# Experimental/Numerical Crossover Jet Impingement in an Airfoil Leading-Edge Cooling Channel

K. Elebiary and M.E. Taslim

Mechanical and Industrial Engineering Department Northeastern University Boston, MA, USA

## Abstract

Technological advancement in gas turbine field demands high temperature gases impacting on the turbine airfoils in order to increase the output power as well as the thermal efficiency. Leading-edge is one of the most critical and lifelimiting sections of the airfoil which requires intricate cooling schemes to maintain a robust design. In order to maintain coherence with a typical external aerodynamic blade profile, cooling processes usually take place in geometrically complex internal paths where analytical approaches may not provide a proper solution. In this study, experimental and numerical models simulating the leading-edge and its adjacent cavity were created. Cooling flow entered the leading-edge cavity through the crossover ports on the partition wall between the two cavities and impinged on the internal surface of the leading edge. Three flow arrangements were tested: 1,2) flow entering from one side (root or tip) of the adjacent cavity and emerging from either the same side or the opposite side of the leading-edge cavity and 3) flow entering from one side of the adjacent cavity and emerging from both sides of the leading-edge cavity. These flow arrangements were tested for five crossover-hole settings with a focus on studying the heat transfer rate dependency on the axial flow produced by upstream crossover holes (spent air). Numerical results were obtained from a three-dimensional unstructured computational fluid dynamics model with 1.1 million hexahedral elements. For turbulence modeling, the realizable  $k - \varepsilon$  was employed in combination with enhanced wall treatment approach for the near wall regions. Other avail-

able RANS turbulence models with similar computational cost did not produce any results in better agreement with the measured data. Nusselt numbers on the nose area and the pressure/suction sides are reported for jet Reynolds numbers ranging from 8000 to 55000 and a constant crossover hole to the leading-edge nose distance ratio,  $Z/D_h$ , of 2.81. Comparisons with experimental results were made in order to validate the employed turbulence model and the numericallyobtained results. Results show a significant dependency of Nusselt number on the axial flow introduced by upstream jets as it drastically diminishes the impingement effects on the leading-edge channel walls. Flow arrangement has immense effects on the heat transfer results. Discrepancies between the experimental and numerical results averaged between +0.3% and -24.5%, however correlation between the two can be clearly observed.

## Nomenclature

AR channel aspect ratio

AR<sub>rib</sub> rib aspect ratio

- $D_h$  racetrack hole hydraulic diameter (2.228 cm)
- *e* rib height
- *h* average heat transfer coefficient on the leading-edge or side walls,  $[(vi/A_{HT}) q_{loss}]/(T_s T_{jet})$
- *i* current through the foil heater on the middle copper piece
- *k* air thermal conductivity

 $\dot{m}_{total}$  total mass flow rate entering the supply channel

 $\dot{m}_5$  mass flow rate through the crossover hole number 5 in

Fig. 1

- $Nu_{jet}$  average Nusselt number based on the crossover hole diameter,  $hD_h/k$
- *P* each crossover hole perimeter, also rib and crossover hole pitch

*P<sub>amb</sub>* ambient pressure

*P<sub>supply</sub>* pressure in the supply channel

 $P_{LE}$  pressure in the leading-edge channel

- $q_{loss}$  heat losses from the middle copper piece to the ambient by conduction and convection as well as the heat losses by radiation to the unheated walls
- $Re_{jet}$  jet Reynolds number  $(4\dot{m}_5/P\mu)$
- $T_{jet}$  air jet temperature
- $T_s$  surface temperature
- Z jet plate distance from the leading-edge channel nose, Fig. 1
- *v* voltage drop across the foil heater on the middle copper piece
- $\alpha$  angle of attack
- $\mu$  air dynamic viscosity at jet temperature
- $\rho$  air density at jet temperature and pressure

# 1 Introduction

In modern gas turbine engines, elevated-temperature gases impose constraints in the design process of early stage airfoils. A two-prong approach to tackle this challenging issue is often carried out. The first prong is to develop modern alloys in parallel with the use of ceramic composites in order to withstand the severe thermal conditions to which the airfoils are subjected. The other prong, which will be the main topic of the current study, is to design novel cooling schemes applied externally through film cooling techniques and internally using a combination of convection and impingement cooling inside the airfoil cavities. This study focuses on the latter prong, particularly the impingement cooling in an airfoil leading edge where extreme thermal stresses often arise. The main objective behind this approach is to achieve the maximum heat removal rate attained by the least use of coolant mass flow rate. Cooling flow typically emerges from the tip side of the airfoil as it loops radially in a serpentine passage to remove heat convectively from different sections of the airfoil. Alternatively, the leading-edge coolant flow will be routed from the adjacent cavity through a series of crossover holes, cast along the partitions wall between the two adjacent cooling cavities. This consequently allows impingement cooling on particular parts of the inner surface of the leading edge. This paper reports on the leading-edge jet impingement and the dependency of the heat transfer coefficient on the number of crossover holes and the nature of flow arrangement. Three flow arrangements were examined in this study. The first flow arrangement simulates a design in which the coolant enters the adjacent cavity from one side of the airfoil (root or tip) and emerges from the opposite side of the leading-edge cavity. The second flow arrangement simulates a design in which the coolant enters the adjacent cavity from one side of the airfoil and emerges from the same

