11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20–25, 2014, Barcelona, Spain

THERMAL FLUID-STRUCTURE INTERACTION BASED OPTIMIZATION OF SECONDARY AIR FLOWS IN ROTOR STATOR CAVITIES OF AIRCRAFT TURBINES

Hannes Lück[1], Michael Schäfer[1], Heinz-Peter Schiffer[2]

[1] Institute of Numerical Methods in Mechanical Engineering
[2] Department of Gas Turbines and Aerospace Propulsion
Technische Universität Darmstadt, Darmstadt, Hessen 64287
[1]Email: lueck@gsc.tu-darmstadt.de

Key words: *Thermal Coupling, CFD, FE, Optimization, Aicraft Turbine.*

The cooling of turbine components like blades and discs is essential since the melting temperatures of the materials used are several hundred degrees below the operating turbine inlet temperature. Cooling air with higher total pressure is required to seal rotor stator cavities and blade surfaces to withstand the hot gases exceeding the combustion chamber. Hereby, up to 20% of the main annulus flow is bled of the compressor stages and led to the turbine stages around the combustion stage. That quantity of compressed main annulus gas does not enter the combustion chamber and cannot contribute to the overall engine cycle. Therefore, the optimization of these so called secondary air flows is considered to be one of the most promising research activities in gas turbines to enhance the global engine efficiency. The accurate evaluation of the disc metal temperature plays the significant role in predicting component life and the corresponding minimum demand of coolant.

Conventional methods conduct finite element analyses and apply semi-empirical correlations of thermal boundary conditions or CFD solutions at the outer disc wall. An enhancement to this pure thermal coupling is the incorporation of thermo-mechanical deformations in a fully coupled Thermal Fluid-Structure Interaction approach (TFSI). The software package ANSYS offers two different types of couplings between its CFD solver CFX and FE solver Mechanical. The two types can be separated into an implicit coupling by utilization of the ANSYS MFX interface and an explicit coupling by use of manual scripts. Hereby, the explicit manual coupling has proven to be the more efficient approach since it splits up the thermal coupling of the fluid heat flux and the solid temperatures in a separate CHT (Conjugate Heat Transfer) solver from the calculation of the corresponding deformation in the FE solver Mechanical.

The validation test case for this work is the Turbine Stator Well test facility at the University of Sussex [1]. The two-stage axial turbine rig was validated by several industrial and academic partners [2]. A surrogate 2D model and a 3D sector model are presented.

The converged TFSI solution is achieved at a hot running geometry that reveals an interstage seal clearance increased by 33 % compared to the cold build state. It could be shown that it is crucial to capture the hot running clearances since the interstage seal flow directly entails the mass flow rate at the rotor stator rim next to the main annulus path. Hereby, the increased interstage seal flow under hot running condition causes a transition of coolant egress to a slight hot gas ingestion at the rim decreasing the cooling efficiency inside the rotor stator cavity.

The manual TFSI approach is coupled with the MATLAB *fmincon* algorithm, using e.g. the gradient-based sequential quadratic programming (SQP) method. The optimization scenario aims for an optimized cooling efficiency and minimized stresses in the rotating assembly at a specified slight egress of coolant at the rim. The chosen design variables are e.g. the axial coolant entry location, the entry angle and the hot running interstage seal clearance. Exemplary, figure 2 compares the default and optimal setting and figure 3 shows the corresponding history of the objective function. The objective function J is defined as the mass flow, normed by an optimal mass flow at the rim, plus the temperature at the rotor wall, normed by the potential temperature drop. In the optimum, the rim mass flow equals the specified coolant egress of 7 g/s. Additionally, the averaged rotor wall temperature could be decreased from 110 °C to 89 °C. Examples with more design variables and a comparison between different optimization methods are presented.

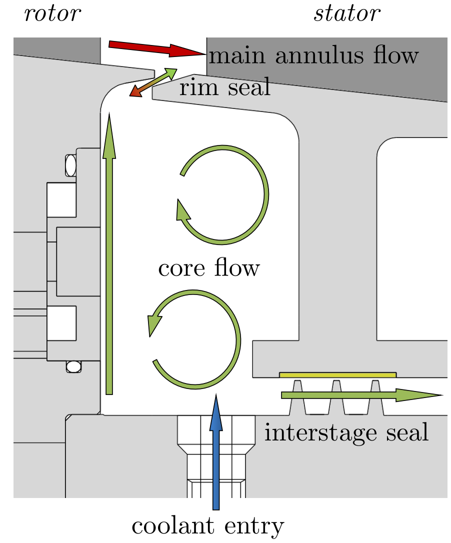

Figure 1: Flow structure in rotor stator cavity of axial two-stage turbine rig

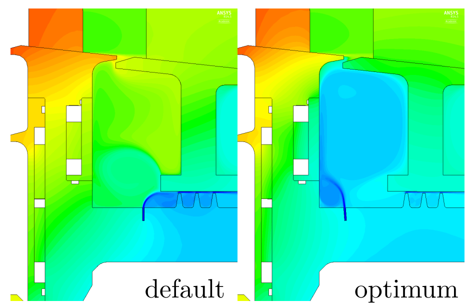

Figure 2: Temperature field at default and optimal configuration

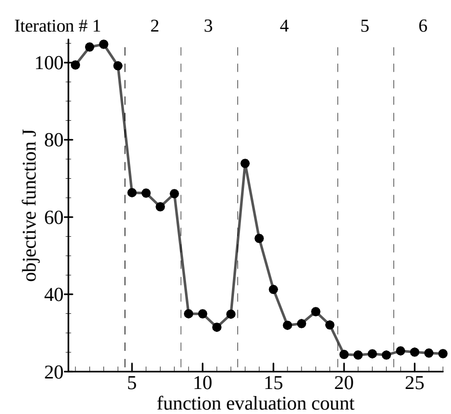

Figure 3: Objective function over function evaluation count

REFERENCES

[1] Coren, D. D., Atkins, N. R., Turner, J. R., Eastwood, D. , Childs, P. R. N., Dixon, J. A., and Scanlon, T. S.. An advanced multi-configuration stator well cooling test facility. *In Proceedings of ASME Turbo Expo 2010*, GT2010-23450.

[2] Dixon, J. A., Guijarro Valencia, A., Coren, D. D., Eastwood, D., and Long, C. A.. Main annulus gas path interactions - turbine stator well heat transfer *In Proceedings of ASME Turbo Expo 2012*, GT2012-68588.