# *Combustor Exit Flow Sensitivity towards Small Changes*

Project Manager
Anton Hilfrich

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

Project Term
2024 - 2024

Clusters
Lichtenberg II Cluster Darmstadt

Additional Software
PRECISE-UNS, Paraview

Institute
Gasturbinen, Luft- und Raumfahrtantriebe (GLR)

University
Technische Universität Darmstadt

## Introduction

The design of the combustion chamber in a jet engine is most crucial when it comes to reducing emissions of pollutants such as soot and NOx. In order to ensure the intended flow behaviour over the range of different operating points of the engine, the combustor must be designed in a robust way. Robust in this context means that a change in the combustor inlet signal is not directly transferred to its output signal. The inlet signal here is the exit flow of the compressor which is located upstream of the combustor. On the one hand, it is difficult to predict this traverse during the design phase and on the other it changes during operation for different operating points. Combustion chambers are usually equipped with cowls that are used to divide and feed the incoming airstream to the injector and the inner and outer annulus. However, the cowl's design acts like an aerofoil which accelerates the air flow near the liner or flame tube. This leads to high velocities just above the mixing ports impairing the feeding of the fresh air. On the one hand, the sensitivity towards changes at the combustor inlet is assumed to be increased by this. On the other hand, this impaired feeding can lead to separation within the mixing port leading to hot gas entering the port and overheating the materials. This can lead to damages to the geometry. This project aims to find means and methods to improve the described feeding to the ports with the goal to increase the reliability of combustion chambers. Another goal is the increased sustainability due to the reduction in damages to the geometry. These investigations are done numerically using computational fluid dynamics. In order to properly model and predict the flow, the fuel spray and also the combustion, great computational resources are required. Additionally, due to the iterative nature of the process, many design variations have to be simulated which would not be possible in reasonable time without the use of the NHR4CES high performance computer.

## Methods

The simulations in this project are performed using PRECISE-UNS which is a cell-based finite volume CFD solver developed by Rolls-Royce. Its main capabilities and specialisation lie in the simulation of aero-engine combustors. In this project different turbulence models were investigated within PRECISE-UNS. All of them were Steady Reynolds-averaged Navier Stokes models which allow for quick geometry variations. After evaluating different models, the $k$-$\varepsilon$ realisable model has been chosen for the turbulence treatment. The combustion is modelled using the flamelet generated manifold approach. The fuel is injected at prescribed positions in the domain using a spray model combined with a Lagrangian particle tracking method.

## Results

First, different turbulence models are evaluated to validate the numerical setup. The simulation results are compared to the very limitedly available experimental data. This is done qualitatively in the outer and inner annulus. The $k$-$\omega$ SST model and SAS model showed significant overprediction of separation along the cowl. The RSM model showed better alignment with experimental data regarding the separation but is computationally more expensive. Lastly, the $k$-$\varepsilon$ realisable model was in good agreement with the experimental data regarding the overall flow phenomena and separation. Due to its lower computational costs, it has been chosen for the further investigations. Different designs have been investigated with regards to their ability to properly feed air to the mixing ports. In order to improve the feeding, the flow in the annulus can be guided towards the casing or better should be made more diffusive so that there is a radially homogenous velocity profile in the annulus. This can be achieved e.g. by changing the cowl’s geometry but also by changing the position and shape of the cowl holes. The studs at the liner seem to have a major influence on the circumferential distribution of the flow in the annulus which in return has an influence on the feeding of the ports.

## Discussion

After choosing promising design modifications to the cowl, a parameter study will be conducted to identify parameters that can be used to evaluate the quality of the feeding to the port during the design. The robustness of these designs against changes from the compressor exit flow have to be evaluated.

## *Last Update:* 2025-01-08 16:23