side of the leading-edge cavity. The third flow arrangement which may find applications in other hot sections allows for the ejection of the coolant flow from both sides of cooling channel. Another feature of the study was to investigate the effect of the axial flow, caused by the upstream crossover holes, on the heat transfer coefficient. This was achieved by varying the number of jets upstream of the target wall on which the heat transfer measurements were performed. Five different geometries, corresponding to five through nine crossover holes, were tested in the study. Each geometry represents a different axial to impingement mass flow rate ratio where measurements were taken. Lastly, impingement heat transfer coefficient variation along the the leading-edge cooling channel from the root to tip is also studied. Most of the data in open literature are for impingement cooling on flat surfaces that are smooth or rib-roughened. Our previous work (Taslim et al. [1]) studied the impingement heat transfer coefficient on the target wall under the crossover jet in the middle of the leading-edge channel. In this study, the effects of the axial flow created by the upstream jets ( spent air) on the impingement heat transfer coefficient as well as axial variation of the impingement heat transfer coefficient from the root to tip are investigated. Generally, experimental as well as numerical studies reported in the open literature deal with parameters affecting the behavior of the heat transfer coefficient. These parameters include the ratio of the distance between the jet and the target surface to the jet hydraulic diameter  $(Z/D_h)$ , the target surface condition (smooth or roughened), target surface curvature, presence of crossflow, number of rows of the holes, hole spacing and arrangement, shape of the holes, jet Reynolds number, etc. More data can be found in the work of Lytle and Webb [2] on the the local heat transfer characteristics of air jet impingement at nozzle-plate spacings less than one, Hollworth and Berry [3] on single jet impingement on a flat surface for Z/Dvalues of 1, 2 and 3, Narayanan et al. [4] on the impinging slot jet flow with two different nozzle-to-surface spacings of 3.5 and 0.5 nozzle exit hydraulic diameters, Glynn et al. [5] on jet impingement cooling with jet diameter varying from 0.5 mm to 1.5 mm and Z/D ranging from 0.5 to 6, San and Shiao [6] on the confined circular air jet impinging on a flat plate with Z/D ranging from 1 to 6, jet plate width-to-jet diameter ratios ranging from 4.17 to 41.7 and seven jet plate length-to-jet diameter varying from 5.5 to 166.7, Chupp et al. [7] on the measured heat transfer coefficients on a concave surface simulating the leading edge of an airfoil with no crossflow and a jet Reynolds number range of 1000 to 10000, Metzger et al. [8] on the heat trasnfer characteristic for jet array impingement with initial crossflow rate-to-total jet flow ratios ranging from 0 to 1, Kercher and Tabakoff [9] on the impingement on a surface under a perforated plate of multiple square arrays of round air jet with Z/D ranging from 1 to 4.8 and jet spacing from 3.1 to 12.5 diameter, Florschuetz, et al. [10-12] on the jet array impingement in the presence of crossflow for inline and staggered patterns with different channel heights, streamwise and spanwise hole spacings, Metzger and Bunker [13], Bunker and Metzger [14] on the cooled turbine airfoil leading-edge regions with

and without film coolant extraction, Van Treuren et al. [15] on the measurements of heat transfer coefficient beneath an array of impinging jets over a range of jet Reynolds numbers, Chang et al. [16] on the impingement heat transfer on smooth and rib-roughened surfaces within arrays of circular jets with Z/D ranging from 1.5 to 3, Huang et al. [17] on the heat transfer distribution for an array of in-line jet impigement with different crossflow orientations, Akella and Han [18] on the impingement cooling on ribbed walls in rotating conditions with rib height-to-channel hydraulic diameter and rib pitch-to-height values of 0.124 and 10 respectively, Rama Kumar and Prasad [19] on impingement on concave surface from a row of impinging jets with Reynolds number ranging from 5000 to 67800 and Z/D value of 1, 3 and 4 and Taslim et al. [20-23] on a series of airfoil leadingedge impingement through crossover holes, on smooth and roughened target surfaces with and without the presence of showerhead film holes along the airfoil leading edge. This study reports on numerical results in conjunction with experimental data produced by geometrically-similar test sections. The choice of the geometry is based on some existing airfoil design that are in operation. The nose and side areas of the leading-edge channel were divided equally into nine nose and side heated areas located under each crossover hole. Comparisons are made between measurement data and the numerically-obtained result.

