AN EXPERIMENTAL AND NUMERICAL INVESTIGATION OF IMPINGEMENT HEAT TRANSFER IN AIRFOILS LEADING-EDGE COOLING CHANNEL

M.E. Taslim and A. Abdelrasoul

Mechanical and Industrial Engineering Department

Northeastern University

Boston, Massachusetts

USA

ABSTRACT

Leading edge cooling cavities in modern gas turbine airfoils play an important role in maintaining the leading edge temperature at levels consistent with airfoil design life. These cavities often have a complex cross-sectional shape to be compatible with the external contour of the airfoil at the leading edge. In some current designs, to enhance the heat transfer coefficient along the leading edge of an airfoil, the cooling flow enters the leading edge cavity from the adjacent cavity through a series of cross-over holes, cast on the partition wall between the two cavities. The cross-over jets impinge on the leading-edge wall then form a cross-flow that moves towards the airfoil tip. In this experimental setup, there were nine cross-over holes with race-track-shaped cross-sections on the partition wall (Fig. 1). To investigate the effects of cross-flow created by the upstream jets (spent air), on impingement heat transfer coefficients, five cross-over flow arrangements were studied. These flow arrangements were for 0, 1, 2, 3 and 4 jets upstream of the 5th jet for which the impingement heat transfer coefficients were measured. Tests were run for a jet to target wall distance ratio, Z/D_h, of 2.81 and a range of jet Reynolds numbers from 5,000 to 27,000 (Fig. 2). For the numerical analyses, all tested geometries were meshed with all-hexa structured mesh of high near-wall concentration. Boundary conditions identical to those of experiments were applied and several turbulence model results were compared. The numerical analyses also provided the share of each cross-over hole from the total flow for different geometries (Fig. 3). Comparisons between the experimental and numerical results are also made. The major conclusions of this study were: a) cross-flow produced by the upstream jets caused a slight reduction in impingement heat transfer coefficients, b) depending on the inlet flow conditions, there could be a significant variation in mass flow rate through the cross-over holes, and c) the numerical predictions of impingement heat transfer coefficients were in good agreement with the measured values thus CFD could be considered a viable tool in airfoil cooling circuit designs.

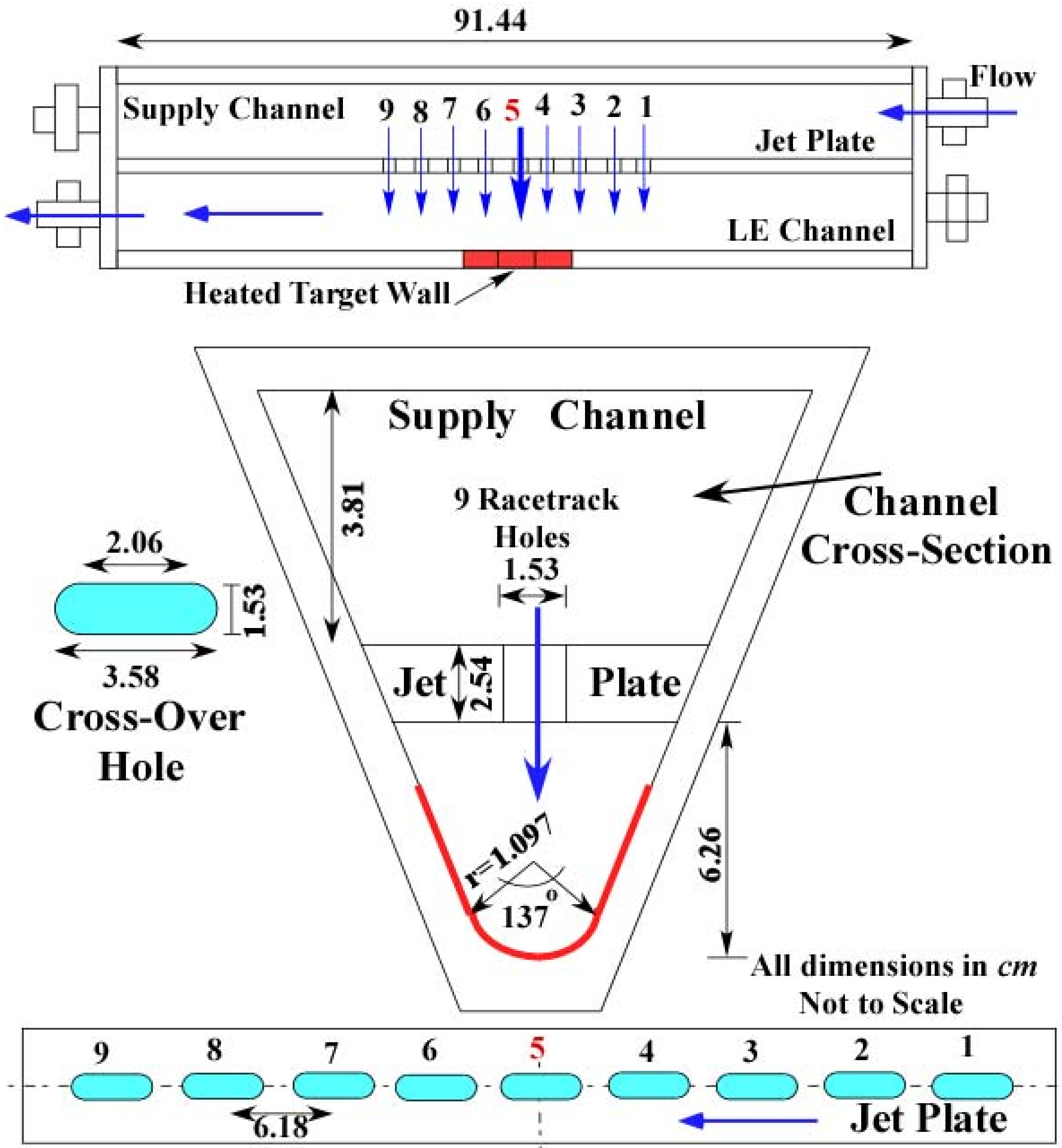

Fig. 1 Schematics of the rig.

