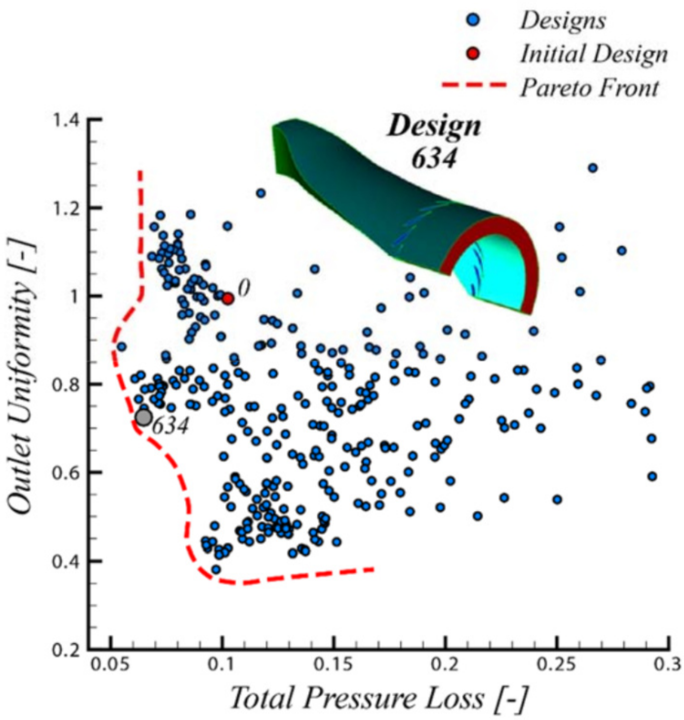
TREVAP (Technologien für Revolutionäre Arbeitsprozesse)



Hessisches Kompetenzzentrum für Hochleistungsrechnen

Project Manager
Faramarz Bakhtiari

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

Project Term
2018 - 2019

Clusters
Lichtenberg Cluster Darmstadt

Software
ANSYS

Additional Software
TRACE

Institute
Gasturbinen, Luft- und Raumfahrtantriebe (GLR)

University
Technische Universität Darmstadt

Introduction

The fact that the efficiency of the currently used aircraft engine components is getting really close to its limits, and the achievement of aviation's long term targets to reduce emissions cannot be attained by using traditional technologies. To obtain a higher efficiency class and therefore to reduce emissions, it is necessary to apply innovative and revolutionary technologies.
The proposed technical solutions are based on cycle processes which are characterized by pressure gain combustion (PGC) methods. This type of combustion technology contributes to a significant thermal efficiency enhancement and low NOx-emission rates at the same time. The simplest form of PGC is an isochoric combustion. The composite cycle engine (CCE) and the wave rotor constant volume combustion (WRCVC) are most popular concepts that are based on PCG. Compared with a conventional engine cycle, these two concepts offer an improvement in total efficiency and therefore an immense potential for fuel consumption. The development of such innovative engine concepts requires a holistic examination of their integrability. The combustor turbine interaction (CTI) is an important factor in this field. Novel combustor technologies affect the turbine design substantially. Studies on spatially inhomogeneous inflow have shown that the impact on the turbine efficiency and blade cooling is significant.
The CCE technology mainly impacts the downstream situated turbine in the form of temporally variable inflow conditions, like periodic pressure fluctuations, that lead to a transient aerothermal load in the high pressure turbine (HPT). Previous studies have shown a negative impact of transient inflow conditions on turbine's efficiency. It is expected that this also affect the cooling concepts in a conventional HPT.
The concept of the WRCVC also changes the interaction of the combustor and turbine. The WRCWC leads to a spatially inhomogeneous flow. In addition, a connecting element is required at the geometric interface. The challenge is to design a transition duct that transfers the flow from a small segment of a circle to a 180 degree circle segment. In order to be able to design a transition duct which on the one hand reduces inhomogeneity and on the other hand requires the shortest possible construction space, a 3D optimisation is applied. Depending on the transition duct, the HPT is subjected to strongly inhomogeneous flow, which also leads to efficiency deficits.
Due to the highly unsteady inflow and therefore unsteady mechanism within the flow domain, usual steady state simulations are usual not expedient. To receive accurate results, greater computing power is needed. Through the use of the high-performance computer, a fully transient simulation (URANS) with high temporally resolution could be realized and thereby meaningful results received.