#### 2 Test Section

Figure 1 shows schematically the test rig layout and the cross-sectional area, the crossover hole geometry and the heated walls. In order to evaluate the heat transfer coefficient at the target wall, an averaged heat flux value in addition to an averaged surface temperature should be measured. This was achieved by using the conventional technique of thermocouples in conjunction with heated walls. The heated walls where all experimental measurements were conducted consisted of nine removable machined copper pieces which were heated by foil heaters attached to the back of the pieces to provide uniform fluxes. A constant equilibrium temperature was set up during the course of the experiment by means of adjusting the ohmic power supplied to the foil heaters attached to each copper piece. The length of the test section was 91.44 cm. A circular wall simulating the leading-edge nose with an inner radius of 1.1 cm and an arc angle of 137° was made of an acrylic plastic with a 9.9 cm long recess in the middle to house the nine copper pieces. A flange on each side of the leading-edge channel facilitated the connection of the side walls to this wall. The circular recess along the inner radius with a depth of 3.2 mm and a length of 18.85 cm allowed the copper pieces to be fitted into the  $Lexan^R$  shell. A removable 2.54 cm thick jet plate corresponding to a  $Z/D_h$ value of 2.81 was made of an acrylic plastic to produce the impingement jets. Nine racetrack-shaped holes were drilled on the jet plate at a constant distance of 6.18 cm of each other (center-to-center). Using this jet plate, five crossover hole arrangements, shown in Fig. 2, were tested by blocking hole



Fig. 1. Schematic of the test rig.

numbers 1 through 4 depending on the tested geometry. The jet plate was attached and sealed to the channel wall in order to simulate the partition wall between the leading-edge and its adjacent cooling cavity in an airfoil. Nine removable copper plates were installed in the acrylic nose piece. This consequently improves the uniformity of the heat fluxes applied to the target plates due to the high conductivity of the copper. The jet emerged from the middle crossover hole ideally impinges on the center of the middle copper piece in the middle of the leading-edge channel where the jet issued from the neighboring holes (4,6) impinged on the adjacent plates as they acted as guard heaters. Custom-made thin etched-foil heaters with thickness of 0.2 mm were glued on the outer surface of each copper piece to provide the necessary heat flux. On the leading-edge channel nose and two side walls, three identical copper pieces, separated by a 1-mm-thick rubber insulator, were mounted next to each other. Heat transfer coefficients were measured on the middle pieces while the neighboring pieces acted as guard heaters to alleviate the effects of any conduction within the target surface. The test section wall temperature was adjusted to a desirable level by varying the ohmic power to these heaters. Six thermocouples were embedded in each of the three middle copper pieces with their beads close to the exposed surface. Three thermocouples were embedded in each guard copper piece. The average of the six thermocouple readings in the middle copper pieces which, at the most, only differed by a fraction of a degree, was used as the surface temperature in the data reduction software for the average heat transfer coefficient. A nominal equilibrium surface temperature of  $45^{\circ}C$ in addition to a jet temperature of  $20^{\circ}C$  were selected. This created a reasonable temperature difference of  $25^{\circ}C$  between the wall surface and air temperatures. AC power was sup-



Fig. 2. Tested corssover holes geometries.

plied to individual heaters through an existing power panel with individual Variacs for each heater. Typical amperage and voltage levels for each heater varied between 0.23-0.4 Amps to 20-45 Volts, respectively. Air properties were evaluated at jet temperature. The trapezoidal supply channel was formed by extending the leading-edge channel side walls, the jet plate and a 1.27 cm thick aluminum back plate as shown in Fig. 1. The end caps were fixed such that it was possible to control the flow and pressure in each channel, thus simulating many conditions that may occur in actual airfoil environments. Three flow arrangements, shown in Fig. 3, were tested. In what we call the "parallel" case, flow entered from the root side of the adjacent cavity and emerged from the tip side of the leading-edge cavity. In the "circular" case, flow entered from the root side of the adjacent cavity



Fig. 3. Schematic of the tested flow arrangements.

and emerged from the same side in the leading-edge cavity. In the third case, called the "1inlet-2outlet" case, flow entered from the root side of the adjacent cavity and emerged from both sides of the leading-edge cavity. Static pressure taps and thermocouples in each channel measured the pressure and temperature at different locations. The test sections were covered on all sides, by 5 cm thick glass wool insulation to minimize heat losses to the environment. Radiational heat loss from the heated wall to the unheated walls, heat losses from the copper pieces to the ambient air were taken into consideration when heat transfer coefficients were calculated. A contact micro-manometer with an accuracy of 0.025 mm of water column as well as a series of oil and mercury manometers measured the pressures and pressure differences between the static pressure taps mounted on both sides of the roughened section for each geometry. For all cases, a critical venture meter was used to measure the total air mass flow rate entering the supply channel. Experimental uncertainties in the local jet Reynolds number and heat transfer coefficients, following the method of Kline and McClintock [24], were 3% and 6% respectively.

# **3** Computational Models

The leading-edge section and its adjacent cavity are symmetric with respect to a plane cutting through the entire model axially. This advantage allowed for solving one half of the model which reduced the computational cost yet provided a good grid resolution. We compared the results of a typical case using two numerical models, one with the whole setup meshed and one with half of the setup meshed. Flow and heat transfer results were compared and showed no noticeable differences. The computational grid consisted of four different main parts – supply channel, crossover holes, inlet/outlet ports and the leading-edge section with preserving the same dimensions and geometrical parameters of the test rig in all parts. With three flow and five crossover jet arrangement, shown in Figs. 2 and 3, a total of fifteen numerical models were tested. Furthermore, each model was run for seven jet Reynolds numbers based on the mass flow rates through the middle crossover hole and fluid properties at the inlet. Acceptable grid independence was reached at a cell count of about 1.1 million with high mesh density toward the leading edge and significantly near the heated walls as shown in Fig. 4. The numerical grid contained unstructured hexahedral elements by its entirety. This feature combined with parallel orientation of the cells to the flow direction has the advantage of precisely computing the fluxes along the control volume faces, thus reducing the formation of the false diffusive fluxes in the domain. Great considerations were given to the target walls  $y^+$  values due to the heavy dependency on them exhibited by not only the heat transfer coefficient but all wall properties. Proper  $y^+$  values were obtained by cell refinements in the near-wall regions as shown in Fig. 5. This resulted in keeping the area-weighted average of the wall  $y^+$  values below five in all tests in compliance with the recommended value for resolving the viscous sub



