# GT2011-46143

# OPTIMIZED IMPINGEMENT CONFIGURATIONS FOR DOUBLE WALL COOLING APPLICATIONS

Preston Stoakes Virginia Tech Blacksburg, VA, USA Srinath Ekkad Virginia Tech Blacksburg, VA, USA

# ABSTRACT

Double wall cooling is a very effective technique for increasing heat transfer in hot gas path components utilizing a narrow channel near the surface of the component. Multiple techniques exist to increase the heat transfer within the narrow channel, including the use of impingement jets, turbulators and microchannels. A preliminary study has been performed using computational fluid dynamics (CFD) to determine the heat transfer benefits of double wall cooling technology when compared to a smooth wall square channel and a ribbed wall square channel. Conjugate CFD simulations of flow through an aluminum channel were performed to include the effects of conduction through the solid and convection within the main channel. The design for the preliminary study consists of a square main channel and a narrow impingement channel connected by a series of holes creating impingement jets on the outer surface of the impingement channel. The study examines multiple parameters to increase heat transfer without increasing the pumping power required. The parameters studied include diameter of impingement jets, jet-to-jet spacing, number of impingement jets, and jet-to-wall spacing. Results show that the impingement channel height-to-diameter ratio has a strong impact on heat transfer effectiveness. This study also provides a new optimization methodology for improving cooling designs with specific targets.

# INTRODUCTION

An effort to raise the operating efficiency of gas turbines has lead to higher gas inlet temperatures in the turbine section. The inlet gas temperature has increased beyond the thermal limits of the material used for the turbine blades and a need has developed for a more effective method of cooling the turbine blade material. The current study focuses on a double wall cooling design with impingement cooling to augment the heat transfer from the outer surface of turbine blades to present a novel optimization method that includes CFD evaluation and experimental validation.

A wide variety of literature discussing gas turbine cooling technology and impingement cooling can be found in Han et al. [1]. The following paragraphs contain a summary of the most relevant literature. Many previous studies have been performed on impingement cooling [2] - [8]. Obot and Trabold [2] studied impingement cooling on a flat plate with three exhaust configurations: maximum, intermediate and minimum cross flow. The test section used was a rectangular channel in which the exhaust could be directed

out one side (maximum cross flow), two sides (intermediate cross flow) or all sides (minimum cross flow). The authors varied jet-to-jet spacing, jet-to-wall spacing and jet Reynolds number and found that for all geometries, the maximum cross flow scheme created the lowest heat transfer due to jet-cross flow interaction while the minimum cross flow scheme created the highest heat transfer.

Gillespie et al. [3] examined double wall cooling with effusion holes using liquid crystal thermography. The jet Reynolds number was varied between 20,000 and 40,000. The authors found Nusselt number distribution on the target plate increases as Reynolds number increases. The authors were also able to calculate the Nusselt number distribution on the impingement plate. The Nusselt number on the impingement plate was found to be elevated near the jet.

Huang et al. [4] studied a square array of impingement jets in a confined impingement channel implementing the intermediate cross flow scheme and two maximum cross flow schemes. One maximum cross flow scheme allowed the air to exhaust in same direction as the inlet flow, while the other forced the air to exit in the opposite direction as the inlet flow. The results of this study are consistent with those found by Obot and Trabold [2]. The intermediate cross flow scheme. The maximum cross flow scheme in which the inlet and exhaust flows are in opposite directions provided the least heat transfer.

Chambers et al. [5] studied the effect of initial cross flow in an impingement channel. The experiments focused on the maximum cross flow scheme. The authors found that higher initial cross flow tends to create lower heat transfer near the entrance of the channel where the mass flow rate of coolant in the impingement jets is lower. However, towards the end of the channel the heat transfer for all cases become similar.

Xing et al. [6] performed both an experimental and numerical study on an array of impingement jets with a impingement channel height-to-plate spacing ratio (H/D) set to 3, 4 and 5. The study examined the Nusselt number (Nu) as a function of both Reynolds number (Re) and H/D for in line and staggered array of impingement jets. The authors confirmed the findings of Kercher and Tabakoff [9] showing Nu is a function of  $Re^{0.8}Pr^{1/3}$ .

