Int. Symp. on Heat Transfer in Gas Turbine Systems
9 – 14 August, 2009, Antalya, Turkey

HEAT TRANSFER TESTING IN ENGINE TURBINE COOLING SYSTEM DEVELOPMENT

Peter Ireland
Turbines SCU, Rolls-Royce plc
Moor Lane, PO box 32, Derby. DE24 8BJ, UK
peter.ireland@rolls-royce.com

ABSTRACT The speed with which a new aircraft engine is developed prevents extensive experimental testing of key turbine components. For this reason, engineers rely increasingly on computer simulations to develop and perfect components before committing to final designs. This paper addresses the use of heat transfer experiments in the context of turbine cooling system development. It reviews how experiments can be used to support both research activity and the Engine Development Programme (EDP). The paper describes the state of the art in Perspex model test technology and the introduction of rapid prototyping (RP). The paper also explains how judicious use of RP test data can be used in cooling system development to arrive at optimal systems.

PERSPEX MODEL TESTING

Perspex model tests represent the state of the art for heat transfer data accuracy and resolution for blade cooling. The technique was originally developed in the 1980s and has been employed with success by many heat transfer research groups. It is also now used by most of the gas turbine companies.

Fully Featured Geometry. The technique is very well suited to measuring detailed HTC distributions in complex geometries- Weigand et al. [2001] and one recent development is application to realistic geometries including most of the cooling passage features. Measurement of local coolant temperature variation through the transient experiment allows highly accurate HTC data to be obtained on all internal surfaces of a turbine blade or vane cooling passage, Poser et al. [2005]. Advanced data analysis and image processing strategies can be employed to allow for local variations in viewing and lighting conditions, Poser et al, [2007].

There are many reports in the literature that show that the effect of rotation on blade cooling heat transfer coefficients can be significant. The strength of the effect depends on whether secondary flows caused by the Coriolis acceleration and/or buoyancy forces, indicated by high rotation number and buoyancy numbers respectively, are significant relative to the secondary flows for a stationary passage. The considerable amount of work involved in an experimental campaign to study rotating effects on HTC means that it is usually not possible to use rotating heat transfer measurements directly in an EDP. For this reason, rotating heat transfer experiments are normally used in research activity. A recently launched European FP7 supported programme, ERICKA[1], will study the effect of rotation on cooling systems with engine realistic geometries. The project will use rotating facilities at Rolls-Royce, Figure 1, and at ONERA to quantify the effect of rotation on fully featured radial flow and impingement systems.

[1] Engine Representative Internal Cooling Knowledge and Applications.

Figure 1 The Rotating Heat Transfer Rig used by Rolls-Royce to study the effect of rotation on blade cooling flows.

Application to EDP Perspex model heat transfer experiments are used both in research activity and as part of an EDP. In the case of the latter tests, the high resolution data can be used to check assumptions used in the cooling design. Typically, the heat transfer data from a static model is compared to the results from CFD. For a blade cooling design, the CFD is then used to predict conditions under rotation as discussed below. A recent example of the comparison of detailed CFD to static rig test data was reported by Jackson et al [2009]. The computational results were registered to pressure drop and heat transfer measurements in a non-rotating passage. This work formed part of an EPD and enabled assumptions used in the thermal model of the blade to be evaluated before the engine was operated.

RAPID PROTOTYPING

Rapid prototyping of components is increasingly used as a means of assessing the performance of new cooling system configurations. The technique can not provide the high resolution HTC data that Perspex model testing can. But rapid prototype model tests can provide crucial flow data at an early stage of a new design. Once the Computer-Aided Design (CAD) geometry of a component is defined, a model can be produced, literally, overnight so that the shape of the part can be quickly reviewed in the design office. The cost of such a plastic component depends on its manufacture time but for an aircraft engine vane or aerofoil the cost is typically only a few hundred pounds. Thomas and Hodson [2008] present a recent review of the application of rapid prototyping in wind-tunnel testing. They compared the geometric accuracy, porosity and practicality of test pieces manufactured by four different rapid prototyping machines.

