Flexible HP-Turbines II

Project Manager
Marius Linne

Researchers
Dominik Ade

Principal Investigator
Prof. Dr.-Ing. Heinz-Peter Schiffer

Project Term
2023 - 2024

Clusters
Lichtenberg II Cluster Darmstadt

Software
ANSYS

Institute
Gasturbinen, Luft- und Raumfahrtantriebe (GLR)

University
Technische Universität Darmstadt

Introduction

The fluid temperatures in the high-pressure turbine of a commercial jet engine or industrial gas turbine by far exceed the melting point of the materials used. This severely limits the ability to obtain experimental data from testing. As a result, the design process of modern high-pressure turbines, which relies heavily on the numerical prediction capabilities of 3D CFD simulations, is also burdened with high numerical uncertainties. The main objective of the Flexible HP-Turbines project is the identification of numerical uncertainties and the investigation of aerothermal interaction mechanisms both within the outer annulus region of the Large Scale Turbine Rig. Therefore, two casing side cooling treatments – one upstream and one downstream of the first vane row – need to be designed using 3D CFD simulations, which are then built, tested, and compared with the numerical results.

Methods

The commercial CFD code Ansys CFX is used for all CFD simulations. Within CFX, the three-dimensional, coupled, pressure-based flow solver is used to solve the steady and unsteady compressible Reynolds-Averaged Navier-Stokes equations. The solver is based on the finite volume method and uses a high-resolution (quasi-2nd order) spatial discretization scheme. The closure problem of the Reynolds-Averaged Navier-Stokes equations is treated by Menter’s SST model, which combines the advantages of the k-ϵ model in free streams and the k-ω model near walls and additionally considers the effect of the transport of the turbulent shear stress. The interface between rotating and stationary components is treated by using a mixing plane or by performing transient simulations.

Results

To identify numerical uncertainties in the outer annulus of a high-pressure turbine, two casing side cooling treatments – one upstream and one downstream of the first vane row – were designed for the Large Scale Turbine Rig using steady-state Reynolds-Averaged Navier-Stokes approaches. Due to an excessive numerical study involving the effects of mesh resolution and overall convergence criteria on the aerodynamic performance of a representative high-pressure turbine geometry could be worked out. It could be shown that especially the type of time- and turbulence modelling has significant effects on the results. To fully resolve effects of stator-rotor-interaction, full unsteady RANS (URANS) simulations are required. Steady RANS or harmonic approaches show high deviations in the prediction of secondary flow structures (vortices) as well as overall efficiency in contrast to the URANS approach. Unfortunately, a fully converged URANS simulation is highly resource intensive, costing around 100 times more than a RANS simulation.

Discussion

While the aerodynamic impact of the upstream cooling ejection on the downstream rotor tip flow field appears to be comparably small, this region is also known to be prone to high numerical uncertainties. For future research, the consideration of more advanced turbulence modelling approaches – potentially scale-resolving models – might therefore be required and benchmarked against the familiar RANS approaches.

Last Update: 2025-01-16 12:21