Many of these previous studies have focused solely on heat transfer on the impingement surface. In the current study, an attempt is made to incorporate the cost of cooling (pumping power) into the evaluation of various impingement cooling designs. An optimization scheme is used to identify various important parameters that could be optimized to improve cooling effectiveness of impingement geometries. A secondary goal of the current study is to capture the effects of conduction heat transfer within the test section to evaluate actual heat transfer. Eventually, these channels will have complex patterns such as ribs, pin fins and/or dimples to improve the double wall cooling methodology.

# METHODOLOGY

The current study contains two major facets: a numerical optimization study and experimental validation. The numerical optimization is performed on a coarse grid to obtain a reliable solution without a large numerical expense. The heat transfer data from the numerical study is used only to establish trends.

Once the numerical optimization is completed, the best performing designs are selected for experimental study. The experimental study uses the lumped capacitance method to obtain local heat transfer coefficients on a metal test section in a transient test. Details of the experimental study will be discussed in more detail in a later section.

#### NUMERICAL SOLVER

The current study uses the commercial computation fluid dynamics (CFD) code ANSYS CFX through the ANSYS Workbench interface. The conjugate simulations used in this study simulated heat transfer in both the air and the aluminum test section. The SST k- $\omega$  turbulence model was chosen based on the work of Parida et al. [10]. The SIMPLE algorithm is used as for pressure-velocity coupling and a second order discretization method is used for all equations.

#### Geometry

The geometry used in the current study consists of a smooth, square main channel closed at one end with an array of impingement holes along one side. The impingement holes connected the main channel to a smooth, rectangular impingement channel. The impingement channel is closed at the opposite end as compared to the main channel. A drawing of the geometry can be found in Figs. 1 and 2. The geometry fits in the maximum cross flow scheme as described by Obot and Trabold [2]. The main channel is 2.54 x 2.54 cm<sup>2</sup> and the thickness (*t*) of all walls is 0.317-cm for all cases; however, the remaining dimensions were varied as described in the optimization section.

#### **Boundary Conditions**

Three types of boundary conditions are needed to complete the geometry definition. The open end of the main channel is assigned a velocity inlet boundary condition and the open end of the impingement channel is assigned an outlet boundary condition. The outside of the impingement wall has a heat flux of 10,000 W/m<sup>2</sup> applied to it while all other walls are constrained to be adiabatic.



FIGURE 1. 2D REPRESENTATION OF GEOMETRY USED FOR SIMULATIONS IN THE CURRENT STUDY

The velocity inlet requires three flow variables to be defined: flow direction, flow temperature and flow velocity. The flow direction is set to normal to the boundary at a temperature of 25°C. The flow velocity is dependent on the Reynolds number of the impingement jet ( $Re_{Jet}$ ) and variations in the geometry as described in the optimization section. The  $Re_{Jet}$  based on the impingement jet diameter was set to 10,000 for all cases. The inlet velocity (u) was calculated using Eqn. 1.

$$u = \frac{\operatorname{Re}_{J_{et}} \pi D N \mu}{4 \rho W^2} \tag{1}$$

where *D* is the diameter of the impingement jets,  $\mu$  is the kinetic viscosity of the fluid,  $\rho$  is the the density of the fluid, *W* is the width and height of the main channel and *N* is the number of impingement jets.

#### Mesh

A tetrahedral mesh was used throughout the geometry. After a grid independence study, it was found that a total of approximately 1.2 million cells provided enough resolution for accurate results. The mesh along the impingement wall was refined using an inflation scheme to produce a boundary layer mesh with a value of  $y^+$  less than unity.

### **OPTIMIZATION**

This section will describe the method used for optimizing the geometry, including which parameters were tested and which parameters were used to determine the optimum design.

