Abstract of paper submitted to Turbine 2000, ICHMT, Izmir, Turkey

Detailed Heat/Mass Transfer Distributions in a Rotating Two-Pass Coolant Channel with Engine-Near Cross-Section and Smooth Walls

L. Rathjen and D. K. Hennecke
Darmstadt University of Technology
Gas Turbines and Flight Propulsion
Petersenstrasse 30
D-64287 Darmstadt
Germany
Fax.: +49-6151-164159
e-mail: rathjen@gfa.tu-darmstadt.de

S. Bock[1] and R. Kleinstück
Motoren- und Turbinen- Union München
Dachauer Straße 665
D-80995 München
Germany
Fax.: +49-1489-97878
e-mail: stephan.bock@muc.mtu.dasa.de

A main objective in turbine blade cooling design is to achieve optimum heat transfer coefficients while minimizing the coolant flow rate. The influence of rotation in coolant channels is of particular importance for heat transfer research in turbine blades. Secondary flow caused by Coriolis forces increases or decreases heat transfer, depending on flow and direction of rotation. Experimental investigations are necessary to get a better understanding of the complex three-dimensional flow structure and its influence on the heat transfer and pressure loss and to validate numerical calculations.
The available data in literature, however, concentrate on test cases with idealised geometries. Therefore, this paper will show results obtained from mass transfer measurements on an engine-near cross-section which is shown in Figure 1. The influence of Reynolds number, Rotational number, engine-near geometry and turning vane on the mass transfer where investigated.

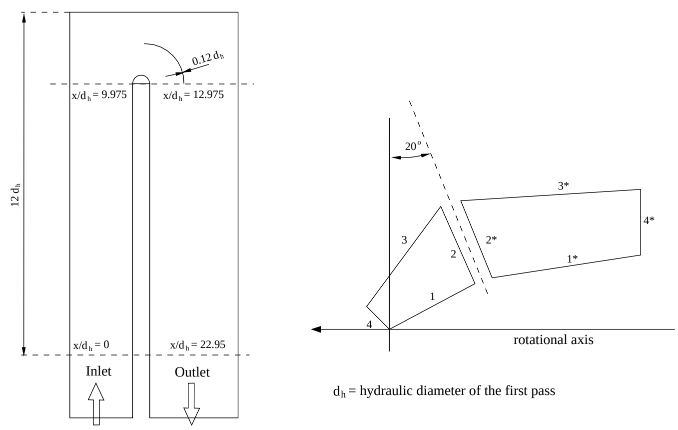

Figure 1: Test model geometry

The model consists of two channels, a radially outward directed one and a second channel

[1]Ph.D. student from the German Aerospace Center (DLR) Cologne, currently located at MTU Munich

with radially inward flow direction connected by a 180° U-bend with turning vane. The length of the test section was 12 hydraulic diameters. This means that the flow is not fully developed at the entry of the bend. The hydraulic diameter of the second pass is 1.5 times bigger than the hydraulic diameter of the first pass. The turning vane mounted in the U-bend has a thickness of $0.12d_h$. Reynolds number and Rotational number are calculated with the hydraulic diameter of the first pass.
The investigation of the influence of Reynolds number and Rotational number were carried out in combinations shown in Table 1.

Re\Ro	0	0.02	0.05	0.1	0.2
10000	×	×	×	×	×
25000	×	×	×	×	×
50000	×	×	×	×	

Table 1: Combinations of Re and Ro numbers

The local mass transfer was investigated using the naphthalene sublimation technique and applying the analogy between heat and mass transfer. The data obtained within these investigations will be presented as Sherwood number ratios. The local mass transfer coefficients were related to those obtained by the Dittus-Boelter-correlation for fully-developed tube flow. The mass transfer coefficients at the 90° turning vane in the bend and at the tip plate were not investigated.
The results, visible in Figure 2 for a Reynolds number of 10000, show very interesting effects on the Sherwood number ratio distribution caused by the three-dimensional flow field. Especially the distribution in the bend and after the bend region shows strong gradients caused by the 180° bend, where several separation zones exist, and the turning of the flow following the airfoil shape. Also the mixing of the two air mass flows after the turning vane leads locally to a strong increase of Sherwood numbers due to the high shear stresses existing in this region. These kinds of contour plots in the paper will show the local effect of rotation.

