Design Solutions for the Coupling of High-Pressure Turbine Vanes to Pressure-Gain Combustors

Panagiotis Gallis

Department of Energy

Politecnico di Torino

Doctoral Program in Energy Engineering (37$^{\text{th}}$ cycle)

Abstract

Nowadays, the increasing energy demand and the expansion of the aviation industry leads the research community to reassess the principles and the operation of the gas turbines. The traditional quasi-isobaric deflagrative burner induces considerable total pressure losses to the system. A promising approach to address this issue involves the use of Pressure Gain Combustion (PGC) cycles, which are typically implemented in isochoric deflagrative or detonative systems. These unconventional machines rise the cycle stagnation conditions leading to the increment of the theoretical thermal efficiency of the gas turbine. Nonetheless, they are accompanied with strong spatio-temporal fluctuating outflow. Therefore, it is essential to investigate the impact of these pulsations on the performance of the subsequent turbine stage and to develop innovative solutions for their effective integration. Under that prism, the current thesis focuses on the interactions between the PGC and the High-Pressure Turbine (HPT) vanes. The present research activity considers the Constant-Volume Combustor (CVC) and the Rotating Detonation Engine (RDE). The study is primarily based on unsteady Computational Fluid Dynamics (CFD) simulations, supplemented by experimental activities using advanced measurement techniques.

After a detailed literature review on the PGC principles, technologies and preceding attempts for the integration with an axial HPT stage, the CVC case is presented. First of all, a 1D model of the entire existing test rig of the CVC is developed aiming at accurately specifying the CVC chamber outlet conditions. Later, the resulting total pressure and temperature are introduced to the existing exhaust system connected with a converging-diverging nozzle. A 3D URANS simulation evaluates the performance of the outtake system. Afterwards, a conceptual design of a spacer, a transition duct and a HPT vane is proposed. A numerical parametric analysis of 81 different transition pieces is conducted, while the optimum case in terms of generated total pressure losses and attenuation is elected. Then, a 3D URANS simulation of the new CVC exhaust system is performed, revealing its superior performance compared to the existing system's numerical analysis.

Afterwards, the transition duct optimum profile is then modified to serve the needs of the experimental test rig and manufactured. In particular, the configuration of spacer, transition duct and a converging-diverging nozzle is proposed for reasons of simplicity. Fast response pressure sensors are introduced in the combustion chamber and the transition duct. In addition, the exhaust domain is employed with windows allowing for Particle Image Velocimetry (PIV) analysis. Two different experimental campaigns are conducted. First, non-reactive tests take place for two different test rig pressure ratio uncovering the pulsating air behaviour inside of the transition duct. Later, reactive experiments with high frequency PIV system take place, while the operating frequency of the machine varies in the range of $15 - 40\ Hz$. This experimental campaign provides for the first time the detailed and accurate description of the CVC outflow, while the transition duct is able to subtract the spatio-temporal variation of the flow field.

Finally, the focus of the thesis is on the RDE. For proper starting of HPT vane passage, inlet diffusive endwalls should be implemented. Although, significant enlarged secondary flows are observed. If the RDE oscillating inflow is considered, these vortical structures will affect the performance of the downstream blade row. Thus, a novel flow control systems is proposed. A series of cooling holes at the hub and shroud, located upstream of the vane's leading edge, are introduced to suppress the motion of the tip and hub passage vortices. 3D URANS calculations proved that the flow control system is able to restrict the motion of these vortical structures, increase the vane efficiency and better guide the flow. Nonetheless, the size of the secondary flows is still large. Thus, the solution of the non-axisymmetric endwalls is considered. A CFD shape optimization aimed at minimizing the kinetic loss coefficient takes place. A Design of Experiments (DOE) of 1000 cases is created, while 3D RANS calculations used to assess them. Genetic Aggregation method creates a response surface, while a gradient-based optimization algorithm provides the optimum solution. The optimization result reveals a significant reduction of the secondary flows promoting the idea of non-axisymmetric endwalls for a transonic vane downstream of an RDE.