Fig. 4. Numerical grid including nine crossover holes and inlet/outlet ports.

layer. The CFD analyses were performed by Fluent/UNS solver by  $Ansys^R$ , Inc., a pressure-correction based, multiblock multigrid, unstructured/adaptive solver. Selection of the turbulence model plays an important role in determining the accuracy of the solution. According to the physical nature of the problem at hand, two-equation RANS models were the most suitable for both accuracy and computational cost. Numerical experimentations showed that the realizable  $k - \varepsilon$  model produced the closest results in comparison with the experimental data. For reaching a closure in the near-

wall regions, the enhanced wall treatment method was employed to resolve the viscous affected regions, in particular, the target walls for which the  $y^+$  values were within the permitted limits. Computational work was performed on a PC with  $Intel^R$  core i7 and 8 parallel processors. For the entire range of the jet Reynolds number (10000-55000), convergence was reached after 8000 iterations with residual sums in the order of magnitude of  $1x10^{-5}$  for the continuity equation and  $1x10^{-8}$  for momentum, energy and turbulence parameters. A typical CPU and wall-clock time for a given geometry was about 84 and 9 hrs, respectively. Constant heat fluxes identical to those in the test runs were applied to nine side areas and nine nose areas. These areas were numbered from 1 to 9 where area 1 is the closest to the root side as shown in Fig. 1. The simulation provided a tool to better visualize the flow field without causing a disturbance often caused by measurement tools. Reporting the mass flow rates which flows through each crossover holes is another advantage which was not a feasible option to perform in the actual test rig. These data aided in correlating the heat transfer coefficient behavior with the impingement/axial mass flow rates at all target areas. The non-dimesionalized form of the



Fig. 5. Detailed mesh around the root end.

heat transfer coefficient i.e. Nusselt numbers were reported and plotted against the Reynolds number for experimental results. In the case of numerical results, as the heat transfer coefficient data were available for nine different positions, only the cases of middle Reynolds were reported for a matter of convenience and without any loss of generality.

# 4 Results and discussion

All tests had several common features. The measured heat transfer coefficients are for the same location in the middle of the leading-edge channel where the jet issued from the crossover hole number 5 impinged on the copper test piece in the middle of the test section. Therefore, in the case of nine crossover holes, there were four jets upstream of the measurements location while in the case of five crossover holes, there were no upstream jets. Heat losses from the middle copper pieces to the ambient were taken into consideration when impingement heat transfer coefficient was calculated. The jet Reynolds number is based on the mass flow rate to the crossover hole number 5 (Fig. 1) for which the heat transfer coefficient is measured. Depending on the number of crossover holes and the nature of flow arrangement, the share of each crossover holes from the total mass flow rate varies significantly. To determine this percentages, thorough analyses of the flow fields for the five crossover holes geometries were performed. The three-dimensional CFD models were run under the test boundary conditions. The results are discussed for each flow arrangement. Reported measured data are for area 5 in the middle of the leading-edge channel. However, the numerical heat transfer results are for the entire leading-edge channel from area 1 through 9 shown in Fig. 1. It should be noted that to distinguish between the measured data and numerically-obtained results, heat trasnfer data points are shown by symbols while the numerical results are shown by lines of different distinctions.

## 4.1 Parallel flow arrangement

Figure 6 is a representative display of the CFD-obtained results for the case of nine crossover holes with parallel flow arrangement. Pressure contours show a decrease in the positive Z direction in the leading-edge section while they show an opposite behavior in the supply channel which consequently creates an increasing pressure difference between the two channels in the positive Z direction. On the other hand, Z velocity (momentum) exhibits a constant decrease in the positive Z direction in the supply channel. These criteria resulted in the high negative velocity values in the Y direction at the 9th hole while it shows positive values through the 1st hole. Furthermore, temperature contours on the symmetry plane indicate the thermal boundary layer growth. It is also seen that the stagnation point prompted the formation of the thermal boundary layer under the 3rd hole. The flow percentages through the corssover holes for the parallel flow arrangement is shown in Fig. 7. A monotonic increase in mass flow percentage through the holes in the positive Z direction is common for all geometries. A significant variation from -7.6% to 26.7% is observed with the negative sign indicating a reversed flow in the case of nine crossover holes which can be explained by examining either the pressure or the Y-velocity contours in Fig. 6. The



Fig. 6. CFD results on the symmetry plane, parallel flow arrangement.

flow momentum in the supply channel and the pressure difference across the crossover holes are the dominant parameters for the distribution of the mass flow rate through the crossover holes. The horizontal lines represent the percentages of flow through each hole if the total flow was distributed uniformly. The percentages did not show any dependency on the Reynolds number for all flow arrangements and geometries. Therefore, only the results of one jet Reynolds number for each case are reported. The reversed flow anomalies in the case of nine-hole geometry can be explained by the significant variation of the pressure difference across the crossover holes and the momentum terms along the channels. Measured and numerically-obtained pressure ratios across the crossover holes ( $P_{supply}/P_{LE}$ ) and across the leading-edge



Fig. 7. Percentage of total flow through each crossover hole, parallel flow arrangement.



