Turbine Working with RDC Exhaust Flow

Aerodynamic Performance Evaluation of Axial Turbine Integrated into a Rotating Detonation Engine

M. Asli, P. Stathopoulos*, Institut of Fluid Dynamics and Technical Acoustic, Chair of Unsteady Thermodynamics in Gas Turbine Processes, TU Berlin*

In Short

- Energy Extraction form Rotating Detonation Engine
- Unsteady Aerodynamic Performance of Axial Turbine Stage
- Total Pressure Loss Measurement
- URANS simulation

Detonative Pressure Gain Combustion (PGC), whether in the form of Rotating Detonation combustion (RDC) or Pulse Detonation combustion (PDC), has the potential to increase the propulsion efficiency of aero-engines and the thermal efficiency of stationary gas turbines. Theoretically, implementing PGC in a gas turbine will increase the cycle performance, due to the lower entropy generation in the combustion process. However, pressure gain combustion results in higher turbine inlet temperatures, while at the same time the exhaust flow of detonative PGC chambers is characterized by strong pressure, temperature and velocity fluctuations. One of the main challenges in the practical implementation of PGC into gas turbines is the lack of designs for turbomachines that can cope efficiently with the PGC exhaust gas. Although still a topic of active research, it is generally accepted that existing conventional turbine expanders interacting with the exhaust of pressure gain combustors will have lower isentropic efficiency, compared to their design operation[1]. If the turbine isentropic efficiency drops below a certain values, the benefits of applying pressure gain combustion will be lost, due to the inability of the turbine to harvest the additional exergy present in the combustor exhaust gas. The benefits of RDE compared to PDE motivate a transition from turbine-PDE to RDE-turbine integration investigations. RDE has much higher detonation wave frequency that is generally more favorable from a turbomachinery standpoint. Figure 1 shows an RDE and a self-sustained detonation wave traveling around the annulus.

Recently, an instrumented guide vane has been set up to characterize RDC flow by Bach et al.[3] at TU Berlin experimentally. The aim was to address the effect of RDC outlet restriction on combustion through the combined effects of elevated initial reactant pressure and reflected shocks interacting with the detonation wave. They also performed pressure measurements upstream and downstream of the vanes, which revealed the average pressure loss through the vanes. The compact geometry, rapid and high temperature detonation phenomena limit the capability of experimental measurement and visualization. Furthermore, the presence of measuring probes mounted on the blades can cause uncertainties in the acquired experimental data. Therefore, numerical analysis can be a promising tool to understand the flow field around the vanes. We have studied the same geometry numerically to look into the details of the vanes behavior and compared the results with the experiments [4]. Additionally, the RDC configurations with different blade setting angle and different geometrical parameters have been modeled numerically in our recent works [5]. Figure 2 indicates flow field inside an stationary vanes mounted at the RDC downstream having 20 degree staggered angle. The objective was to evaluate how different vane types and different blade solidities affect the flow in the turbine blade row. According to the Figure 3, as blade setting angle increases, pressure distribution becomes more unstable in more than 50% of the time period. In decreasing part of the total pressure trend, three local peaks were generated in non-zero blade setting angles. A pressure pulse which goes through the staggered vanes become more unsteady by increasing blade setting angle, while the amplitude does not change noticeably. This generated unsteadiness would be unfavorable for turbine rotor blades, regardless of frequency of the fluctuation and its impact on turbomachinery

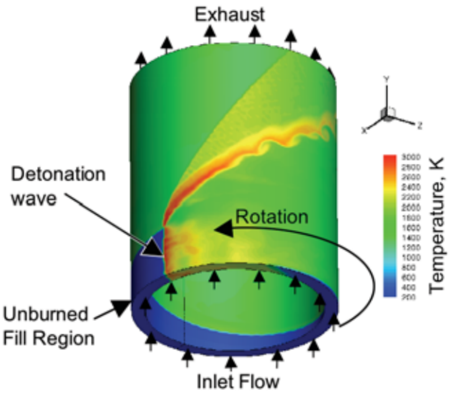

Figure 1: *RDE temperature flow field[2].*

performance.

Following the above mentioned research works and based on the knowledge gained by characterizing the RDC exhaust flow by both experimental and numerical works, a set of axial turbine stages have been designed. In the current work, the turbine stages including rotor and stator, are modeled at the RDC downstream. As the flow is considered non-reactive, the 3D computational domain will be the exhaust duct of RDC together with the turbine stage. The working fluid is considered to be a mixture of nitrogen (N_2) and water vapor (H_2O) that are products of Hydrogen-air combustion. The mixture is assumed to be a perfect gas thermally and calorically. Inlet boundary of the domain is applied as time and location dependent variables based on the experimental RDC exhaust flow measurements. Unsteady Reynolds-averaged Navier Stokes (URANS) simulations are carried out using ANSYS-CFX solver, which provides suitable environment for turbomachinery aerodynamic simulation. The main objective here is to explore total pressure loss mechanisms associated with turbine stage design parameters through numerical simulation of different design configurations in the highly unsteady flow field of RDC. Furthermore, performance of turbine stages will be evaluated to give feedback to the one dimensional design process and further adaption. According to the scope of simulation which is unsteady with a frequency of ~5-7kHz, high speed computational resources are required to achieve accurate results in reasonable time.

WWW

`http://www.fd.tu-berlin.de`

More Information

[1] D. Van Zante, E. Envia, M.G. Turner, The *ISABE-2007-1260* , E-16138 (2007).

[2] C.A. Nordeen, D. Schwer, F. Schauer, J. Hoke, B. Cetegen, T. Barber *49th AIAA Aerospace Sciences Meeting* , AIAA 2011-803 (2019). doi: 10.2514/6.2011-803

[3] E. Bach, M.D. Bohon, O.C. Paschereit, P. Stathopoulos, *AIAA Scitech 2019 Forum* , (2019). doi:10.2514/6.2019-0476

[4] M. Asli, C. Cuciumita, P. Stathopoulos, C.O. Paschereit *Proceedings of the ASME Turbo Expo 2019* , (2019). doi: 10.1115/GT2019-91263

[5] M. Asli, P. Stathopoulos, C.O. Paschereit *Proceedings of the ASME Turbo Expo 2020, (accepted)* , (2020).

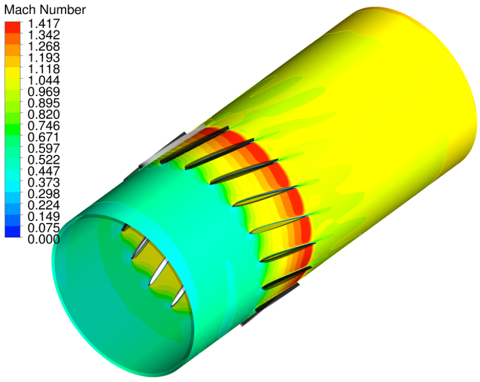

Figure 2: *Simulation of stationary blade row mounted at RDE downstream[4]*

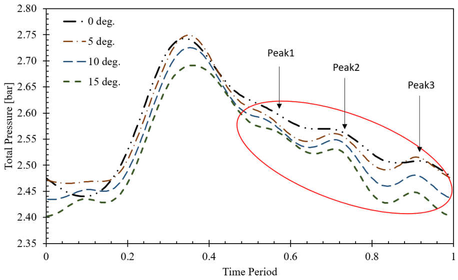

Figure 3: *Total pressure distribution at 1 blade chord downstream of the vanes[4]*