[Extended Abstract]

Development of a Novel Test Rig to Investigate Explosion Safety in Gas Turbine Enclosures

Jan Kowalski, David Engelmann, Chair of Thermal Turbomachinery, Ruhr-Universität Bochum, Bochum, Germany
Martin Lauer, Ronald Mailach, Chair of Turbomachinery and Flight Propulsion, Technische Universität Dresden, Dresden, Germany
Michele Cagna, MAN Diesel & Turbo SE, Oberhausen, Germany

Introduction

Due to the still growing demand for more flexible power plant solutions industry-type gas turbines of intermediate power class (up to 25 MW) have moved into the focus of research efforts over the past years. Since the dimensions of these machines often allow being set up on a single frame they are usually designed as so-called gas turbine packages including the electric generator on the same frame in case of a power generation setup. This specific design is commonly accompanied by the whole machine being engulfed in an acoustic enclosure. These so-called compartments reduce the emitted noise level of the gas turbine and protect it from environmental influences when operated as an outdoor installation. The enclosure separates the gas turbine from the surroundings, therefore the whole compartment has to be actively ventilated for two main reasons: Firstly, the ventilation flow removes heat losses generated from the Brayton cycle itself and thus keeps machine components at acceptable temperature levels. Secondly, the ventilation flow has to dilute and remove clouds of fuel from the compartment in case of gas leakage due to e.g. seal failure in the fuel system. Though the latter case is unlikely during engine operation it must still be considered. The British Health and Safety Executive (HSE) [1] reported 134 natural gas leakages in the UK over a period of 13 years leading to a total of 19 incidents of fuel ignitions in gas turbine enclosures, thus underlining the importance of careful ventilation flow design to ensure explosion safety.

To investigate in detail the mixing of a fuel leakage jet within a state-of-the-art MGT-6100 gas turbine package from MAN Diesel & Turbo SE, Petrick [2] conducted CFD investigations using ANSYS CFX with $k-\omega$ SST turbulence model on the whole compartment, taking into account thermal convection as well as leakages at different positions. The numerical grid used for that survey was chosen based on an intensive grid resolution and grid type study comparing hexahedrons, tetrahedrons and tetrahedrons combined with prism elements. The chosen mesh consists of about 15.5 million tetrahedron elements including prism layers near bounding walls. The numerical model is illustrated in figure 1. Petrick concluded the current package design to meet modern explosion safety requirements but underlines the importance of applying appropriate boundary conditions for heat transfer. Therefore a research project was launched aimed at creating validation data for CFD investigations of the aforementioned gas turbine package, which will be discussed in this paper. The focus, however, will be on the design process and numerical results.
Project goals and approach

Development work was carried out with three distinct long-term goals to accomplish: Firstly, validation data for CFD investigations should be created for cold, heated, as well as leakage configuration. Secondly, a detailed experimental and numerical assessment of heat transfer and buoyancy effects shall be conducted in different operation conditions. Lastly, the optimization potential of reducing or redirecting the ventilation flow will be explored.

To fulfill the defined goals a scaled test rig of the gas turbine enclosure was designed, which can be seen in figure 2. The test rig features all relevant components of the machine including the gas turbine, gear, generator, as well as complex pipework of the oil and fuel system in reasonably simplified degree of detail. The machine itself is inactive and mainly functions as an obstacle for the ventilation flow. Initially the test rig will be set up in cold configuration but the later addition of heating elements and gas leakage (e.g. CO₂ as model gas) is considered. The design process has been carried out taking into account aspects of similarity and transferability of the scaled results to the real size machine, as will be discussed in detail in this paper. Vahidi et al. [3] conducted similar research on a scaled gas turbine package. They concluded that for hydraulic and thermal analogy a scaled test rig has to be operated with the same Reynolds (Re), Nusselt (Nu) and Grashof (Gr) number as the actual engine [3]. While Re- and Nu-similarity are technically feasible, deviations of the Gr number are unavoidable due to its cubic dependency of the length scale and the high temperature level of the gas turbine walls.

To assess these deviations the design process was accompanied by numerical investigations based on the work of Petrick [2]. During the procedure different values for the scaling factor k (model scale 1 : k) were examined analytically. From this set two values for k where considered as being practicable for the test rig and therefore were investigated in detail and compared to the real-scale results using CFD methods. These simulations were carried out in Ma- and Re-similarity operation condition achieved by varying the volume flow rate entering the package and the flow passing through the actively ventilated generator respectively. The investigations concluded that the test rig should be downscaled to 1 : 3 and that the ventilation system has to be able to accomplish both Ma- and Re-similarity. To achieve Re-similarity during testing a total of four dual-stage counter-rotating fans had to be installed generating a volume flow of up to $Q_{encl} = 16,000$ m³/h. Like in the actual turbine package these blowers are set up in parallel-path air flow arrangement at the air inlet as well as at the air outlet side. In addition to k the influence of upper temperature limits (staff safety) as well as the generator cooling flow were examined. These studies showed e.g. that the generator flow has a strong influence on the overall flow field. Therefore the scaled test rig features eight additional high speed fans operating at 11,000 RPM to include and to control the flow through the generator.

Since the velocity field within the compartment is of main interest, PIV measurements were chosen as the primary measurement technique. Therefore all side and roof walls except the ones which are flange-mounted to the package fans were constructed from transparent material. Due to the addition of heating elements in the following phase of the project perspex was discarded and borosilicate glass was chosen for its higher temperature resistance as well as for its high optical quality. In addition to PIV several local pressure and temperature probes were installed to monitor the operation point, as well as CTA probes for turbulence intensity measurements. The experimental setup will be described in detail in this paper.

During the course of the test rig development the design of the actual gas turbine package was optimized further, leading to slight differences in component arrangements. To accomplish maximum accuracy of the project results a new numerical flow model was created to be used for validation purposes. A grid sensitivity analysis was carried out as well as investigations on the flow field in cold configuration. Each will be discussed in this paper. The results were used to identify relevant measurement planes for PIV measurements. Selected experimental results from the test rig in cold configuration will be presented as well.
Figure 1. Numerical base model [2] used and modified during the design process

Figure 2. Final construction of the downscaled (1:3) package test rig

References