Fig. 8. Pressure ratios across the crossover holes and across the leading-edge channel for the parallel flow arrangement.

channel  $(P_{supply}/P_{amb})$  for this flow arrangement are shown in Fig. 8. These pressure ratios did not vary with the number of crossover holes. This behavior is explained by the fact that the exit ports represent the limiting opening area, much less than the total area of the crossover holes, even in the case of five crossover holes. Because the pressure ratios are independent of the number of crossover holes, they would not correlate with the jet Reynolds number. Therefore, for different crossover hole geometries, they are compared for the same amount of flow entering the supply channel. The cluster of hollow symbols on the lower side of the graph corresponds to the pressure ratios across the crossover holes while the upper cluster of solid symbols represents the pressure ratios across the entire leading-edge channel. Numerical results show the same behavior with a maximum difference of



Fig. 9. Comparison between the measured versus CFD heat transfer results on nose and side areas under crossover holes no. 5 for the parallel flow arrangement.

a fraction of a percent. Figure 9 makes a comparison between the measured data and numerically-acquired Nusselt number versus the local jet Reynolds number on area 5 (Fig. 1), in the middle of the leading-edge channel. A common expected feature can be detected in all cases which is the monotonic increase of the Nusselt number with the jet Reynolds number. Examining Fig. 9 gives a rise to a rigorous comparison of the Nusselt number on nose and side areas. A very high heat transfer coefficient on the nose area is exhibited for the 5-hole geometry when compared to the side area. The physical explanation for this behavior is that area 5 benefits from all but impingement effects since there are no upstream jets. As the number of holes increases, area 5 will experience the effect of upstream jets and the resulting axial flow. This axial flow, created by the upstream jets, may divert the



Fig. 10. CFD heat transfer results for the parallel flow arrangement with seven experimental data points for area 5.

fifth crossover jet and reduce its heat transfer effects. The crossflow, however, will brush against the side walls and increase the corresponding rate of heat removal. That is why we see a systematic decrease in the nose heat transfer coefficient and a systematic increase in the side heat transfer coefficient as the number of the upstream jets increases. With a percentage variation from -7.1% to -24.5%, the numerical results, for most cases, were less than the measured values although the trends for all cases were identical. One data point from Chupp et al.(1969) is also shown which is in a reasonable agreement with our data given the differences in both geometry and their low range of Reynolds numbers. As we will see shortly, cross-geometry variation for the parallel flow arrangement (Fig. 9) is much wider than the other two flow arrangements and this is confirmed both by test and numerically-obtained results.

Figure 10 shows the variation of the Nusselt number across the leading-edge channel (areas 1 to 9) on both nose



Fig. 11. CFD contours of Nusselt number for the five crossover hole geometries and the parallel flow arrangement.

and side areas for the parallel flow arrangement and for all tested geometries. The numerical results in this figure and the ensuing Figs. 17 and 22 correspond to the same amount of total coolant flow rate entering the supply channel. The numerical results for area 5 show the same trend as those of measured data points with a reasonable quantitative agreement. On the nose area, maximum and minimum differences of 14% and 6% are measured, respectively while the these numbers on the side area are 21% and 13%, respectively. Given the complexity of the geometry and flow structures, these differences fall in the applicable range in the early stage of the airfoil design process, particularly when parametric



Fig. 12. CFD streamlines for the five crossover hole geometries and the parallel flow arrangement.

studies are performed. The local peaks in Nusselt number occurring on nose areas 3, 4 and 5 correspond to the flow stagnation points where the impingement effects are dominant. These points are shown in Fig. 11 by the contours of Nusselt number and in Fig. 12 by the divided streamlines. Evidenced by Fig. 12, crossflow prevails for the remainder of the nose areas while a combination of cross and transverse flows are in effect for all side areas. The upper part of Fig. 10 shows the trend for the side areas with an increase in Nusselt number in the positive Z direction beyond the stagnation points for all geometries which is supported by the presence of a constantly strengthening crossflow (as new jets with higher percentages of mass flow rate are introduced) shown in Fig. 12. Figure 13 shows the iso-surfaces of



Fig. 13. Iso-surfaces of the Y and Z velocities.

the Y and Z velocities for the nine-hole geometry in a paral-

lel flow arrangement. This representative case clearly show the decreasing level of the crossover jet penetration into th leading-edge channel in the flow direction due to the presence of the crossflow, even though the percentage share c the crossover hole from the total mass flow rate is increasing in the flow direction. The reversed flow through the firs crossover hole, also referred to previously when the flow per centages were discussed in Fig. 7, is clear in the left sectio where Y-velocity iso-surfaces are depicted. Furthermore, th most effective impingement, performed by the third jet, i clearly evident.