#### **Independent Parameters**

The optimization scheme analyzed a total of four geometric parameters. The diameter (*D*) of the impingement hole was varied between 0.794 mm and 6.35 mm. The number impingement jet rows (*N*) was varied from five (5) to eleven (11). The other geometric parameters were non-dimensionalized using the hole diameter. The other parameters tested were the jet-to-wall spacing ratio (*H*/*D*), varied from one-half (0.5) to four (4) and jet-to-jet spacing ratio (*S*/*D*) which was varied from two (2) to five (5). A summary of independent parameters can be found in Table 1. Each independent parameter was given four set points. A total of 256 cases were examined numerically. When varying *S*/*D* or *N*, the total length of the test section was defined as 11(S)+25.4 mm.



FIGURE 2. 3D REPRESENTATION OF GEOMETRY USED FOR SIMULATIONS IN THE CURRENT STUDY

	Range		
Parameter	Min	Max	
D	0.794 mm	6.35 mm	
N	5	11	
H/D	1/2	4	
S/D	2	5	

TABLE 1. SUMMARY OF INDEPENDENT TEST PARAMETERS.

## **Dependent Parameters**

The suitability of each set of test parameters was evaluated based on two dependent parameters. The most important parameter examined is the average Nusselt number (Nu) on the impingement surface. Pumping power (P) required to pump the coolant through the test section was also selected and is defined by Eqn. 2.

$$P = Qdp$$

where Q is the volumetric flow rate of the coolant, calculated as shown in Eqn. 3 and dp is the pressure drop across the test section.

(2)

$$Q = uW^2 \tag{3}$$

# Determination of Optimal Design

The determination of the optimal design uses an effectiveness parameter which compares the heat transfer and pumping power of each design to a reference design using Eqn. 4. The reference design used for this study had 11 rows of 6.35-mm holes. The jet hole-to-jet hole spacing ratio was S/D = 2 and impingement channel height-to-wall spacing ratio was H/D = 0.5. All designs found to have an effectiveness parameter above 1 are considered to be better than the baseline design and all designs found to have an effectiveness parameter below 1 are considered to be worse. The use of pumping power instead of pressure drop across the test section allowed the volumetric flow rate for each design to be accounted for.

$$Eff = \frac{h_0}{\left(\frac{P}{P_0}\right)^{\frac{1}{3}}}$$
(4)

#### **RESULTS AND DISCUSSION**

After CFD simulations had been completed on all designs, the data was analyzed and compared as described in the optimization section. The results are presented below.

#### **Optimization Results**

A summary of effectiveness parameter for all designs considered in the optimization study is shown in Fig. 3. The effectiveness parameter was found to have a minimum value of 0.99 and a maximum value of 54.5 for the designs considered. The design with the highest effectiveness parameter had 5 rows of impingement jets with a diameter of 6.35 mm, a jet hole-to-jet hole spacing of 2 and impingement channel height-to-wall ratio of 4. This design however was rejected in favor of the second best design due to manufacturing and application space concerns. The second best design consists of 5 rows of impingement jets with a diameter of 3.175 mm, a jet hole-tojet hole spacing ratio of 5 and a impingement channel height-to-wall spacing ratio of 2 and was found to have an effectiveness parameter of 51.66. Results from the simulation of the second best design will be discussed in more detail in the following section. In general, fewer impingement jets were found to lead to a higher effectiveness parameter, this trend can likely be attributed to the lower volumetric flow rate of the coolant, which produces a lower pumping power.

A closer look at Nu shows that for each 10% reduction in rows of impingement jets results in approximately 10% reduction in average heat transfer. The effect of H/D on the effectiveness parameter has a different effect depending on the jet diameter. For larger jets, a higher value of H/D was found to increase the effectiveness coefficient. Here, the larger impingement channel should lead to a decrease in pressure drop across the test section, which would lead to lower pumping power. For smaller jets, the effectiveness parameter is found to be higher at H/D as low as unity.

The four best designs from the optimization study were selected to be tested experimentally. The parameters of these designs can be found in Table 2.

