ABSTRACT

Enhancing the efficiency of transonic blades in aircraft engines is a key aspect in designing. Complex flow structure caused by shock waves and interaction with the boundary layer may lead to non-stationary effects influencing the dynamic state of the system, decreased efficiency, as well as fluctuations in mass flow rate. Many studies focus on utilizing innovative techniques for reducing boundary layer separation through turbulence generation, which positively affects system stabilization. The key question is about the influence of the location of the laminar-turbulent transition on the structure of the interaction region, how close to the shock wave one should induce the laminar-turbulent transition and reduce the separation zone.

In the initial stage of the work, the characteristic flow structures in the transonic compressor cascade were analyzed within the framework of both two-dimensional and three-dimensional model, as well as the examination of the influence of active flow control on the corner structures in the blade passage channel.

The second stage focuses on the design process of a measurement setup aimed at reproducing a flow representative of the transonic compressor cascade, specifically the suction side of the profile. In this part, the shape of the walls of the test section was designed based on the streamlines determined from the flow in the linear cascade. The analysis also included studying the impact of the profiles' positioning, in relation to the walls of the measurement setup and the angle of inflow on the configuration of shock waves. Furthermore, the influence of flow control holes utilized in the corners was examined based on three-dimensional flow calculation.

In the subsequent stage, a detailed description of the computational model and the measurement results obtained in the transonic laboratory at IMP PAN. The experimental data validate the numerical model by showcasing the velocity distribution upstream of and downstream of the profiles, the isentropic Mach number, the wave pattern, the visualization of density gradient and oil flow visualization in close proximity to the suction side of the lower profile, the static pressure distribution and the aerodynamic wake.

In the final (fourth) stage of the work, an analysis was conducted on the influence of the position of the elements inducing laminar-turbulent transition on the airfoil, using two cases and in comparison with experimental data. The first configuration was chosen to induce laminar-turbulent transition as close as possible to the shock wave to achieve a thin turbulent boundary layer ahead of the shock wave. In the second configuration, it was decided to induce a laminar-turbulent transition immediately after the leading edge of the airfoil to obtain a turbulent boundary layer that can develop over a longer distance compared to the previous configuration. Additionally, a numerical analysis was conducted to investigate the influence of the shock wave interactions with the boundary layer on the height of the step.

To achieve this, two additional geometries were prepared, resulting in a total of three variations with different step heights for both the first and second configurations. The obtained results were compared with the reference case to evaluate the differences.

The conclusions indicate that no significant qualitative changes are observed for small step heights on the airfoil compared to the reference flow. Increasing the dimensions of the step leads to the thickening of the boundary layer ahead of the interaction zone, widening the aerodynamic wake and reducing efficiency. A positive impact of inducing laminar-turbulent transition was observed only in one experimental case. Further research is needed to better understand the discrepancies between experiments and RANS. The existing setup designed for this study can be successfully used for further research.

Gdańsk, 30.05.2023