### 4.2 Circular flow arrangement



Fig. 14. Percentage of total flow through each crossover hole, circular flow arrangement.

The flow percentages through the corssover holes for the circular flow arrangement is shown in Fig. 14. In this flow arrangement, the profile behaved differently from the parallel flow arrangement. For the 5-, 6-, and to some extent the 7-hole geometries there is a monotonic flow percentage decrease in the positive Z direction. For the 8and 9-hole geometries, a quasi bell-shaped curve can be noticed with a peak occurring at the 4th crossover hole. Measured and numerically-obtained pressure ratios across the crossover holes  $(P_{supply}/P_{LE})$  and across the leading-edge channel  $(P_{supply}/P_{amb})$  for this flow arrangement are shown in Fig. 15. Again, for the same reason given for the parallel flow arrangement, these pressure ratios did not vary with the number of crossover holes. The two distinct clusters of hollow and solid symbols on the lower and upper side of the graph correspond to the two pressure ratios. Numerical results show the same behavior with a maximum difference of a fraction of a percent. Figure 16 makes a comparison between the measured data and numerically-acquired Nusselt number versus the local jet Reynolds number on area 5, in the middle of the leading-edge channel. In the circular arrangement, the influence of the crossflow is much more pronounced since for all hole geometries, there exist four up-



Fig. 15. Pressure ratios across the crossover holes and across the leading-edge channel for the circular flow arrangement.

stream jets with various levels of flow percentage depending on the tested geometry. The strong crossflow gives rise to the heat transfer coefficient on the side walls. Also, as expected, the downstream jets play no role on the heat transfer coefficient of area 5 and that is why Nusselt numbers, both on the nose and side areas, correlate so well with local jet Reynolds number. Numerically-obtained heat transfer results, shown in Fig. 17, also confirm this conclusion. With a percentage variation from -4% to -13%, the numerical results are in a reasonably good agreement with the test data. The well-correlated data with the local jet Reynolds num-



Fig. 16. Comparison between the measured versus CFD heat transfer results on nose and side areas under crossover hole no. 5 for circular flow arrangement.



Fig. 17. Contours of Nusselt number in the circular flow arrangement for all tested geometries.

ber in this flow arrangement which is also confirmed by the numerical results is of great significance. Figure 18 shows the variation of the Nusselt number across the leading-edge channel (areas 1 to 9) on both nose and side areas for the circular flow arrangement and for all tested geometries. For 5and 6- hole geometries, the overall trend shows an increasing Nusselt number from areas 1 through 4 both on the nose and side areas. These areas do not benefit from any jet impingement. However, the returning flow creates a channel flow and gives rise to the convective heat transfer coefficient. Other geometries showed a consistent decrease in the side Nusselt number from areas 1 through 9 which is explained



Fig. 18. CFD heat transfer results for the circular flow arrangement with seven experimental data points for area 5.

by the fact that the amount of crossflow in the leading-edge channel is at its maximum on area 1 and at its minimum on area 9. In this flow arrangement, impingement in the absence of any upstream crossflow only occurs under the ninth hole and that explains the relatively high heat transfer coefficient on the ninth nose area. Again, any local peak in the nose heat transfer coefficient indicates the presence of an effective jet at that location. The numerical results for area 5 show the same trend as those of measured data points with a reasonable quantitative agreement. On the nose area, maximum and minimum differences of 21% and 6% are measured, respectively while the these numbers on the side area are 24% and 9%, respectively. Similar to the aforementioned discussion, the overall agreement is reasonable.

#### 4.3 1inlet-2outlet flow arrangement



Fig. 19. Percentage of total flow through each crossover hole, 1inlet-2outlet flow arrangement.

The flow percentages through the corssover holes for the 1inlet-2outlet flow arrangement is shown in Fig. 19. The 1inlet-2outlet flow arrangement, have a trend similar to that of the parallel flow arrangement. A monotonic increase in mass flow percentage through the holes in the positive Z direction is observed for 9-, 8- and 7-hole geometries while the 4- and 5-hole geometries do not show a significant change before the ninth hole. The negative value for the 9-hole geometry indicates a reversed flow through the first crossover hole. Measured and numerically-obtained pressure ratios across the crossover holes  $(P_{supply}/P_{LE})$  and across the leading-edge channel  $(P_{supply}/P_{amb})$  for this flow arrangement are shown in Fig. 20. Again, for the same reason given for the other flow arrangements, these pressure ratios did not vary with the number of crossover holes. The two distinct clusters of hollow and solid symbols on the lower and upper side of the graph correspond to the two pressure ratios. Numerical results show the same behavior with a maximum difference of a fraction of a percent. The heat transfer results, shown in Fig. 21, behave very similarly to the case of circular flow arrangement i.e. the heat transfer coefficients



Fig. 20. Pressure ratios across the crossover holes and across the leading-edge channel for the 1inlet-2outlet flow arrangement.

