JOURNAL OF AIRCRAFT
Vol. 46, No. 2, March–April 2009

Engineering Notes

ENGINEERING NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes should not exceed 2500 words (where a figure or table counts as 200 words). Following informal review by the Editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

Simulation of Heat Transfer from Hot-Air Jets Impinging a Three-Dimensional Concave Surface

Mathieu Fregeau,* Mohammad Gabr,† and Ion Paraschivoiu‡
Ecole Polytechnique de Montreal, Montreal, Quebec H3C 3A7, Canada
and
Farooq Saeed§
King Fahd University of Petroleum and Minerals, Dhahran 31261, Saudi Arabia

DOI: 10.2514/1.39846

Nomenclature

a, b	=	coefficients
d	=	piccolo-hole (jet) diameter
e	=	eccentricity of a conical curve
H	=	nozzle-to-surface distance
h	=	heat-transfer coefficient
k	=	thermal conductivity
M	=	Mach number
Nu	=	Nusselt number based on the hole diameter, hd/k
n	=	empirical coefficient
p	=	focal point of a conical curve
$\dot{q}$	=	heat flux
Re_{jet}	=	Reynolds number based on jet diameter and mean jet velocity, $\bar{u}d/\nu$
s	=	coordinate along the surface with its origin aft of the center of the jet axis, $y = 0$ plane
T	=	temperature, K
u	=	velocity along the jet axis
$\bar{u}$	=	mean jet velocity
W	=	nozzle-to-nozzle distance
x, y, z	=	coordinate system with its origin aft of the center of the jet exit
μ	=	dynamic viscosity
ν	=	kinematic viscosity, μ/ρ
θ	=	orientation angle of the jet
ρ	=	fluid density

Subscripts:

anti	=	from the anti-icing system
ave	=	averaged
jet	=	at the exit of the piccolo tube (jet condition)
max	=	at the maximum point of the indexed variable
min	=	at the minimum point of the indexed variable
s	=	at the surface

Received 17 July 2008; revision received 16 December 2008; accepted for publication 7 December 2008.

*Graduate Research Student, Departement de Genie Mecanique C.P. 6079, Succursale Centre-Ville,
†Graduate Research Student, Departement de Genie Mecanique C.P. 6079, Succursale Centre-Ville,
‡Bombardier Aeronautical Chair Professor, Departement de Genie Mecanique C.P. 6079, Succursale Centre-Ville. Senior Fellow AIAA.
§Assistant Professor, Mail Box 1637; farooqs@kfupm.edu.sa. Lifetime Member AIAA (Corresponding Author).

Introduction

TO ENHANCE flight safety under natural icing conditions, one of the several key tasks outlined in the Federal Aviation Administration (FAA) In-Flight Aircraft Icing Plan is to ensure the validity and reliability of icing simulation and modeling methods currently being used/developed [1]. In an effort to support the objectives of the FAA Icing Plan and to facilitate Bombardier Aerospace in the certification process, the main focus of research under the J.-A. Bombardier Aeronautical Chair at Ecole Polytechnique, Montreal, has been the development of a reliable ice-accretion and anti-icing simulation code CANICE [2–6]. The development of CANICE has been geared toward the specific needs of Bombardier Aerospace. In particular, the anti-icing simulation is modeled after what is most commonly used on the Bombardier Aerospace regional jets: a hot-air anti-icing system. The anti-icing system uses hot air from the engine compressor bleed. A system of external mounted ice detectors, with a sensing probe oscillating with a set frequency that decreases as ice accumulates on the surface, acts as a warning system.

A basic model for a hot-air anti-icing system is being used in CANICE. In this model, hot air from the engine is assumed to impinge upon the inner surface of the airfoil leading edge, or the slat (Fig. 1) in the case of a multi-element configuration. The inner region (region Ω in Fig. 1) is then modeled with a local internal convection coefficient h_{anti} that is known a priori. The heat flux $\dot{q}_{\text{anti}}$ coming from region Ω is then evaluated with the help of the jet temperature T_{jet} and the local-surface temperature T_s:

$$\dot{q}_{\text{anti}} = h_{\text{anti}}(T_{\text{jet}} - T_s) \tag{1}$$

A limitation of this method is that the internal heat flux $\dot{q}_{\text{anti}}$ or the convection coefficient h_{anti} and temperature T_{jet} are based on empirical relations for a hot-air jet impinging on a flat plate [7,8]. A good review of such empirical relations is found in [9]. Note that this local distribution of internal heat flux or the convection coefficient h_{anti} is purely based on the local distribution on a flat plate and therefore does not account for the curvature of the internal wall region Ω of an airfoil leading-edge or the wing slat. Another limitation of such a model is that it fails to provide an accurate estimate of the hot-air flux requirements and the drain on the engine power as a result of operating the hot-air anti-icing system.