Geometry of the SL model

Staircase roughness All rapid prototyping methods result in a surface roughness which can affect the flow under study. A typical modern SL machine uses a step size in the range 50-100μm. The sides of the perimeter of each layer are essentially vertical which means that models with surfaces that are not vertical take on a staircase form. These steps inevitably create a periodic surface roughness. The size of the step can be minimised to reduce the roughness, but this is at expense of increased build time and cost. The height and pitch of the roughness elements depend on the angle at which the surface is built relative to a horizontal platform. For small components and high speed flows, the roughness height can result in a surface that is not hydraulically smooth. In cases where the flow swept surface is on the outside of the component, the surface roughness can be reduced by gently polishing with sand paper. However, in the case of a cooling passage, it is not possible to access the internal surface.

The potential effect of this roughness on flow through passages was considered in detail by Mittal et al. [2005]. Figure 2 shows four circular pipe tests pieces grown at different angles to study the effect of staircase roughness on friction factor. Each piece is shown with inlet and outlet flanges. Four, 100mm long pieces had internal diameters of 5 mm and were grown at 0°, 20°, 45°, and 90° to the platform. Two smaller pipes with diameter 3mm and one square section pipe with hydraulic diameter 5mm were also tested.

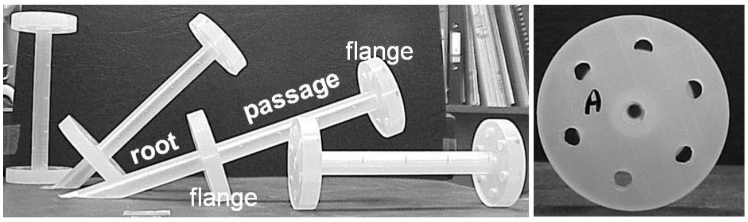

Figure 2 Circular pipes grown at different angles to platform, Mittal et al. [2005].

The SL produces an internal surface roughness which varies around the perimeter. The shape of each circular section pipe surface is analogous to the shape formed from stacking a set of washers with elliptic holes. The step size varies around the perimeter of the passage. The steps form grooves that are inclined to the flow direction at the build angle.

To achieve a range of engine representative Reynolds numbers in the laboratory the experiments were run inside of a pressurised chamber. The chamber was pressurised to 6.89 bar (100 psi), and the test pipe vented through a calibrated choked orifice. As the pressure drops in the chamber, the density and temperature of the air also decrease and the mass-flow through the pipe reduces. A set of low range, piezo-resistive differential pressure transducers were used to measure the static pressure difference between pairs of adjacent tappings. The transducers were placed inside the pressure chamber in order to minimise common-mode error and to enable the use of sensitive transducers.

Figure 3 shows the measured friction factor as a function of Reynolds number for all of the pipes tested. The top left hand chart shows the data for a smooth brass pipe which was used to confirm the accuracy of the experimental method. The SL pipe data confirm that the surface roughness has increased the friction factor. The green dashed line was calculated from the Colebrook [1939] equation where the area average equivalent sand grain roughness of the SL process has been estimated from a Talysurf measurement of the greatest roughness line multiplied by $2/\pi$ and by the cosine of the build angle. This data can be used to evaluate the impact of SL roughness on model test results.

CONCLUSIONS

The paper addresses the means by which flow and heat transfer experiments can be used effectively in a modern EDP.