on the side area are much higher than those on the nose area. Again, this behavior can be explained by the flow structure. Impingement effects on the area 5 are weakened by the presence of crossflow which has a better chance to form due to the presence of one outlet on each side of the leading-edge channel. It can be seen that the highest Nuselt number is achieved for the 6- and 8-hole geometries for the side and nose areas, respectively. Different flow structures are responsible for the heat transfer behavior of different geometries. Numerically-obtained heat transfer results, shown in Fig. 22, also confirm the monotonic decrease of the Nusselt number on the target area under the crossover hole number 5 as the number of crossover holes increase. The numerical



Fig. 21. Comparison between the measured versus CFD heat transfer results on nose and side areas under crossover hole no. 5 for 1 inlet-2 outlet flow arrangement.

results for area 5 show the same trend as those of measured data points with a reasonable quantitative agreement. With a percentage variation from +0.3% to +17%, the numerical results, for most cases, were higher than the measured values although the trends for all cases were identical. Figure 23 shows the variation of the Nusselt number across the leading-edge channel (areas 1 to 9) on both nose and side areas for this flow arrangement and for all tested geometries. Since the flow is let out of the leading-edge channel from both ends, the stagnation point, where impinging jet is most effective on the nose area, for all geometries but 5-hole is located on area 6 where we see the peak Nusselt number on the nose area. Although the overall heat transfer coefficient did not change more than 10% across the geometries, the 5-hole geometry still was the most efficient with an overall Nusselt number of about 98.



Fig. 22. Contours of Nusselt number in the 1inlet-2outlet flow arrangement for all tested geometries.

Again, the well-correlated data with the local jet Reynolds number in this flow arrangement which is also confirmed by the numerical results is of great significance. For the 5-hole geometry, the nose-area peak is shifted to the seventh area, dictated by the flow structure. On these exact nosearea peaks, the side Nusselt numbers are at their minimum since they do not benefit from either a strong crossflow or upward action of the impinging jets. These peaks and valleys are also confirmed in Fig. 22 where the contours of Nusselt number for all geometries are shown. In closing, an overall average heat transfer coefficient analysis for the entire heat transfer surfaces (nose and side areas) was performed and a comparison was made to identify the most efficient geometry for each flow arrangement. The 9-hole geometry, in the parallel flow arrangement, produced the highest overall Nusselt number of about 78. Therefore, it is concluded that, unless certain locations require higher heat transfer coeffi-



Fig. 23. CFD heat transfer results for the 1inlet-2outlet flow arrangement with seven experimental data points for area no. 5.

cients, for the same amount of cooling flow, more crossover holes produce a higher overall heat transfer coefficient for the parallel flow arrangement. In the circular flow arrangement, however, the 5-hole geometry produced the highest overall Nusselt number of about 105. A physical explanation for this behavior is that, when the flow is in circular arrangement, a preferred design is to force the flow through fewer crossover holes, located towards the end of the supply channel to benefit from a few strong jets at that location and a strong crossflow elsewhere. For the 1inlet-2outlet flow arrangement, although the overall heat transfer coefficient did not change more than 10% across the geometries, the 5-hole geometry still was the most efficient with an overall Nusselt number of about 98.

#### 5 Conclusions

The major conclusions of this study were:

a) Crossflow produced by the upstream jets has an immense effect on the convective heat transfer coefficient as it impedes the impingement flow from reaching the nose areas. However, the side areas exhibited an improvement in heat removal rate resulting from the deflected jet flow.

b) The axial velocity field in the supply channel combined with the pressure distribution across the two channels are responsible for establishing the variation of the mass flow rate through each crossover hole.

c) There exist a significant variation in both nose and side Nusselt numbers along the leading-edge channel. The nature of this variation depends on the flow arrangement.

d) Highest Nusselt number values, both on the nose and side areas, were generally produced in the circular flow arrangement. In particular, the 5-hole geometry had a superior performance.

e) The numerical predictions of impingement heat transfer coefficients, using the realizable  $k - \varepsilon$  turbulence model in conjunction with the enhanced wall treatment, were in a reasonable agreement with the measured values. Therefore, CFD could be considered a viable tool in airfoil cooling circuit designs.

## 6 REFERENCES

[1] Taslim, M.E., Abdelrasoul, A., 2009, An Experimental and Numerical Investigation of Impingement Heat Transfer In Airfoils Leading-Edge Cooling Channel, Int. Symp. on Heat Transfer in Gas Turbine Systems.

[2] Lytle, D. and Webb, B.W., 1994, "Air Jet Impingement Heat Transfer at Low Nozzle-Plate Spacings," Int. J. Heat and Mass Transfer, Vol. 37, pp. 1687-1697.

[3] Hollworth, B.R. and Berry, R.D., 1977, "Heat Transfer From Arrays of Impinging Jets with Large Jet-to-Jet Spacing," ASME paper # 78-GT-117.

[4] Narayanan, V., Seyed-Yagoobi, J. and Page, R.H., 2004, "An Experimental Study of Fluid Mechanics and Heat Transfer in an Impinging Slot Jet Flow," Int. J. Heat and Mass Transfer, Vol. 47, pp. 1827-1845.

