent Inc. 2005. See URL <http://www.fluent.com>.