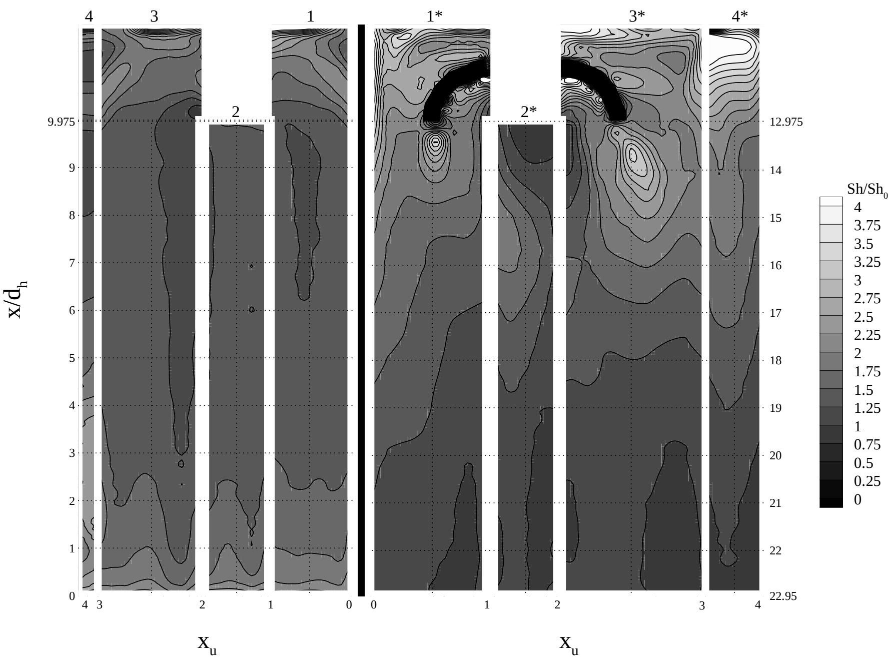

Figure 2: Local Sherwood number ratio distribution at Re=10000 and Ro=0

Figure 3 shows the behaviour of the spanwise averaged Sherwood number ratios over the streamwise coordinate x/d_h.
Again the strong influence of the bend on the mass transfer becomes clear. In the second

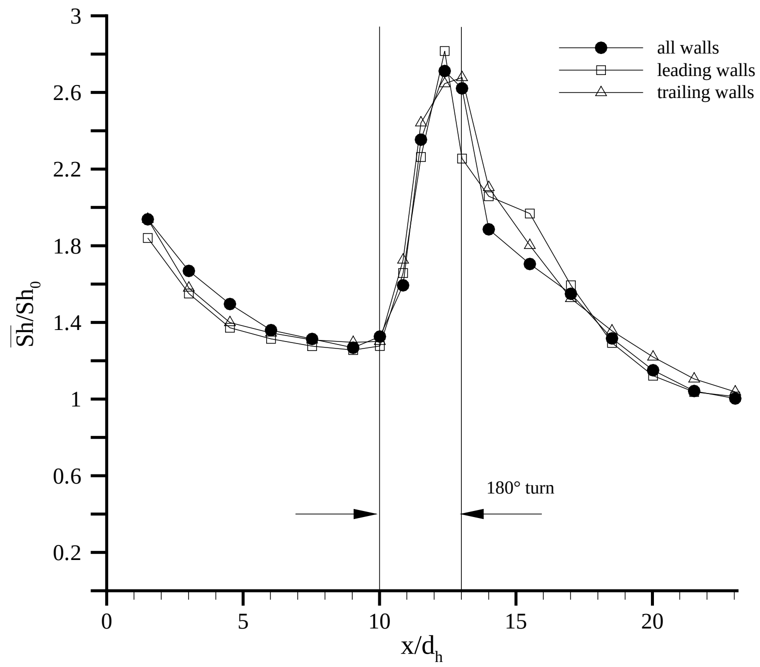

Figure 3: Spanwise averaged Sherwood number ratios along x/d_h

pass there is locally a difference between leading and trailing wall values due to the additional turning of the flow following the airfoil shape. In the paper, pictures of this kind will demonstrate the influence of Coriolis force on the leading and trailing wall values.
These results give a detailed picture about the mass transfer conditions affected by the flow field in such an engine-near coolant channel. The available data base is useful for validation of heat/mass transfer calculations as well.
In addition to the experiments, CFD calculations have been performed to gain a better understanding for the underlying flow phenomena and to validate the CFD software used.

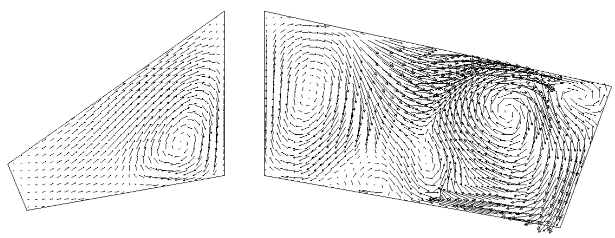

Figure 4: Secondary flow field shown in cross section close to the bend for Re=50000 and Ro=0

In a second part, this paper will show the results of these CFD calculations. The complicated secondary flow field that develops from the shape of the test section and the overlaying Coriolis forces is displayed in pictures like Figure 4. Speed vectors in different planes, as well as Nu/Nu_0 plots corresponding to the experiments are displayed and compared to the experimental findings. The calculated Nu/Nu_0 distributions are then averaged along streamwise locations in order to be compared to the experimental results as presented in Figure 3.

Altogether, an interesting comparison between CFD calculations and extensive experimental research is presented in this paper. Local Sherwood and Nusselt number ratios are displayed and the flow field is analyzed to gain new insight into the physics involved in internal cooling of turbine blades.