INVESTIGATION OF THE VENTILATION FLOW IN A GAS TURBINE PACKAGE ENCLOSURE

J. Kowalski 1* - F. di Mare 1 - S. Theis 2 - A. Wiedermann 2, M. Lange 3, R. Mailach 3

1 Chair of Thermal Turbomachines and Aeroengines, Ruhr-Universität Bochum, Germany
2 MAN Energy Solutions SE, Oberhausen, Germany
3 Chair of Turbomachinery and Flight Propulsion, Technische Universität Dresden, Germany

* Corresponding Author: jan.kowalski@rub.de

ABSTRACT
The ventilation flow of a gas turbine enclosure was investigated both experimentally and numerically in a scaled test rig. Its gas turbine components were inactive and mainly functioned as flow obstructions. Measurements were used to provide realistic boundary conditions and verification data for URANS CFD investigations. In the test rig the active generator cooling flow was created by several fans, modelled by momentum sources in the CFD. Two different modelling approaches were compared to PIV data. The modelling of the ventilation flow was found to impact flow regions nearby the generator but also far downstream with one method showing better agreement with the experiment. Examining the flow predicted by the chosen CFD model, no relevant backflow from the engine to the upstream gear section could be observed. Therefore, the investigation of stagnation regions was focused on the engine section to identify worst case leakage configurations for future investigations.

KEYWORDS
CFD, URANS, Gas Turbine, Enclosure Ventilation, Explosion Safety

NOMENCLATURE
Abbreviations
CFD Computational Fluid Dynamics
CPT Combined pressure and temperature
EP Evaluation plane
GGI General Grid Interface
IGCF Internal generator cooling flow
MEVF Main enclosure ventilation flow
PIV Particle Image Velocimetry
SST Shear Stress Transport model
TFT Through flow time
TRM Test rig CFD model
URANS Unsteady Reynolds averaged Navier Stokes equations
Latin symbols
c [m/s] Velocity magnitude
u [m/s] Velocity component in x-direction
y⁺ [-] Dimensionless wall distance
Subscripts
xz, yz [m/s] Projected velocity magnitude

INTRODUCTION
Installing gas turbines in an acoustic enclosure is common for engines of intermediate power class, i.e. up to 25 MW. These enclosures reduce the noise emitted by the engine and protect it from weather conditions in case of an outdoor installation but also require special consideration regarding explosion safety. Even though no explosive atmosphere exists during regular engine
operation, fuel leakages due to system failures might occur and must be taken into account. Usually, gas turbine enclosures are actively ventilated to dilute and to extract possible leakages. This dilution ventilation is regarded by Ivings et al. (2004) as the most common safety measure to prevent explosions in gas turbine enclosures. Nevertheless, studies by the British Health and Safety Executive (2008) revealed the occurrence of 134 gas leakages in enclosures within the UK alone during a period of 13 years, resulting in a total of 19 incidents of actual fuel ignition. Therefore, careful ventilation flow design is crucial for safe engine operation. Usually, individual investigation of each engine’s ventilation flow configuration is required. Two methods are commonly adopted. The first approach is to model a fuel jet leakage analytically and to evaluate the size of the fuel/air mixture cloud above a certain concentration level for a given ventilation flow rate, as described by Ivings et al. (2008). The second approach is to model the whole enclosure including an occurring leakage jet using Computational Fluid Dynamics (CFD) methods. While the first method represents a simplified approach, it was found to be rather conservative by Gant and Ivings (2005). However, even though CFD considers very complex phenomena, including their respective interaction mechanisms, choosing suitable model parameters as well as the application of appropriate boundary condition is still crucial to acquire reliable flow field data. Hence, detailed measurements are needed to verify the chosen CFD model.

Experimental investigations of an enclosure ventilation flow were conducted by Vahidi et al. (2006). They developed an enclosure test rig with a model scale of 1:6, which they operated with heated engine walls. Their test rig featured a very simple geometry and piping systems were omitted completely. Though local pitot probe measurements were conducted, the focus of their work was on the leakage concentration of CO₂ as a replacement for natural gas and information given about the flow field mainly results from CFD data. Based on these studies they carried out numerical simulations and derived a design procedure for package ventilation systems (Bagheri and Vahidi 2006). Further investigations were conducted by Ivings et al. (2008). They developed a test enclosure with multiple in- and outlets as well as a large obstruction in which they carried out different leakage scenarios using a tracer gas with the same molecular mass and density as CH₄. The experimental results were then used to verify their CFD results. As in the aforementioned work, the test configuration was rather simple and apart from concentration data, no detailed flow field information is provided.

