NUMERICAL INVESTIGATION OF SECONDARY FLOWS IN LOW PRESSURE AXIAL COMPRESSOR STAGE FOR CONTRA-ROTATING TURBOFAN ARCHITECTURES

Balazs Farkas, Hungary
Supervisor: Associate Prof. J.-F. Brouckaert

Contra-rotating fan engines are considered to be a possible alternative for the new generation of the future environmental friendly engines for commercial airliners. Although there are certain similarities such an architecture is significantly different from the one of the present high bypass ratio turbofans which implies that the conventional design concepts has been used so far might has to be reconsidered.

The problem in case of the booster stages is that the LP compressor rotor is mounted on the same shaft as the second fan stage which results in a lower rotational speed and therefore a much higher loading as encountered in conventional high bypass ratio aero-engines. One way to decrease the loading is to increase the circumferential speed by increasing the mean radius of the rotor but it means that the blade aspect ratio has to be decreased accounting the same mass-flow rate which makes the compressor more sensitive to the secondary flows.

To investigate the effect of the new design a one and a half stage compressor representing the front row of a similar booster stage was designed and manufactured at VKI and subsequently tested in the closed loop test facility (VKI-R4). This study represents the numerical survey on the test section which was performed parallel with the experimental work to estimate the potential of the available CFD codes and to allow of better view on governing phenomenons of the operating compressor. In this study, particular interest was given to investigate the effect of the seal-leakage flow around the stator hub platform on the performance. To study the effect of the seal-leakage flow three different seal cavity configurations with different seal-tooth gaps sizes were simulated in comparison with no-cavity configuration. This set of investigations allowed to assess the different models by comparison with the results obtained experimentally. This comparison was made on the global performance of the stage, including the impact on the stability range, as well as on the flow field itself in particular in the rotor and stator exit planes. The computations were performed by using Numeca developed code Fine-Turbo with steady RANS solver.

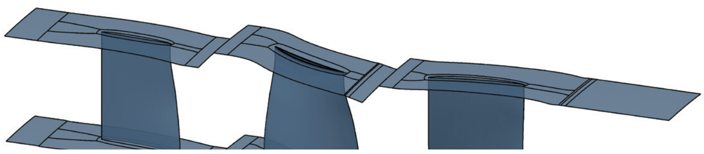

Figure 1: 3D representation of the numerical domain