Methods

The aerothermal evaluation of the transiently loaded HPT is done by using a finite volume discretization of the Reynolds-Averaged Navier-Stokes equations (RANS).

All simulations are performed using the commercial codes ANSYS CFX® and Fluent®. Both solvers use a pressure-based approach. However, they differ in the pressure-velocity coupling. CFX® is a so-called coupled solver, in which all momentum equations and the pressure equation are solved in a matrix. Fluent® in contrast, is an uncoupled solver which means that the SIMPLE scheme has been used for pressure-velocity coupling. Both codes solve the compressible RANS equations based on the finite volume discretization method. The high resolution (quasi-2nd order) spatial discretization scheme is applied for the CFX® computations, while the 2nd order scheme is used in Fluent®. For the temporal discretization, the 2nd order Backward Euler method is used.
Two turbulence models were used in the investigations. For global studies, the two-equation shear-stress turbulence model (SST k-ω) developed by Menter is used. A ω-based Reynolds Stress turbulence Model (BSL-RSM) is used for a close look at the vortex structures of the cooling jet. The compressible fluid is modelled as an ideal gas with a temperature dependent heat capacity. In some case, a model for species transport is activated. This permits the use of two gases in the same fluid volume. The unsteady computations are based on a dual-time stepping scheme with 15 inner iterations.

Results

Part of the project was to quantify the underlying effects of transient inflow on the cooling mechanism in the HPT, such as film cooling. The numerical investigation of various test cases have shown that periodic pressure fluctuations, depending on magnitude and frequency, can significantly affect the film cooling characteristics. The film cooling effectiveness decreases due to flow mechanical effects. Under certain conditions, sufficient cooling is no longer possible. With the knowledge gained from the various test cases and further investigations on engine conditions, a realistic estimation should indicate to what extent conventional cooling concepts can satisfactorily perform in the case of highly unsteady inflow.
In the case of the WRCVC, a transition duct could be created by analytical methods. This geometry was used as initial geometry for a 3D optimization. With the help of the optimization, a geometry could be designed, which shows a clear improvement compared to the initial geometry. Despite a significant improvement with regard to flow inhomogeneity, the investigations at the HPT have shown that efficiency losses are to be expected.

Discussion

All gathered results so far are numerical. In the case of interaction between film cooling and pressure fluctuation, we will obtain experimental data for one of the test cases. These data will be used to estimate how good the numerical predictions are in the case of transient inflow. To reduce the numerical model error, numerically more complex methods are applied (LES). Furthermore, the findings from the test cases have to be transferred to engine geometries in order to close the gap between basic research and applied research.

Figures

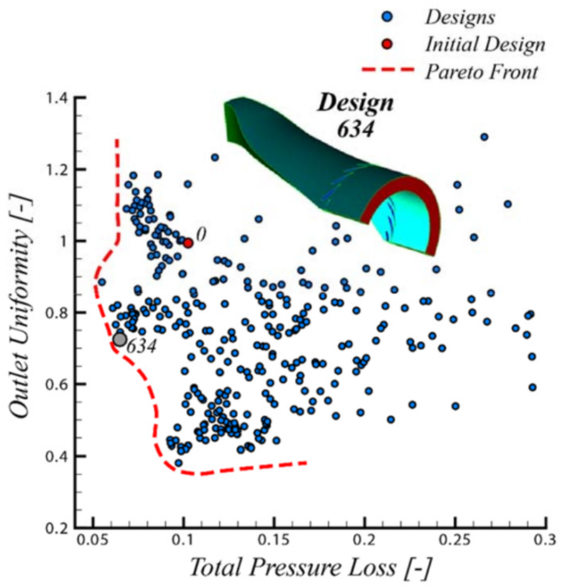


Figure 1: Transition Duct Optimisation Database.