To create detailed ventilation flow field data, a cooperation project among MAN Energy Solutions SE, the Technische Universität Dresden and the Ruhr-Universität Bochum was started whereby a scaled test rig simulating a gas turbine package was developed and investigated experimentally. The experimental data was then used to set up and to verify a CFD model of the enclosure ventilation flow, which shall be discussed in detail in this paper.

EXPERIMENTAL INVESTIGATION

The test rig was designed based on a comprehensive similarity study which was conducted utilizing analytical as well as CFD methods. The results were documented in detail by Kowalski et al. (2017). Based on these studies, a model scale of 1:3 was chosen along with appropriate operation conditions for the investigations. The test rig and the experimental setup shall be discussed in the following sections.

Experimental Facility

The test rig features the main engine components such as the gas turbine, gear, generator as well as extensive pipework of the fuel and oil system, as shown in figure 1a. The gas turbine
components are inactive and mainly function as flow obstructions. However, the engine can be outfitted with heating elements to include heat transfer and leakage gas can be injected via the fuel system piping, as illustrated in figure 1b. For the results discussed in this paper, the test rig was operated in Ma-similarity conditions, adiabatic wall conditions and without leakage. The whole power package is placed in an enclosure, which features several easily removable window segments for Particle Image Velocimetry (PIV) measurements as well as maintenance work. To generate the main enclosure ventilation flow (MEVF), the test rig is equipped with four dual stage counter-rotating fans. As in the actual engine, they are set up in a parallel-path air flow arrangement with two fans at the inlet and two at the outlet duct, which allows the investigation of realistic fan failure scenarios. The generator of the actual engine is actively ventilated for cooling purpose. This internal generator cooling flow (IGCF) must be distinguished from the MEVF. During the design process the influence of the IGCF on the MEVF became evident. Hence, the test rig’s generator is also actively ventilated by a total of eight small-scale high-performance fans. As can be seen in figure 1a, the enclosure inlet fan ducts are aligned directly towards the generator flow inlet. This results in the IGCF consisting partially of the MEVF. The fraction of the MEVF, which does not enter the generator, escapes either towards the top or the sides of the generator, where it interacts with the IGCF when reentering the enclosure (see figure 1b).

![Figure 1](image_url)

**Figure 1:** Downscaled (1:3) package enclosure test rig developed by Kowalski et al. (2017)

**Measurement Setup**

The test rig features four combined total pressure - total temperature (CPT) probes, one at each enclosure in- and outlet duct. For temperature measurements the pitot tube is combined with a Type-T thermocouple (probe setup 1). The CPT probes can be traversed throughout the whole duct cross section at two circumferential positions. The total pressure and temperature profiles at the inlet ducts were used to measure the MEVF rate. To obtain velocity data from the pitot measurements, each in- and outlet duct also features four pressure taps for static pressure measurements (probe setup 1). The IGCF can be monitored with a pitot rake consisting of 10 individual pitot tubes mounted in front of the generator inlet (probe setup 2), which can be traversed laterally. Pressure signals are transferred to two pressure transducer arrays (Scanivalve ZOC17IP/8PX) and are recorded by a NI PXI data acquisition system, placed in a controlled temperature room.

Flow field measurements were conducted using a Dantec 2D2C PIV system featuring a Litron Bernoulli laser with a repetition rate of 15 Hz and an energy output of 200 mJ per pulse.
at 532 nm and an Andor HiSense Zyla 5.5 sCMOS camera with 5 megapixels resolution and 16 bit dynamic range. Data was processed with the Dantec Dynamics Studio software. For seeding purpose, a DEHS aerosol was inserted into the MEVF at each inlet duct downstream of the pitot probe via two small tubes with several drilled holes. To align the PIV light sheet optic and camera, modular profile frames were attached on the package enclosure’s frame system.

**Evaluation Procedure**

For measuring the MEVF rate, CPT probes were placed at 13 radial positions for each circumferential alignment. The probe positions were chosen so that each was placed on the mean radius of equally sized subsections in each duct’s cross section. Therefore, the individual measurements are automatically weighted according to their corresponding fractional part of the duct’s cross section and can be averaged arithmetically to determine the MEVF.

PIV data was processed using the Dantec Dynamics Studio software suite. After acquiring a set of double frame images, the images were masked if reflections occurred or if obstructions prevented complete illumination of the whole measuring plane. Then the resulting mean for each pixel’s grey scale value was calculated and subtracted from each individual image, thus eliminating background illumination. Afterwards, each double frame image was processed utilizing an Adaptive PIV approach with a $5 \times 5$ neighbor as well as velocity range validation. The resulting vector fields were then averaged arithmetically.