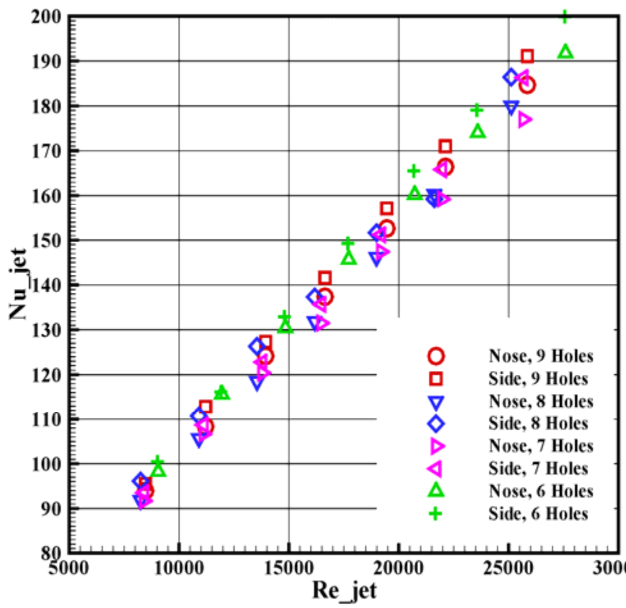

Fig. 2 Impingement jet Nusselt number versus jet Reynolds number on the nose and side walls for cases of 9, 8, 7 and 6 cross-over holes.

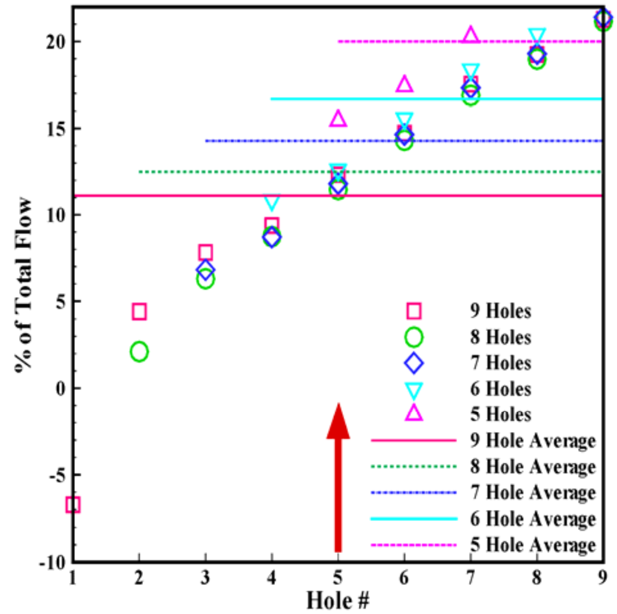

Fig. 3 Percentage of total flow through each cross-over hole for cases of 9, 8, 7, 6 and 5 cross-over holes.

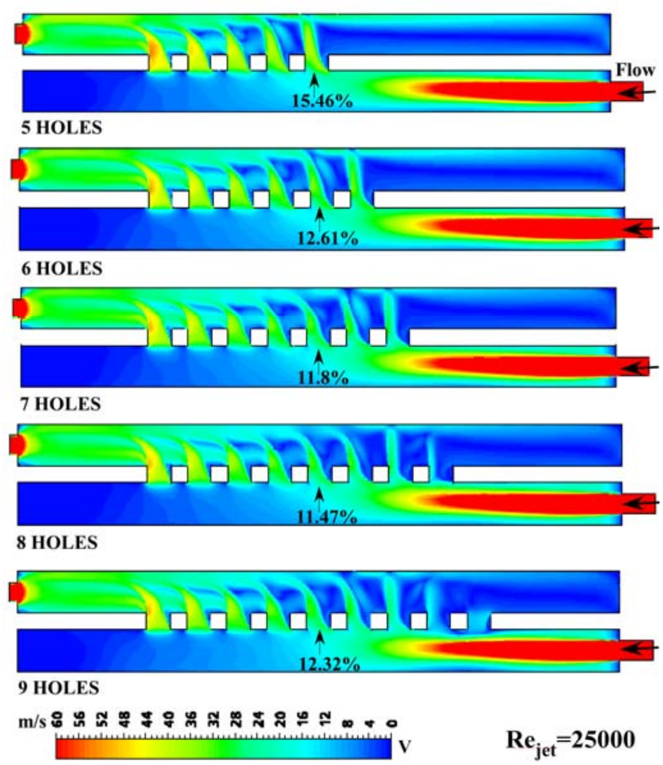

Fig. 4 CFD contours of velocity magnitudes for cases of 9, 8, 7, 6 and 5 cross-over holes.