Strengthening of the air pollutions and noise reduction norms in last decades (see the research agenda by ACARE, Advisory Council for Aeronautics Research in Europe) forces the engineers to design new more efficient components of aero-engines. To reach these goals the improved strategy for simulation of the high-Reynolds number flows has to be developed.

Computational fluid dynamics (CFD) is widely accepted tool, in academia and in an engineering, for prediction of both laminar and turbulent flows. In a wide range of applications, the solution of high-Reynolds number flows might be obtained by employing the cost-effective methods known as time-averaged Navier-Stokes equations. But there exist a wide range of flows, characterised by complex flow dynamics, which requires an application of expensive CFD strategy, namely Direct Numerical Simulation (DNS). In DNS, the equations of motions are solved on very dense meshes using very small time steps in order to capture, besides the energy-bearing eddies, the very small dissipative Kolmogorov scales.

Flow inside a modern turbofan engine is an example of the complex flow system, for which an application of the standard fully-turbulent time-averaged Navier-Stokes techniques might result in significant errors in obtained simulation results. In this inherently unsteady flow the laminar boundary layers on the surfaces of the turbine rotor blades and stator vanes are exposed to strong perturbations caused by acoustic waves. The traveling waves may trigger the transition to turbulence inside the pseudolaminar boundary layers. As consequence, the shear stress forces acting on the surfaces of blades and vanes might be altered in time course. To address the complexity of the flow physics the high-fidelity tool, like DNS, has to be employed. Unfortunately, this method cannot be applied in engineering practice due to the high-Reynolds number of the flow. So the reduced-order methods have to be elaborated, which allow to model the overall flow physics at low numerical cost. The aim of the current research is a development of a new reduced-order method based on statistical approaches, like the time-averaged Navier-Stokes equations, for numerical simulation of the aero-acoustics coupling mechanism.

The experiments and DNS will be performed during the project to gain insight into the flow-physics. In following, the two reduced-order flow-acoustics coupling strategies will be developed and tested. The first method will be based on resolving the flow unsteadiness in the separated laminar boundary layer caused by the acoustic excitation mechanism by means of 3D URANS (Unsteady Reynolds-averaged Navier-Stokes) and/or hybrid RANS-LES (Large Eddy Simulation) techniques. The second method will be based on generation of the synthetic turbulence field outside of the boundary layer flow in order to mimic the presence of the acoustic waves in numerical simulation using URANS. This synthetic turbulence level will affect the laminar boundary layer boundary layers and activate the flow transition from laminar to turbulent state. The developed reduced-order methods, might be later applied in design process of the aero-engine components, without necessity to use very expensive DNS approach.