**Test Rig Design Verification**

Although the pitot rake could be used for measuring the IGCF, this was obtained from 46 individual PIV measurements near the generator top, which provide a higher spatial resolution. For these measurements, the light sheet was inserted vertically into the test rig with the camera being positioned sideways. The IGCF was obtained by evaluating the time averaged velocity component $u$ over the generator top height at the same position as the pitot rake. The IGCF inflow profile is illustrated in figure 4a. The results of the volume flow measurements are given in table 1 along with the target values from the design process as well as the relative deviation. The MEVF was found to be 0.4% below the target value whereas the IGCF was 1.7% too high. The reason the IGCF’s deviation was greater than the MEVF’s is due to the fact that the IGCF as well as the whole flow field around the generator is very sensitive to changes of the generator fan speed. Since the IGCF consists partly of the MEVF and they interact significantly with each other, especially during reentry of the IGCF into the package enclosure. Therefore, changing the flow rate of the IGCF influences how the MEVF passes through and around the generator. Nevertheless, the deviation of the IGCF can be regarded as being sufficiently small and the operation condition requirements to be fulfilled.

**NUMERICAL INVESTIGATION**

In the following sections the CFD methods and results will be discussed. Therein, details about the numerical method and modelling approach, the chosen spatial and temporal resolution, as well as the numerical setup will be given. Best practice guidelines for numerical ventilation flow investigations were described by Ivings et al. (2003), which were considered within the
scope of this work. Although the main focus of this section will be on the verification process, the evaluations of the ventilation flow will be discussed as well.

**Solution Method**

The CFD investigations were conducted using ANSYS CFX release 19.0, which is an unstructured solver utilizing a finite volume method. For convergence acceleration, the RANS equations are solved using an Algebraic Multigrid approach in conjunction with an Incomplete Lower Upper factorization algorithm. For the temporal discretization a Second Order Backward Euler scheme was used. All simulations were carried out using the Shear Stress Transport (SST) $k - \omega$ turbulence model.

**Spatial and Temporal Resolution**

For mesh generation, an unstructured approach utilizing automated meshing with ANSYS ICEM 19.0 was chosen. Tetrahedral elements in the core flow region were combined with pyramid elements near walls for boundary layer resolution. Great care was taken to ensure that the CFD mesh represents the test rig geometry as close as possible. Only extremely small features like screws were omitted and the geometry was only slightly modified in regions, which otherwise would have lead to very small element angles. Unlike in many investigations found in the open literature, the piping of the fuel and oil system were included in the CFD model as well. This approach was taken to avoid modelling errors introduced by e.g. porous media. Though able to enforce pressure losses caused by pipe flows, these are unable to describe the resulting flow structures let alone the interaction with other flow phenomena in close proximity to the piping (e.g. stagnation zones). To ensure sufficient spatial resolution of the CFD model, a comprehensive grid sensitivity study was conducted. Therein, grid convergence was investigated first without fully resolving the boundary layer and afterwards with a resolved boundary layer. For the latter case, the dimensionless wall distance $y^+$ was kept below 1 around the engine and constant during mesh refinement. By investigating grid convergence of the pressure loss coefficient with and without boundary layer resolution, sufficient mesh resolution for free flow and wall bounded phenomena can be attained independently. More details on the results of the grid study are given by Kowalski et al. (2017).

One of the modelling challenges is the wide range of time scales, ranging from an estimated Kolmogorow time scale near the package inlet ducts of around $2.1 \cdot 10^{-5}$ s up to an through flow time (TFT) around 0.164 s. In order to choose a suitable time step, a time step study was conducted. This revealed that for acceptable convergence (maximum residuals below $1 \cdot 10^{-3}$) a time step of $1 \cdot 10^{-6}$ s has to be chosen. However, due to the high element count (42 m cells) and the long simulation time ($10 \cdot$ TFT) this time step was regarded as unfeasible computationally. Therefore, coarser time steps were considered up to $2 \cdot 10^{-5}$ s and their influence on
the local velocity were investigated over a simulation time of $0.1 \cdot TFT$. This study revealed that choosing coarser time steps than $1 \cdot 10^{-6}$ s leads to a smoothing of the local velocity fluctuations. Regarding the time-averaged results, simulations with coarser time steps show good agreement to the reference. However, concerning the stability of the simulation a time step of $2 \cdot 10^{-3}$ s was regarded as too coarse and therefore a time step of $2 \cdot 10^{-4}$ s was found to be a suitable trade off between temporal accuracy, numerical stability and computation cost.

