Abstract

In this paper, the commercial ANSYS Fluent software was used to achieve a numerical survey of the effect of five (05) turbulence models' formulation on the aero-thermal computational fluid dynamics validation of two 30:1 scaled models reproducing an original internal ribbed trailing edge prototype. Tests were conducted for stationary and rotation conditions, for Re = 10,000–40,000 and Ro = 0–0.23. Particle image velocimetry and thermochromic liquid crystal experimental data were employed to check the consistence computational fluid dynamics results qualitatively and quantitatively, aerodynamically and thermally, for various working conditions. Numerical predictions revealed that the choice of the turbulence model affects the accuracy of results. Concerning the shear stress transport k-w model, limiters defined in the eddy viscosity formulation induce a surplus estimation of the turbulence kinetic energy (k) which leads to noticeable discrepancies in terms of velocity profiles and recirculation zones. Also, numerical calculations confirmed former experimental assumptions concerning origins of the aerodynamic structures and heat transfer's features, especially, those related to the increase of the cooling temperature balance efficiency, the appearance/disappearance of the horseshoe structures within the trailing edge region and velocities/boundary layers' profile variations. The obtained results assist the understanding and the forecast of the flow field behavior, throughout the design process, by the assessment of the aerodynamic and thermal performances within the considered blade's cooling system.