NUMERICAL SIMULATION OF FLOW AND HEAT TRANSFER CHARACTERISTICS OF ASYMMETRIC BLADE LEADING EDGE STRUCTURE

Wang Lingyang[1], Mao Junkui[1,2], Wang Feilong[1,2]
[1]College of Energy and Power Engineering of Nanjing University of Aeronautics and Astronautics, Nanjing, China, wly_1230@163.com, mjkpe@nuaa.edu.cn, wangfl@nuaa.edu.cn
[2]Jiangsu Provincial Key Laboratory of Aeronautical Power System, Nanjing, China

Keywords: leading edge of blade, impingement cooling, composite cooling structure, numerical simulation.

The blade leading edge have the highest thermal load in the entire turbine blade and the impingement and film cooling structure are widely used in this region. The leading edge of blade tends to have a large curvature, which leads to its complex heat transfer and flow characteristics. However, at present, most of the related studies are focused on a semi-cylindrical structure or a triangular structure with rounded corners, which are symmetric. In consideration of aerodynamic requirements, the leading edge of turbine blade often presents an asymmetric concave structure and the pressure surface and suction surface are in different pressure environment. Therefore, this paper proposed a simplified asymmetric leading edge model of the real blades, and then investigated the effect of position of impingement holes on heat transfer and flow characteristics with steady and unsteady simulation. In addition, the influence of the asymmetric flow from the film holes is considered.

The leading edge model with impingement cooling structure only and with impingement-film composite cooling structure are studied respectively. For model with impingement cooling structure only, three models with different impingement holes offset distances (*e/d*=-1.5, 0, and 1.5) are studied. The inlet Reynolds number (*Re*) are 10000, 20000 and 30000 respectively. For the model of impingement-film composite cooling structure, five rows of film holes are added and the suction surface and the pressure surface each has a row. Simulating the real pressure distribution by changing the mass flow ratio of the fluid flowing out of the film holes of suction surface and pressure surface respectively. The numerical calculations are performed with the commercial computational fluid dynamics (CFD) software ANSYS Fluent 15.0. A pressure-based separation solver was used to solve the problem. SIMPLE algorithm is used for pressure velocity coupling and flow is considered as incompressible. The scheme of discretization is of second-order upwind accuracy. After verification, Realizable k-ε was selected as the turbulence model.

Fig.1 and Fig.2 shows the variation of the average Nusselt number (*Nu*) and the heat transfer non-uniform coefficient of the target surface with *Re* for three models (*e/d* = 1.5, 0 and -1.5). The results shows that the average *Nu* of the target surface with e/d=1.5 is the largest, which is 11.7% higher than that of the model with *e/d*=-1.5 for *Re*=20000. However, the thermal uniformity of *e/d*=-1.5 is the best. Fig.3 shows streamline diagrams of the cross-sections of the three impingement holes (Jet-1, Jet-3 and Jet-5) at different moments. The transient analysis shows that the flow period is 0.006 seconds and the flow has strong instability. Then, by adding film holes to the model with *e/d*=1.5, the impingement-film composite cooling structure is obtained. As shown in Fig.4 and Fig.5, the analysis shows that compared to the structure with impingement holes only, the heat transfer capacity of the composite structure does not change significantly, but the thermal uniformity of the target surface is significantly improved. For *Re*=20000, the heat transfer non-uniform coefficient decreases by 13.8%. At last, the outflow mass flow ratio from film holes was changed to simulate the real pressure. As shown in Fig.6, it was found that with the increase of the mass flow gap between the film holes on the pressure surface and the suction surface, the average *Nu* of the target surface decreases and the heat transfer non-uniform coefficient increases compared with the pressure-outlet.

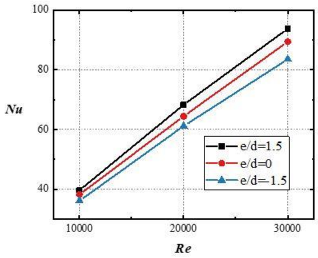

Fig. 1 – Average *Nu* of the target surface of the three models(*e/d*=1.5,e/d=0 and e/d=-1.5) varies with the *Re*

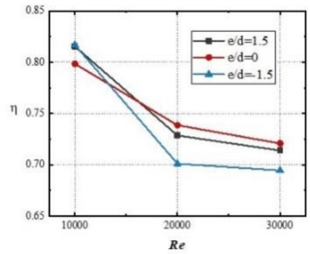

Fig. 2 – Heat transfer non-uniform coefficient of the target surface of the three models(*e/d*=1.5,e/d=0 and e/d=-1.5) varies with the *Re*

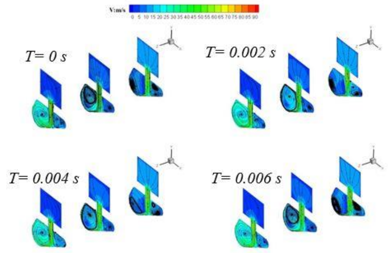

Fig. 3 – Streamline diagrams of the cross-sections of Jet-1, Jet-2 and Jet-3 at different moments

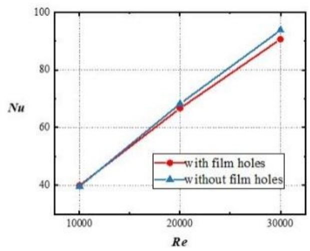

Fig. 4 – Comparison of the average *Nu* on the impact target surface with and without film holes under different *Re*

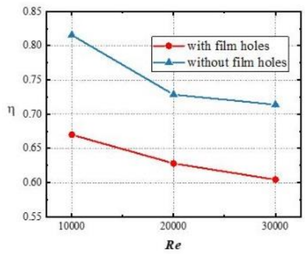

Fig. 5 – Comparison of the heat transfer non-uniform coefficient of the impact target surface with and without film holes under different *Re*

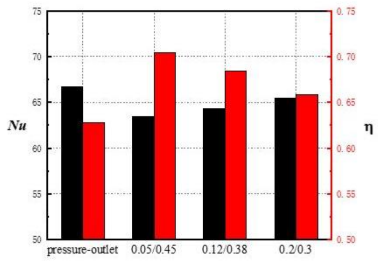

Fig. 6 – Comparison between the pressure outlet and the asymmetric outflow

From the results, it can be found that for the asymmetric leading edge structure, the offset impact holes can enhance the heat transfer of target surface and the film holes are beneficial to enhance the thermal uniformity of target surface. However, the asymmetrical outflow is harmful to the overall heat transfer of the structure.

Authors

Wang Lingyang, Master's degree in reading at College of Energy and Power Engineering of Nanjing University of Aeronautics and Astronautics, mainly research in turbine blade cooling.

Mao Junkui, Professor/doctoral advisor, dean of College of Energy and Power Engineering of Nanjing University of Aeronautics and Astronautics. His main research fields include thermal analysis and management of aero-engines, cooling design of high-temperature components, active control of air system and tip clearance, multi-scale thermal analysis of ceramic matrix composites, and numerical simulation of gas-thermal-solid coupling.

Wang Feilong,associate professor and master supervisor of College of Energy and Power Engineering of Nanjing University of Aeronautics and Astronautics. He is mainly engaged in thermal analysis and management, heat and mass transfer, efficient cooling, and multiphase and multiphysical field simulation of aero-engine / traditional heat exchanger.