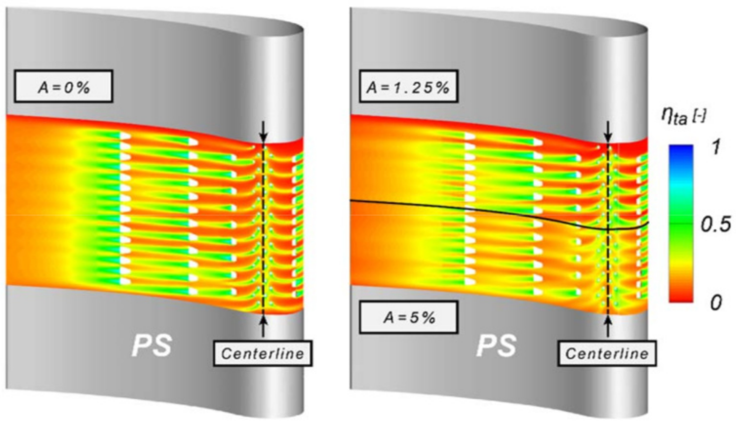


Figure 2: EPFL Cascade – Comparison of Film Cooling Effectiveness at different Pressure Amplitudes.

Reference

Kaiser, S.; Donnerhack, S.; Lundbladh, A.; Seitz, A.: A Composite Cycle Engine Concept with Hecto-Pressure Ratio, 51st AIAA/SAE/ASEE Joint Propulsion Conference, Orlando, FL, 2015 https://www.bauhaus-luftfahrt.net/fileadmin/user_upload/2015_AIAA_Propulsion_and_Energy_Kaiser_A_Composite_Cycle_Engine_Concept_with_Hecto-Pressure_Ratio_O.pdf

Kaiser, S.; Seitz, A.; Vratny, P.; Hornung, M.:Unified Thermodynamic Evaluation of Radical Aero Engine Cycles, ASME Turbo Expo, Seoul, 2016. https://doi.org/10.1115/GT2016-56313

Bakhtiari, F.; Schiffer, H.P.: Numerical Approach to the Modelling of Transient Interaction of Prospective Combustor Concepts and Conventional High Pressure Turbines, Journal of Propulsion and Power Research, 2019 https://doi.org/10.1016/j.jppr.2019.01.008

F. Bakhtiari et al., "Numerical Investigation of Film Cooling Effects under Transient Inflow", ISABE-2019-24319, Canberra, Australia, 2019

Akbari, P.; Nalim, M. R.: Review of Recent Developments in Wave Rotor Combustion Technology, Journal of Propulsion and Power, 2009 https://doi.org/10.2514/1.34081

Wilcox, D.: Reassessment of the Scale-Determining Equations for Advanced Turbulence Models, AIAA Journal, 1988 https://doi.org/10.2514/3.10041

Giles, M. B.: Non-Reflecting Boundary Conditions for the Euler Equations, Tech. Rep. CFDL-TR-88-1, CFD Lab., MIT, 1988 https://doi.org/10.2514/3.10521

Acton, E.; Cargill, M.: Non-Reflecting Boundary Conditions for Computations of Unsteady Turbomachinery Flow, In 4th Int. Symp. Unsteady Aerodynamics and Aeroelasticity of Turbomachines and Propellers, 1988

Yang, H.; Nürnberger, D.; Weber, A.: A conservative zonal approach with application to unsteady turbomachinery flows. In DGLR Jahrestagung, 2002 https://elib.dlr.de/49430/1/YANG__DGLR2002-073_Volltext.pdf

Last Update: 2022-06-14 21:22