#### EXPERIMENTAL STUDY

The experimental test sections were constructed from aluminum to include the effects of conduction in the experiment. A lumped capacitance based transient experimental technique was used to determine the Nusselt numbers for the target plate. Detailed temperature data on the outer wall of the test plate was collected at 10 Hz using a FLIR SC-640 infrared camera. The test section was mounted on a test rig designed to provide steady and uniform flow from a compressed air line. A schematic of the test section and test rig is shown in Figure 4. Compressed and metered air enters from a 50-mm round pipe and expands to a 35.56 cm x 15.24 cm plenum. The plenum allows the flow to form a uniform velocity before passing through the mesh heater. The length of the plenum is 52-cm before the heater. At the exit of the plenum, a mesh heater constructed of 304stainless steel woven wire with a wire diameter of 20 microns is placed. The mesh heater used is similar to the heater presented by Esposito et al. [9]. A 5-cm spacer separates the mesh heater and the reduction nozzle leading to the test section to complete the test rig. Power is supplied to the mesh heater by a welding machine power source providing low voltage, high amperage DC power. The mesh heater allows the assumption of a true step change in mainstream temperature during the transient test. The heater reaches steady state temperature in less than 33 milliseconds and therefore a step change in temperature is assumed in the calculations. The heater temperature was measured using an IR camera with a 60 Hz frame rate. After capturing the images, the IR camera images showed that the temperature on the heater was uniform immediately at the first image which is 33 milliseconds after initiation of the heater and remained steady after that instant.

Experiments were conducted on each test section at three different average jet Reynolds numbers: 5,000, 10,000, and 15,000. Total flow was measured using an orifice meter. Pressure data was collected using three Omega PX-137 differential pressure transducers. Mainstream and reference surface temperature of the test section were measured with Type K thermocouples. Pressure and temperature data was collected using an Omega OMB-DAQ-54 data logger.

Transient temperature data was used to calculate the heat transfer coefficient using the lumped capacitance model, as shown in Eqn 5, where  $T_{ms}$  is the temperature of the mainstream flow,  $T_w$  is the temperature of the wall at time t,  $T_{w0}$  is the initial wall temperature,  $\rho$  is the density of the aluminum, C is the specific heat of the aluminum and l is the thickness of the target surface. The heat transfer coefficient was calculated at each time step and averaged from 10 to 20 seconds.



Eff: 0 5 10 15 20 25 30 35 40 45 50 55 FIGURE 3. EFFECTIVENESS RESULTS OF OPTIMIZATION STUDY. EACH PLOT SHOWS VARIATION OF DIAMETER AND NUMBER OF ROWS OF IMPINGEMENT JETS. EACH PLOT SHOWS A DIFFERENT VALUE OF H/D AND L/D.

# TABLE 2. TEST SECTION DIMENSIONS

Design	D (mm)	H/D	S/D	Ν
1	3.175	2	5	5
2	3.175	1	2	5
3	3.175	2	4	5
4	6.350	2	2	5

$$\frac{T_{ms} - T_w}{T_{ms} - T_{w,0}} = e^{-\left(\frac{h}{\rho Cl}\right)t}$$
(5)

The lumped capacitance method makes the assumption of a conduction time scale much smaller than the convection time scale, represented by the Biot (Bi) number in Eqn 6. This assumption was met in the experiments, with  $Bi \sim 0.001$ .

4



FIGURE 4. TEST RIG AND TEST SECTION



FIGURE 5. NUSSELT NUMBER ON IMPINGEMENT SURFACE WITH RE = 15,000 FOR (A) DESIGN 1, (B) DESIGN 2, (C) DESIGN 3 AND (D) DESIGN 4. PARALLEL LINES REPRESENT BOUNDS OF IMPINGEMENT CHANNEL. CIRCLES REPRESENT LOCATION OF IMPINGEMENT JETS

$$Bi = \frac{hl}{k} < 0.01 \tag{6}$$

Contours of Nusselt numbers for Designs 1-4 with Re = 15,000 are shown in Figure 5. The two parallel lines represent the span wise bounds of the impingement channel; no cooling is observed in the area outside of these lines. The circles denote the location of the impingement jets. The cooling in designs 1, 3 and 4 is concentrated under the impingement jet region while the cooling for design 2 is more uniform and seems slightly downstream. Design 4 has the highest Nusselt numbers throughout the target surface, although there is a drop off after the impingement region. The more uniform Nusselt number profile in design 2 is likely due to the low jet-to-target spacing ratio.