### Numerical Setup

Two different model setups were investigated that mainly differ in the way the IGCF is modelled. A brief overview of the applied boundary conditions is given in figure 2 and table 2. At the enclosure inlets (1), the velocity profile obtained from the CPT probe measurements is set for both ducts. At the enclosure outlets (2), the static pressure (slightly below atmospheric pressure) is set based on pressure tap measurements. To include the IGCF induced by the generator fans, a subdomain approach was taken (3) by modelling the fans via a global momentum source. In the first test rig model TRM-A, the generator top consists of one single subdomain and the momentum source is induced in x-direction (figure 2b). In the second model TRM-B, the generator top is split into two subdomains and the momentum source is applied in opposing, outward facing y-directions (figure 2c). For TRM-B, the two subdomains are separated by a thin wall, as in the actual test rig. The subdomains are connected to the enclosure domain by General Grid Interfaces (GGI) (4). All walls feature an adiabatic, no-slip boundary condition (5). Due to the test case being adiabatic and the low Ma-numbers, the isothermal energy model was chosen to speed up the solution process.

### Model Verification

To verify the numerical results, both CFD models shall be compared to PIV data. The position of the corresponding evaluation planes (EP) is shown in figure 3. The PIV images were evaluated as described beforehand, the CFD runs were averaged arithmetically over the whole simulation time. The results for EP-GN are given in figure 4 for the velocity component $u$, which corresponds to the IGCF. Since $u$ is oriented perpendicular to EP-GN, the distribution of $u$ was obtained by 46 individual PIV measurements, in which the light sheet was aligned in xz-direction and traversed laterally in front of the generator intake. The experiment (4a)
reveals that each of the two inlet jets of the MEVF coheres before entering the generator top. The position of the jets in y-direction corresponds with the inlet duct axis but the left jet is slightly distorted and displaced upwards compared to the opposing jet. Another feature of the IGCF is the presence of a backflow in the middle of the generator intake. A portion of the MEVF entering the generator is not transported outwards by the generator fans but is redirected to the generator intake by a guide plate. Despite the slight differences between the two jets, the IGCF can be described as rather symmetrical.

The IGCF could be matched integrally with deviations of +6% (TRM-A) and +4% (TRM-B). This is mainly due to the high sensitivity of the flow field to the interaction of the MEVF and the IGCF around the generator, as discussed beforehand. Generally, both CFD results show the main IGCF features, i.e. the occurrence of the two jets as well as a backflow region but they also differ from each other. When the results of TRM-A (4b) are compared to the experiment, it becomes evident that the backflow region is shifted towards the left jet, which in return is displaced towards the side of the generator. Not only does this result in a quenching of the left jet profile, the position of the shear layer between the inflow and backflow differs significantly from the experimental results. When regarding TRM-B (4c), the position of the backflow region and the shear layer is in better agreement with the experimental data. It must be noted that for TRM-B a recirculation region can be observed in the upper middle section of the EP. This flow structure could not be observed in the experiment, and results from the backflow exiting the right side of the generator top. However, TRM-B is regarded to be in better accordance to the experiment than TRM-A.

In addition to the generator intake velocity, further PIV measurements have been compared to the numerical results. To ensure comparability of the PIV and CFD data, the magnitude of the velocity vector projected on the EP (e.g. $c_{xz}$) will be examined. When investigating the flow field near the enclosure exit (figure 5a-5c), the PIV data shows that the flow accelerates as it is sucked towards the package outlet and that it enters the EP diagonally, as it originates from overflowing the gas turbine components. Each CFD model shows reasonable agreement with the experimental data, both in the position of the suction area as well as the velocity magnitude. For each CFD model a more shallow inflow angle compared to the experiment occurs. Herein, both models give similar results.

By investigating the low velocity flow region below the gas turbine diffusor (figure 5d-5f), the PIV data reveals that the flow enters the EP rather slowly coming from the bottom section below the diffusor and is accelerated while flowing diagonally towards the enclosure outlet. TRM-A overestimates the velocity magnitude of the flow entering the investigated region and it enters facing downwards in contrast to the experimental findings. Using TRM-B, the inflow angle as well as the inflow velocity magnitude are in better agreement with the experiment.
Both models show an upward turning of the flow on the left-hand side of the EP, which could not be observed in the experiment. However, TRM-B can be found to compare better with the experimental data even though slight differences occur in terms of kinematic similarity. It must be noted that especially in such stagnation regions the standard deviation of the experimental data was found to be relatively high with respect to the resulting mean values. Although the considered flow region is located far downstream of the package inlet, it is still affected by the different configurations of the generator flow. This underlines the importance of the accurate modelling of such relevant flow features when conducting enclosure flow investigations. Due to the presented findings, TRM-B was chosen as the most suitable CFD model.

