HEAT TRANSFER AND FLOW FIELD INVESTIGATION IN RIB-ROUGHENED TRAILING EDGE CHANNEL WITH CROSSING-JETS

Filippo Coletti, Italy
Supervisor: Prof. T. Arts
Promoter: Prof. T. Arts (Université Catholique de Louvain, Belgium)

Modern gas turbine engines use active cooling in the key components of the turbine to prevent failure under the extreme operating conditions they face. Internal cooling is one of the primary techniques used for this purpose: air is bled from the high pressure compressor, by-passes the combustor, and enters through the root of the blade. In modern turbine blades, there are three to four sets of cooling passages, one for the leading and trailing edge areas and one to two sets for the mid-chord passages. Cooling the trailing edge represents a challenging task, since the aerodynamic requirement of high slenderness is conflicting with the need of integrating internal cooling cavities.

The objective of the present research is to investigate the heat transfer and the flow field within a trailing edge internal cooling channel, characterized by a trapezoidal cross-section and inclined ribs on one wall. Slots on two opposite walls provide entrance and discharge of the coolant. The study is performed on a scaled-up model machined out of Plexiglas.
The channel with and without ribs is characterized with respect to Reynolds number and pressure drop using pressure taps, Pitot tubes, and a hot wire. The flow field is qualitatively investigated using surface flow visualization techniques such as ink dots tracers and wool tufts, obtaining a first flow model. The flow model is refined thanks to a PIV measurements campaign conducted on several planes in the central part of the channel. Highly three-dimensional flow structures are detected and characterized.

Spatially resolved distributions of the convective heat transfer coefficient are obtained on the ribbed wall (including the rib surface) and the opposite smooth wall. For this purpose, the wall temperature is measured using thermocromic liquid crystals.

SNECMA, industrial partner of this project, performs numerical simulations using the Reynolds averaged Navier-Stokes solver CEDRE (ONERA). The predictions are compared with the experimental data.

The influence of geometry variations such as rib-slot relative position and rib tapering on the aero-thermal performance of the channel is analysed. Four geometrical configurations are characterised and compared.

The combined effects of convection between coolant and channel surface and conduction within the wall are investigated following a conjugate approach, where coupling between the two heat transfer modes is considered. For this purpose a ribbed wall machined out of conductive material is used. Highly spatially resolved distributions of non-dimensional wall temperature are obtained by means of infrared thermography. The conduction through the wall is solved using a finite element analysis, allowing the evaluation of the local heat flux and Nusselt number distributions at the fluid-solid interface. The results are directly compared with purely convective heat transfer distributions. It clearly appears that the effect of wall conduction can by no means be neglected. The effect of the solid conductivity is assessed comparing three different materials for the rib-roughened wall. The computed patterns of heat flux across the wall confirmed the major role of the ribs in the heat extraction process.

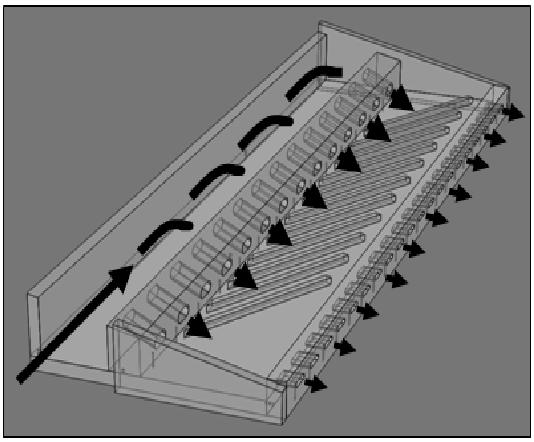

Figure 1: Test section