Spanwise average of Nusselt number within the impingement channel for all designs and Reynolds numbers are compared in Figure 6. In the figure, x/S = 0 is the location of the first impingement jet; the two vertical lines represent the impingement area. All cases show a maximum value of Nu near x/L = 2 which corresponds to the location of the third impingement jet. The spanwise results show the effect of Reynolds number and also the effect of cross-flow. Case 2 shows the least effect of crossflow with a much flatter Nusselt number distribution along the channel. Design 1 and 4 show stronger effect of cross-flow downstream of the 3<sup>rd</sup> hole.

To help explain the location of the maximum heat transfer, velocity data is extracted from the CFD study. Velocity vectors and contours of velocity for the designs at Re = 10,000 on a plane bisecting one row of jets are shown in Fig. 7.



FIGURE 6. SPAN WISE AVERAGE OF NUSSELT NUMBER FOR EXPERIMENTAL CASES: (A) DESIGN 1, (B) DESIGN 2, (C) DESIGN 3, (D) DESIGN 4. JETS ARE LOCATED IN THE AREA BETWEEN THE VERTICAL LINES.



FIGURE 7. CONTOURS OF VELOCITY FOR DESIGNS AT *RE* = 10,000 ON A PLANE BISECTING ONE ROW OF IMPINGEMENT JETS: (A) DESIGN 1, (B) DESIGN 2, (C) DESIGN 3, (D) DESIGN 4

The velocity contours show the velocity is nearly constant in each of the jet holes; however the velocity of the flow exiting the first two impingement holes is lower than the velocity exiting the latter impingement holes. The lower velocity exiting the first row of holes causes the lower heat transfer on the target plate at that location. At the last set of impingement jets, the cross flow of exhaust air is high, which deflects the jets away from the target plate. The flow from the third and fourth holes is not deflected as much by the cross flow, which allows for higher heat transfer at that location.

The velocity contours for design 2 in the upper right figure show a high velocity toward the exit of the impingement channel. The high velocity correlates with the lower jet-to-wall spacing ratio (H/D=2) for design 2, and helps explain the Nusselt number profile after the jet region in Figures 5b and 6b compared to the other designs. The local Nusselt number for design 2 does not drop as much as the other designs after the jet impingement region.

Design 4 with an average jet Reynolds number of 15,000 has the highest Nusselt number of all cases with a maximum value of 52.2. All designs show similar Nusselt number profile at an average jet Reynolds number of 5,000 and increasing Nusselt number as the Reynolds number increases for all cases, which is consistent with previous research. The Nusselt number throughout the length of the test section is most consistent for design 2 and for all the Re = 5,000 cases, the Nusselt number is highest near the exit for design 2. For the other two values of Reynolds number, design 4 has the highest Nusselt numbers near the exit.

To verify the CFD simulations, the mass flow rate through each jet was calculated using pressure data from the experiment. Total pressure was measured in the main channel and static pressure was measured in the impingement channel at each jet location. They were then used to calculate mass flow through each jet hole using Eq. 7 as described by Gritsch, et al. [11].

$$\dot{m} = C_D p_{mc} \left(\frac{p_{ic}}{p_{mc}}\right)^{(\kappa+1)/(2\kappa)} \sqrt{\frac{2\kappa}{(\kappa-1)RT}} \left(\left(\frac{p_{mc}}{p_{ic}}\right)^{(\kappa-1)/\kappa} - 1\right) \frac{\pi}{4} D^2$$
(7)

