HEAT TRANSFER AND TURBULENT EFFECTS IN A COOLED ROTATING CHANNEL

B. Tartinville* and Ch. Hirsch*
*Numeca International, Belgium

1. Introduction

The fuel efficiency of jet engines used for aircraft propulsion is dependent on the performance of many key engine components. One of the most important is the turbine whose efficiency has a large influence on the engine fuel consumption and hence its CO_2 emissions. Improving the performance of gas turbines requires increasing the turbine inlet temperature. Therefore, the turbine must operate with high efficiency in the most hostile environment of the engine. Insofar as blade lifetime strongly depends on its surface temperature, it is necessary to design an efficient cooling system that protects the blades from thermal stresses and from enhanced life cycle fatigue. However, film cooling efficiency depends on numerous interacting parameters - for instance the free-stream turbulent intensity, the Reynolds number, the presence of secondary flows and the film cooling blowing ratio (see for instance Dunn [2001] for a review of turbine film cooling). Thus, the design of turbine cooling systems remains one of the most challenging processes in engine development. Modern high-pressure turbine cooling systems invariably combine internal convection cooling with external film cooling in complex rotating flow systems whose individual features interact in complex ways. The heat transfer and cooling processes active are at the limit of current understanding and engine designers rely heavily on empirical tools and engineering judgment to produce new designs.
Therefore, Computational Fluid Dynamics (CFD) becomes indispensable in order to understand the various interactions that influence cooling efficiency. Furthermore, CFD could also be of great interest in the design of internal cooling passages and cooled turbine blades. Because the flow in the internal cooling passage is turbulent and can occur in a rotation frame of reference, it is of crucial importance to be able to accurately predict the turbulent field in such a complex system.

Various turbomachinery applications are presented in these notes, where rotation could have an impact on the efficiency of the cooling system. In order to illustrate the complexity of the flow that have to be treated by CFD, in the first part of the paper calculations performed on a cooled high pressure turbine (including all the details of the cooling system) will be presented. Afterwards, a review of the various methods proposed in the literature in order to treat rotation correction inside turbulence models will be presented. Finally results obtained for a rotating U-bend using the Spalart-Allmaras turbulence model with rotation/curvature corrections will be presented and compared to experimental data.