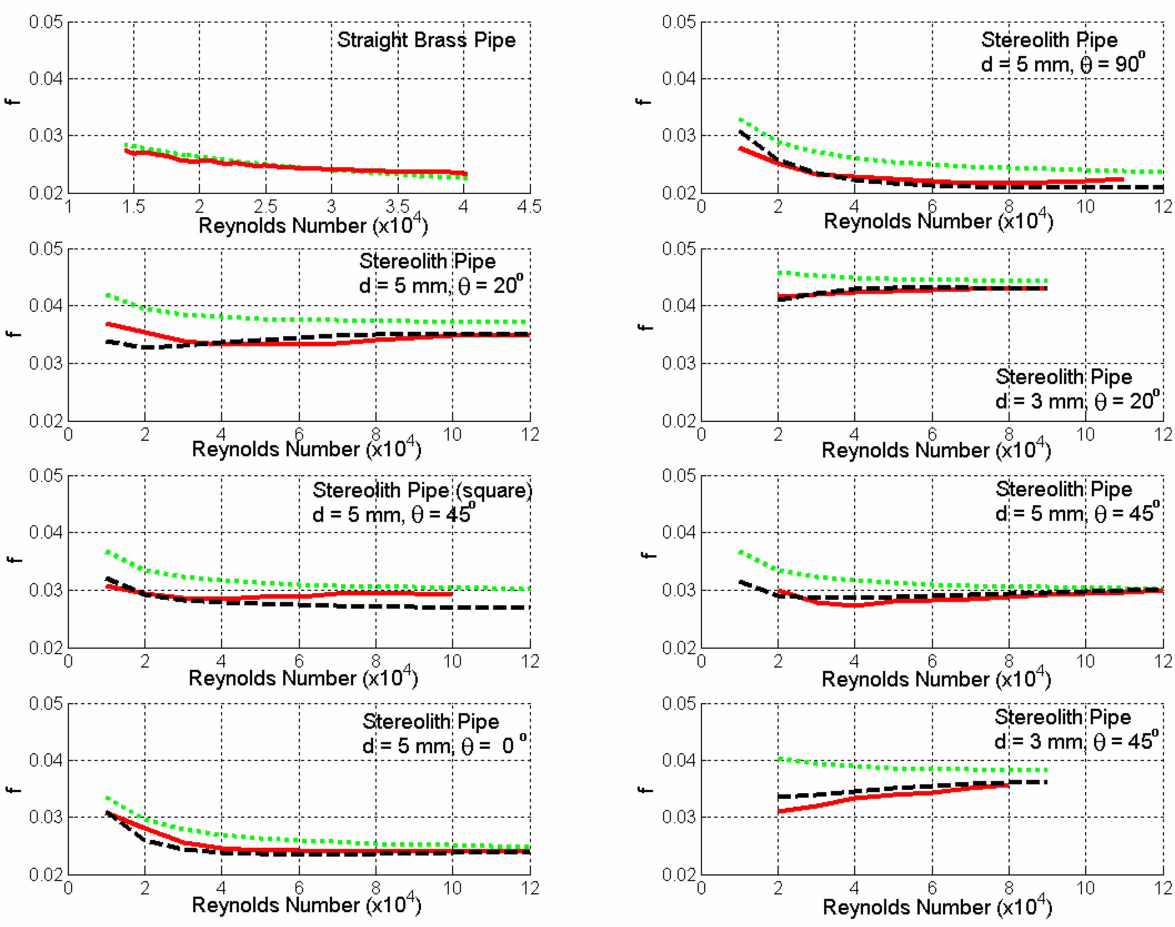

Figure 3 Friction factor results: experimental (red solid), law of the wall prediction (black dashed), and Colebrook [1939] prediction (green dotted), Mittal et al. [2005].

REFERENCES

Colebrook, CF. [1939], Turbulent Flow in Pipes, with particular reference to the Transition Region between the Smooth and Rough Pipe Laws." J. Inst. Civ. Eng. Lond. Vol. 11, pp 133-156.

Jackson, D., Ireland, P.T. and Cheong, B. [2009], Combined experimental and CFD study of a HP blade multi-pass cooling system, paper GT2009-60070,ASME Turbo Expo Orlando.

Mittal, V., Ireland, P.T. and Mitchell, M., [2005] Predicting the friction factor and Nusselt number in stereolith. pipes, *9th UK National Heat Transfer Conference, Manchester.*

Poser, R.; von Wolfersdorf, J.; Lutum, E. [2007], Advanced Evaluation of Transient Heat Transfer Experiments Using Thermochromic Liquid Crystals, *Proc. IMechE Vol. 221 Part A: J. Power and Energy, , 793-801*.

Poser, R., von Wolfersdorf, J. and Semmler, K. [2005], Transient Heat Transfer Experiments in Complex Passages , *ASME Heat Transfer Conference, San Francisco, HT 2005-7226*.

Thomas, R. and Hodson, H.P. [2008], Rapid prototyping for wind tunnel testing and instrumentation, *XIX Biannual symposium on measuring techniques in turbomachinery, April 7-8*; edited by G. Paniagua ISBN 978-2-930389-32-X.

Weigand, B., Semmler, K., and von Wolfersdorf, J. [2001], Heat Transfer Technology for Internal Passages of Air-Cooled Blades for Heavy-Duty Gas Turbines , in *Heat Transfer in Gas Turbine Systems*, Annals of the New York Academy of Sciences, Vol. 934, pp. 179-193, 2001.