Politechnika Śląska w Gliwicach

Wydział Inżynierii Środowiska i Energetyki

Instytut Maszyn i Urządzeń Energetycznych

NUMERYCZNA OCENA ZJAWISK CIEPLNO-PRZEPŁYWOWYCH W WYBRANYCH WĘZŁACH STOPNIA TURBINY GAZOWEJ

Praca doktorska

Mgr inż. Krzysztof Bochon

Promotor:

Dr hab. inż. Włodzimierz Wróblewski prof. nzw. w Pol. Śl.

Gliwice, 2012

Numerical investigation of fluid flow and heat transfer phenomena in selected parts of the gas turbine stage

PhD thesis – abstract

Author: Krzysztof Bochon, M.Sc. Eng.

Supervisor: Prof. Włodzimierz Wróblewski

The work includes issues related to the fluid flow and heat transfer in selected fragments of the flow path of the gas turbine. Its main objective was to optimize the geometrical configuration of the tip seal with honeycomb land of the counter-rotating LP turbine of the open rotor aero-engine and the Conjugate Heat Transfer (CHT) analysis of the seal area and rotating cavity above the seal.

In order to perform efficient optimization, the geometrical model of the seal has been simplified on the basis of calculations, allowing to assess their impact on the result quality. Much attention was paid to the mesh preparation. In the following part a definition of boundary conditions, physical model and the parameters taken into account during optimization were discussed.

Optimization was based on CFD calculations performed with the use of the commercial code ANSYS-CFX. The aim of optimization was to reduce the leakage through the seal. Optimization was performed using the commercial tools available in the ANSYS Workbench environment, using the Goal Driven Optimization. The results were compared with calculations made with the use of the in-house optimization code which is based on the evolutionary algorithm. The best geometrical configuration was obtained with the use of the Goal Driven Optimization. Geometrical configurations of the seal obtained with the use of various optimisation methods showed a similar tendencies of change. Optimization was supplemented by sensitivity analysis.

Best solution was verified on the extended geometrical model and dense mesh. Verification confirmed good quality of the results obtained during the optimization and significant leakage reduction, amounting 17.26%. Using verification calculations, the extensive analyses of the seal were carried out, in which such parameters as total and static pressure, flow velocity, static entropy, and the turbulence kinetic energy were taken into account. This allowed to assess the impact of geometry on the distributions of these parameters and the location of the main places of losses generation.

The seal analysis was extended by the calculations for changed pressure at the outlet of the seal and for the increased three times pressure level throughout the entire computational domain. Pressure changes had no significant effect on the character and structure of the flow. In addition, calculations were performed for decreased sealing clearance over the fins, which showed very strong influence of this parameter on leakage reduction and on the limiting of the mixing effect of the leakage with the main flow.

Conjugate Heat Transfer analysis were preceded by the calculations for rotating cavities presented in the literature and for the target cavity, separated from the seal. Their goal was to verify the computational model, the overall assessment of the phenomena and the choice of computational domain size. The CHT analysis of the entire seal area was made for two different thermal conductivity coefficients of metal, what caused different distribution of the temperature and different heat exchange conditions. In both cases, very low values of relative air velocity in the chamber and the lack of a clear convective motions were noticed. The change of thermal conductivity caused a slight difference in the Nusselt number.

The research was made possible by financial support of the European Union (EU) within the project ACP7-GA-2008-211861 “DREAM” Vali**d**ation of **r**adical **e**ngine **a**rchitecture syste**m**s.