![Figure 5: Local velocity field comparison of PIV and CFD data](image)

**Evaluation of the Ventilation Flow**

In this section, the ventilation flow will be analysed to identify worst case leakage scenarios for future investigations. At first, the occurrence of backflow from the gas turbine to the gear section of the enclosure will be checked. For this purpose, the distribution of $u$ around the gas turbine intake (EP-GI) is regarded as given in figure 6 with positive values indicating a flow towards the enclosure outlet. It becomes evident, that most of the cross section’s flow is moving towards the gas turbine section. When regarding the flow near the intake, regions of backflow can be observed, especially in the bottom region. These flow structures are caused by the flow separating almost completely from the intake and do not represent actual backflows from the gas turbine to the gear section of the enclosure but are found to be local recirculation zones. Therefore, the ventilation flow can be regarded as capable of separating both enclosure sections and the search for critical leakage scenarios can be focused on the gas turbine section.
To identify critical flow regions for future leakage scenarios, stagnation zones have to be evaluated since leakages could possibly accumulate there. Due to the three-dimensional nature of the ventilation flow, instead of only investigating discrete EPs, a volumetric approach was taken. For this purpose, cells with a velocity magnitude $c < 0.5 \text{ m/s}$ were identified. This value was chosen since it is slightly above the maximum laminar flame propagation velocity of $\text{CH}_4$. To avoid confusing boundary layers with stagnation zones, only cells at a certain distance from bounding walls were considered. The results are given in figure 7 and show, that the ventilation flow is well designed, as only rather small and scattered stagnation regions are present in the engine section. For future leakage investigations three main regions of interest can be identified. The first one is located directly above the combustion chambers of the engine (L1), the second one occurs underneath the gas turbine diffusor (L2) and the last one in the pit underneath the engine (L3). Though L1 and L2 were already identified by Kowalski et al. (2017) using stationary CFD data, L3 could first be detected by the transient investigations presented here.

![Figure 6: Flow around the engine’s gas turbine intake (EP-GI TRM-B)](image)

![Figure 7: Core flow stagnation regions ($c < 0.5 \text{ m/s}$) near the gas turbine (TRM-B)](image)

**CONCLUSIONS**

The ventilation flow of a MAN Energy Solutions SE gas turbine enclosure was investigated experimentally and numerically using a scaled test rig. The experimental facility and measurement setup were described. A CFD model of the test rig was created and the results from the measurements were used to apply appropriate boundary conditions as well as to verify the chosen model. Two different approaches were chosen for modelling the engine’s actively ventilated generator flow and were compared to experimental PIV data. This verification process revealed that representing the generator fans with two independent, outward facing momentum sources (TRM-B) leads to better agreement with the experimental findings than using only one momentum source facing in the main flow direction (TRM-A). The verification process also showed the sensitivity of the flow in a stagnation region far downstream to the modelled generator flow. The results from TRM-B were then used to identify regions of interest for future leakage scenarios. No relevant backflow from the engine to upstream regions of the enclosure could be observed and therefore the investigation was focused on the engine section of the enclosure. Three scattered stagnation regions could be identified with one region first being identified by the transient simulations presented here. These three stagnation regions should be considered for leakage investigations. Though the flow field results presented here are machine-specific,
the general workflow of verifying and evaluating ventilation flows can be assumed to be transferable to other engines. Future work will include the investigation of fan failures by altering the boundary conditions at the in- and outlets and the implementation of under expanded leakage jets by resolving these jets to conduct leakage scenarios in the whole enclosure.

ACKNOWLEDGEMENTS
The investigations were conducted as a part of the joint research program COORETEC-Turbo. The work was supported by the German Federal Ministry for Economic Affairs and Energy (BMWi) under grant number 03ET7030B. The authors gratefully acknowledge COORETEC-Turbo and MAN Energy Solutions SE for their support and permission to publish this paper. Furthermore, the authors would like to express great appreciation to Martin Lauer for his help in the construction of the test rig and the conduction of the PIV experiments as well as to Yannik Kolberg for his help in the post processing of the CFD results.

REFERENCES