[5] Glynn, C., ODonovan, T. and Murray, D., 2005, "Jet Impingement Cooling" 9th UK National Heat Transfer Conference, Manchester, England.

[6] San, J. and Shiao, W., 2006, "Effects of Jet Plate Size and Plate Spacing on the Stagnation Nusselt Number for a Confined Circular Air Jet Impinging on a Flat Surface," Int. J. of Heat and Mass Transfer, Vol. 49, pp. 3477-3486.

[7] Chupp, R.E., Helms, H.E., McFadden, P.W., and Brown, T.R., 1969, Evaluation of Internal Heat Transfer Coefficients for Impingement Cooled Turbine Blades, J. Aircraft, Vol. 6, No. 1, pp. 203-208.

[8] Metzger, D.E., Yamashita, T. and Jenkins, C.W., 1969, Impingement Cooling of Concave Surfaces With Lines of Circular Air Jets, J. Engr. for Power, Vol. 93, No. 3, pp. 149-155.

[9] Kercher, D.M. and Tabakoff, W., 1970, Heat Transfer by a Square Array of Round Air Jets Impinging Perpendicular to a Flat Surface Including the Effect of Spent Air, J. Engr. for Power, Vol. 92, No. 1, pp. 73-82.

[10] Florschetz, L.W., Berry, R.A., and Metzger, D.E., 1980, Periodic Streamwise Variation of Heat Transfer Coefficients for Inline and Staggered of Circular Jets with Crossflow of Spent Air, J. Heat Transfer, Vol. 102, No. 1, pp. 132-137.

[11] Florschetz, L.W., Truman, C.R., and Metzger, D.E., 1981, Streamwise Flow and Heat Transfer Distribution for Jet Impingement with Crossflow, J. Heat Transfer, Vol. 103, No. 2, pp. 337-342.

[12] Florschetz, L.W., Metzger, D.E., Su, C.C., Isoda, Y., and Tseng,H.H., 1984, Heat Transfer Characteristics for Jet Arrays Impingement with Initial Crossflow, J. Heat Transfer, Vol. 106, No. 1, pp. 34-41.

[13] Metzger, D.E., and Bunker, R.S., 1990, Local Heat Transfer in Internally Cooled Turbine Airfoil Leading Edge Regions: Part I -Impingement Cooling Without Film Coolant Extraction, J. Turbomachinery, Vol. 112, No. 3, pp. 451-458.

[14] Bunker, R.S., and Metzger, D.E., 1990, Local Heat Transfer in Internally Cooled Turbine Airfoil Leading Edge Regions: Part II -Impingement Cooling With Film Coolant Extraction, J. Turbomachinery, Vol. 112, No. 3, pp. 459-466. [15] Van Treuren, K.W., Wang, Z., Ireland, P.T., and Jones, T.V., 1994, Detailed Measurements of Local Heat Transfer Coefficient and Adiabatic Wall Temperature Beneath an Array of Impinging Jets, J. of Turbomachinery, Vol. 116, No. 2, pp. 269-374.

[16] Chang, H., Zhang, D., and Huang, T., 1997, Impingement Heat Transfer from Rib Roughened Surface Within Arrays of Circular Jet : The Effect of the Relative Position of the jet Hole to the Ribs, Paper # 97-GT-331.

[17] Huang, Y., Ekkad, S.V., and Han, J.C., 1998, Detailed Heat Transfer Distributions Under an Array of Orthogonal Impinging Jets, J. Thermophysics and Heat Transfer, Vol. 12, No. 1, pp. 73-79.

[18] Akella, K.V. and Han, J.C., 1999, Impingement Cooling in Rotating Two-pass Rectangular Channels with Ribbed Walls, J. Thermophysics and Heat Transfer, Vol. 13, No. 3, pp. 364-371.

[19] Rama Kumar, B.V.N. and Prasad, B.V.S.S.S., 2006, Computational Investigation of Flow and Heat Transfer for a Row of Circular Jets Impinging on a Concave Surface, paper # GT2006-90851.

[20] Taslim, M.E., Pan, Y. and Spring, S.D., 2001, An Experimental Study of Impingement on Roughened Airfoil Leading-Walls with Film Holes, J. of Turbomachinery, Vol. 123, No. 4, pp. 766-773.

[21] Taslim, M.E. and Setayeshgar, L., 2001, Experimental Leading-Edge Impingement Cooling Through Racetrack Crossover Holes, Paper # 2001-GT-0153.

[22] Taslim, M.E., Setayeshgar, L. and Spring, S.D., 2001, An Experimental Evaluation of Advanced Leading Edge Impingement Cooling Concepts, J. of Turbomachinery, Vol. 123, No. 2, pp. 147-153.

[23] Taslim, M.E., Bakhtari, K., and Liu, H., 2003, Experimental and Numerical Investigation of Impingement on a Rib-Roughened Leading-Wall, J. of Turbomachinery, Vol. 125, pp. 1-10.

[24] Kline, S.J. and McClintock, F.A., 1953, Describing Uncertainty in Single-Sample Experiments, Mechanical Engineering, Vol. 75, Jan., pp. 3-8.