where  $C_D$  is the discharge coefficient,  $p_{mc}$  is the pressure in the main channel,  $p_{ic}$  is the pressure in the impingement channel, R is the specific gas constant, T is the temperature of the flow and  $\kappa$  is the ratio of specific heats. The mass flow data is used to calculate the cross flow to jet mass flux ratio  $(G_c/G_j)$  as well as the actual Reynolds number in each jet. For our hole geometry and flow Reynolds number range, the  $C_D$  was estimated to be around 0.8. The actual jet Reynolds number normalized by the nominal jet average Reynolds number is shown in Fig. 8. The actual jet Reynolds number stays within 25% of the nominal average value at each jet hole location for all designs and nominal average Reynolds numbers. The nominal average Reynolds number does not appear to have an effect on the distribution ratio, however the geometry does. The largest deviation from the nominal Reynolds number is about 25% and occurs in design 4. Comparing the profile of each design suggests diameter and impingement channel height to jet diameter ratio affect the distribution of mass flow more than jet to jet spacing ratio.

Figure 9 shows the mass flux ratio  $(G_{c}/G_{j})$  for each experimental case. As with the Reynolds number variation, the nominal Reynolds number does not affect the profile of the mass flux ratio.

#### **Effectiveness Parameter of Experimental Designs**

To compare with the optimization study, the effectiveness parameter for each experimental case was calculated. For the experimental designs,  $Nu_0$  and  $P_0$  are calculated using Eqns 8 – 10, where  $D_h$  is the hydraulic diameter of the main channel,  $Re_D$  is the Reynolds number based on  $D_h$ , *f* is the friction factor, *Pr* is the Prandtl number of air, and *L* is the total length of the test section. The effectiveness parameters for each case are shown in Fig. 10.

$$P_0 = Qdp = Q\left(f\frac{\rho u^2}{2D_h}L\right)$$
(8)

$$f = (0.790 \text{Re}_D - 1.64)^{-2}$$
(9)  
$$N_L = 2.66 + 0.0668 (D_h/L) \text{Re}_D \text{Pr}$$

$$Nu_0 = 3.66 + \frac{0.000 \, (M_h)^2 \, (M_D)^{1/2}}{1 + 0.04 [(D_h/L) \, \text{Re}_D \, \text{Pr}]^{2/3}}$$
(10)



FIGURE 8. RATIO OF ACTUAL REYNOLDS NUMBER TO AVERAGE REYNOLDS NUMBER AT EACH JET FOR (A) DESIGN 1, (B) DESIGN 2, (C) DESIGN 3 AND (D) DESIGN 4



FIGURE 9. CROSS FLOW TO JET MASS FLUX AT EACH JET FOR (A) DESIGN 1, (B) DESIGN 2, (C) DESIGN 3 AND (D) DESIGN 4



EXPERIMENTAL CASES

Design 4 at  $Re_D = 10,000$  and  $Re_D = 15,000$  are the only two cases with an effectiveness parameter above unity at 1.05 and 1.25, respectively. At  $Re_D = 5,000$ , the best design is design 1 with an effectiveness parameter of 0.51. The effectiveness parameter for all designs increase as the jet Reynolds number increases. Design 3 is the worst of all the designs, with the effectiveness parameter and all jet Reynolds numbers below 0.10. The levels are expected as impingement creates higher pressure drop as well as higher heat transfer coefficients. It is very difficult to compare channel flows to impingement geometry. It is important to note that the performance of design 3 did not match the optimization results; however, the remainder of the designs all performed at a similar level to what was predicted by the CFD optimization.

## **CONCLUSIONS AND FUTURE WORK**

A comprehensive optimization-CFD study of heat transfer in a confined impingement channel has been completed with a maximum crossflow scheme. Four independent parameters were studied simultaneously to perform the optimization. The results presented above show fewer impingement holes provides comparable cooling while using less coolant and with a lower pressure drop.

The four best test sections from the optimization study were studied experimentally using the transient lumped capacitance technique. At Re = 10,000 and Re = 15,000, the fourth best design in the numerical optimization performed the best. At Re = 5,000 the best design from the numerical optimization performed the best. For all four designs, the heat transfer and effectiveness parameter increased as Reynolds number increases. Design 3 did not perform as well as was predicted, and further study is needed to explain the underperformance.

