Numerical Study of Turbulent Heat Transfer in a Rotating Two-Pass Cooling Channel

Michael Göhring, Tobias Krille, Jens von Wolfersdorf

Institute of Aerospace Thermodynamics, University of Stuttgart, 70569 Stuttgart, Germany

michael.goehring@itlr.uni-stuttgart.de

Long Abstract

Introduction

Internal cooling channels of turbine blades are essential components of gas turbines. By increasing the turbine inlet temperature, the thermal efficiency can be augmented significantly. However, the melting temperatures of the blade materials are already exceeded by far and therefore some part of the compressed air is used to cool the blades. For an efficient cooling system the processes and dependencies of the specific cooling method have to be understood in detail.

Feasible internal cooling techniques are for example impingement cooling, swirl chambers and ribbed multi-pass channels. A summary of these is given by Han et al. [1]. Regarding the gas turbine rotor blade, the effects of rotation are inevitable and have to be accounted for. One of the first works on rotating cooling channels has been carried out by Johnson et al. [2]. They considered the effects of rotation in terms of Coriolis and centrifugal forces and determined segmental area averaged Nusselt numbers. Thorough parameter studies were done by varying the Reynolds number, the rotation number and the centrifugal buoyancy, following the dimensions of real engine applications. This is why a lot of other works have been compared to this case, not only experimental, but also numerical studies [3]. Hence, the case by Johnson et al. qualifies well for comparison.

Complex shaped multi-pass channels with ribs are state of the art. Yet, the flow phenomena are not understood completely. For the validation of a numerical setup a two-pass cooling channel based on the geometry of Johnson et al. [2] is incorporated. Both the experimental results from Johnson et al. and the numerical results from Bonhoff et al. [3] are employed for detailed comparisons. The aim of the presented work is to establish a numerical setup that is able to appropriatly predict the heat transfer in cooling channels. In the future this setup will be used to simulate more complex channel geometries.

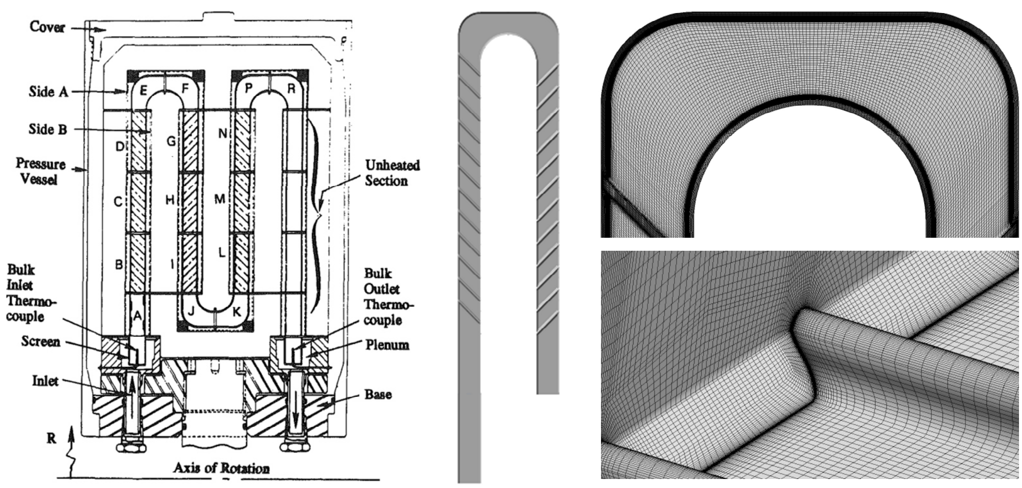

Figure 1. Four-pass cooling channel by Johnson et al. [2] and modeled two-pass channel with two numerical grid detail views

Numerical Setup

For the numerical studies only the first two of four legs of the reference cooling channel have been modeled. The block-structured numerical grid has been generated with ANSYS ICEM CFD 15.0. For the evaluation of the heat transfer the dimensionless wall distance has to be restricted. The grid points closest to the wall are ensured to have a value of about 1. To assess the grid independence, a grid convergence index (GCI) study has been carried out.

The computational fluid dynamics are performed with the finite-volume solver ANSYS CFX 15.0, which is a fully implicit, coupled multigrid solver. For the stationary solution of the problem the Reynolds averaged Navier-Stokes equations are utilised, the closure problem is undertaken by two different turbulence models. On the one hand, the widely used and robust Shear Stress Transport (SST)-model by Menter is applied. In a lot of previous works good agreement has been reached with this model when compared to experimental heat transfer investigations. However, one weakness of two-eqaution models is the insensitivity to streamline curvature and system rotation. Therefore, a curvature correction is used for this model. On the other hand, an explicit algebraic Reynolds stress model is employed, which is a further development of the two-equation models. Altough it is not as accurate as differential Reynolds stress models (RSM), it incorporates the anisotropy of the Reynolds stresses and captures the effects of rotation. Besides, the computational costs are considerably less compared to RSM.

The spatial discretization is handled with the default method in CFX called High Resolution scheme and blends automatically between first and quasi-second order. When using the SST-model, convergence cannot completely be reached. That is why the discretization scheme is shifted towards first order. By this convergence can be improved.

Results

The numerically received results are compared to the reference data in terms of segmental area averaged Nusselt numbers for suction and pressure side. The numerical results are additionally evaluated in a more in-depth way. First, the averaging procedure is refined, so that a more detailed trend is specifiable. Moreover, contour plots of the local Nusselt number distribution are presented. They provide a more detailed impression of the heat transfer task which can be connected to the calculated flow field.

Selected setups of Reynolds number, rotation number and rotational buoyancy are simulated and compared to the reference data. One important characteristic of the rotation induced Coriolis force is the impact on the mean flow, which is pushed towards the trailing side in the radial outward cooling channel. By this, the heat transfer coefficient experiences a significant augmentation. This behavior is even more prominent for higher rotation numbers. The influence of the turbulence modeling is finally discussed, concluding with a detailed analysis of the interaction of fluid flow and heat transfer.

References

[1] J.-C. Han. Fundamental Gas Turbine Heat Transfer. *Journal of Thermal Science and Engineering Applications*, 5:021007-1–021007-15, 2013.

[2] B. V. Johnson, J. H. Wagner, G. D. Steuber and F. C. Yeh. Heat Transfer in Rotating Serpentine Passages With Trips Skewed to the Flow. *ASME Journal of Turbomachinery*, 116:113-123, 1994.

[3] B. Bonhoff, U. Tomm, B. V. Johnson and I. Jennions. Heat Transfer Predictions for Rotating U-Shaped Coolant Channels With Skewed Ribs and With Smooth Walls. *International Gas Turbine Aeroengine Congress & Exhibition*, Orlando, Florida, 1997.