HEAT TRANSFER IN HIGH SUBSONIC VELOCITY ENVIRONMENTS BEHIND THE FAN OF A GAS TURBINE

Francesco Baldani, Italy
Supervisor: Prof. T. Arts
Promoter: Prof. W. Bosschaerts (RMA, Belgium)

The proposed work is a Ph.D. oriented study with the aim to develop innovative tools and models for a better understanding of heat transfer phenomena in high subsonic velocity environments. The aim is to study the heat transfer from a hot surface to a cold flow under conditions similar to those encountered by aircraft turbine engines, hence using the main flow of the engine to study the heat exchanger just downstream the fan. The core of the study is the solution of the so called "inverse heat conduction problem" that consists in imposing the object surface temperature as boundary condition and computing via numerical methods the resulting heat flux. The heat transfer coefficient is to be studied both locally and globally, since a global study of *h* is fundamental in the preliminary design phase while a local study of *h* is useful to understand the effects of the geometry on the local flow conditions and on the achievable heat exchange. Recalling that the convective heat flux can be expressed as $\dot{q} = h(T_{surf} - T_{\infty})$ a combination of experimental and numerical work will be required in order to compute the heat transfer coefficient. In fact to accomplish the task one should first carefully measure T_{surf} (using for instance the I.R. imaging technique) and estimate the air bulk temperature T_{∞} (from T_{inlet} and T_{outlet}) then only a numerical simulation could be used to solve the problem given the non uniform spatial distribution of the heat flux density $\dot{q}$ coupled with the need for an important accuracy. The computational model should faithfully represent the experimental set up. For this purpose an extreme care is required for the boundary conditions that are to be superimposed to perform the calculation (e.g. a non-uniform layer of the paint deposited on the model surface, acting as thermal barrier could lead to erroneous temperature measures and consequently to problems when computing the conjugated heat transfer or analyzing the transient heat transfer). The experimental campaign has to be carefully performed. The surface temperature will be investigated with a non-intrusive measurement techniques, the Infra-Red camera imaging that will be carried out analyzing first simple heat exchanger geometries to acquire insight in the physics, gain expertise and develop data reduction procedures and tools. In a next phase more complex geometries with industrial relevance in the domain of turbine engines will be examined. It should be clear that to deepen the understanding of the heat transfer occurring in installations as finned heat exchangers one has always to consider that a simple superposition of the heat exchange phenomena is not enough. Indeed one should rather talk of "conjugate" heat transfer this latter being an actual coupled interaction of conduction and convection. For instance neglecting the effect of the wall conduction that greatly affects the temperature distribution will result in imposing unreal thermal boundary conditions at the solid-fluid interface leading to artificial heat transfer rates (and coefficients). Another crucial issue is the characterization of the wind tunnel flow field. The facility that will be exploited for most of the experiments is a blow-down wind tunnel where the air coming from a pressurized tank can be heated. The velocity in the test section during the test ranges between 30 m/s and 230 m/s and the temperature varies between 290 K and 320 K giving us the chance to have flow conditions satisfying the similarity rules to those appearing at the inlet of an aircraft engine during various flight conditions. To characterize the flow field obtainable in the wind tunnel the C.T.A. (Constant Temperature Anemometry) technique and the P.I.V. (Particle Image Velocimetry) technique will be exploited. The flow being highly turbulent we expect the forced convection to be the major phenomenon occurring. Nevertheless this behavior should be carefully checked to understand if just velocity measurements will be required or if also temperature measures should be performed. In the latter case one should also check if the chosen measurement technique is sensitive enough to catch the influence of fluid temperature variations within the domain of confidence. In the frame of the wind tunnel flow field analysis CFD simulations are foreseen.

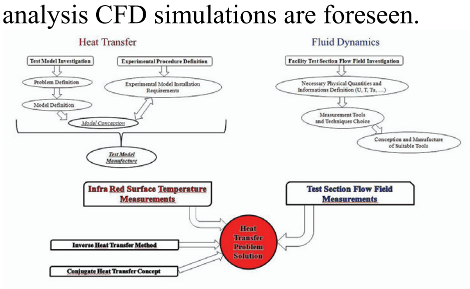

Figure 1 : Proposed Study Solution Approach

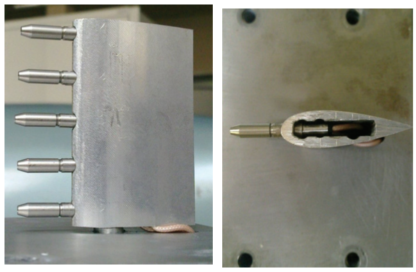

Figure 2 : Conceived and manufactured H.W. rake