Future work will include high fidelity CFD simulations that are capable of improved prediction of conjugate heat transfer. Additional designs will be tested to attempt to create a more uniform Nusselt number profile throughout the test section. The test sections will then be modified with complex turbulators and pin fin arrangements in the impingement channel to increase both the convective heat transfer in the impingement channel and the conduction into the main channel.

# NOMENCLATURE

- Bi Biot Number
- C Specific Heat
- D Diameter of impingement jets
- D<sub>h</sub> Hydraulic diameter of the main channel
- dp Pressure drop across test section
- f Friction Factor
- H Impingement channel height (jet-to-wall spacing)
- h Convective heat transfer coefficient
- κ Ratio of specific heats
- L Overall length of the test section
- 1 Thickness of target plate
- N Number of rows of impingement holes
- Nu Nusselt Number
- P Pumping Power
- p Pressure
- ρ Density
- Q Volumetric Flow Rate
- R Specific gas constant
- $Re_D$  Reynolds number based on  $D_h$
- $Re_{Jet}$  Reynolds number based on D
- S Distance between rows of impingement holes (jet hole-tojet hole spacing
- T<sub>ms</sub> Main stream air temperature
- T<sub>w</sub> Wall temperature
- t Thickness of walls, time
- u Main channel inlet velocity
- W Width and height of main channel
- μ Kinetic viscosity of working fluid

# ACKNOWLEDGMENTS

The authors would like to thank U.S. DOE-NETL for research funding. The work was funded through the Regional University Alliance (RUA) with NETL and URS Corp. The authors are thankful to Dr. Mary Anne Alvin of NETL and Dr. Vijay Jain of URS for their support.

# REFERENCES

[1] Han, J.C., S. Dutta, and S.V. Ekkad, 2000. *Gas Turbine Heat Transfer and Cooling Technology*. Taylor & Francis, New York.

[2] Obot, N., and Trabold, T., 1987. "Impingement heat transferwithin arrays of circular jets: Part 1 - effects of minimum, intermediate, and complete cross flow for small and large spacings". *Journal of Heat Transfer*, **109**, pp. 872–879.

[3] Gillespie, D.R.H., et al., 1998 "Full Surface Local Heat Transfer Coefficient Measurements in a Model of an Integrally Cast Impingement Cooling Geometry". *Journal of Turbomachinery*, **120**, pp. 92–99.

[4] Huang, Y., Ekkad, S., and Han, J., 1998. "Detailed heat transfer distributions under an array of orthogonal impinging jets". *Journal of Thermophysics and Heat Transfer*, **12**(1), January - March, pp. 73–79.

[5] Chambers, A., et al., 2005. "The effect of initial cross flow on the cooling performance of a narrow impingement channel". *Transaction of the ASME*, **127**, April, pp. 358–365.

[6] Goodro, M., et al., 2008. "Effects of hole spacing on spatiallyresolved jet array impingement heat transfer". *International Journal of Heat and Mass Transfer*, **51**, pp. 6243–6253.

[7] Hoberg, T.B., A.J. Onstad, and J.K. Eaton, 2009. "Heat transfer measurements for jet impingement arrays with local extraction". *International Journal of Heat and Fluid Flow*, **31**, pp. 460-467.

[8] Xing, Y., et al., 2010. "Experimental and Numerical Investigation of Heat Transfer Characteristics of Inline and Staggered Arrays of Impinging Jets". *Journal of Heat Transfer*, **132**(9), SEP.

[9] Kercher, D., 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". *ASME Journal Engineering for Power*, **92**, pp. 73–82.

[10] Parida, P., Ekkad, S., and Ngo, K., 2010. "Innovative liquid cooling configurations for high heat flux applications". In 12th ITherm Conference, Las Vegas, NV, USA.

[11] Gritsch, M., Schulz, A., Wittig, S. 1998. "Discharge Coefficient Measurements of Film-Cooling Holes with Expanded Exits". *Journal of Turbomachinery*, **120**, pp. 557 – 563.