To address these limitations, a more rigorous treatment of the internal hot-air region Ω is needed. A review of literature reveals that only a few experimental and theoretical/numerical studies have been carried out to study the heat-transfer mechanism in the internal hot-air region Ω [9–12]. Moreover, these studies have focused on

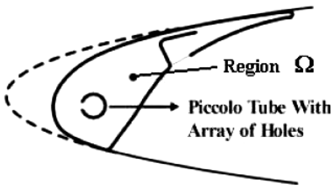


Fig. 1 RAE 2822 airfoil with a modified leading edge to incorporate a typical slat.

specific concerns that neither address the issues related to the design of a hot-air anti-icing system nor highlight variables that might play an important part in an optimum design of such a system. More recently, interest in heat transfer through curved surfaces has gained much attention, as reflected in numerical studies [13–16]. An evaluation of the jet impingement heat-transfer correlations for piccolo-tube application was also carried out by various investigators [13,15].

Hence, a need for an in-depth analysis of a hot-air anti-icing system becomes imperative. The use of computational fluid dynamics (CFD) tools to model the internal hot-air region in conjunction with a 3-D ice-accretion code requires extensive computational resources, and therefore these tools are not cost-efficient to accomplish anti-icing simulation or optimum design of hot-air anti-icing systems for aircraft wings. One way to avoid such a large computational overhead and make the anti-icing simulation more efficient is to employ numerical correlations for the range of operation of the anti-icing system. These tools can be used to develop numerical correlations for internal heat transfer from a single or an array of hot-air jets impinging on the internal surface of a typical wing slat. Moreover, numerical simulations could also be used to determine the effect of various important variables that can play an important role in the simulation and design of such systems, such as nozzle-to-surface height H, nozzle-to-nozzle spacing W, arrangement of hot-air jets, hot-air jet orientation θ, optimum height and spacing H/W, temperature of the jet T_{jet}, external surface temperature distribution T_s, Reynolds number of the jet, and diameter of the jet.

The different variables that can influence the heat transfer in terms of the surface Nusselt number distribution produced by a linear array of hot-air jets impinging on a 3-D concave surface can be expressed as follows:

$$Nu = f\left(\frac{s}{d}, \frac{y}{d}, M_{\text{jet}}, \frac{H}{d}, \frac{W}{d}, Re_{\text{jet}}, T_{\text{jet}}, T_s, e, p, \theta\right) \tag{2}$$

where e and p are related to a conical curve of the general form:

$$y^2 + (1 - e^2)x^2 - 2px + p^2 = 0 \tag{3}$$

Relation (2) implies that for a complete study, a total of at least $3^{10} = 6561$ cases should be analyzed to obtain three data points for every variation, assuming second-order correlations. To simplify the task, only the most important variables were selected for the present parametric study. The choice of these selected variables was based on a survey of previous numerical/experimental studies. The fixed parameters are listed in Table 1. This reduces surface Nusselt number distribution dependence to 5 parameters:

$$Nu = f\left(\frac{s}{d}, \frac{y}{d}, M_{\text{jet}}, \frac{H}{d}, \frac{W}{d}\right) \tag{4}$$

According to Brown et al. [10], hole diameter plays an important role in heat-transfer performance of anti-icing systems and thus should be considered for future study. Table 2 summarizes the different values of parameters H/d, W/d, and M_{jet} used in this study. These values are based on existing wing leading edges and piccolo-tube dimensions.

Although it is desirable to have choked flow (i.e., $M = 1$) at the jet exit, the jet Mach number was restricted to 0.8 because of numerical stability problems encountered in choked-flow cases. Nine geometries were modeled and a total of 27 cases were studied.

Table 1 Fixed parameters for the study

Variables	Values
Piccolo-hole diameter d	2.5 mm
Jet orientation	Horizontal
Curve shape	Curve shape[a]
Piccolo-tube diameter	$20 \times d$, or 50 mm ($\approx$2 in.)
Internal configuration	No baffles
T_{jet}	Constant at 400 K
T_s	Constant at 260 K
Impinging surface material	$k = 202.4\ \text{W/m}\cdot\text{K}$[b], 2 mm thick
Piccolo material	$k = 202.4\ \text{W/m}\cdot\text{K}$, 7 mm thick

[a]The center of the piccolo tube matches with the center of the circle defining the curved surface.
[b]Aluminum.

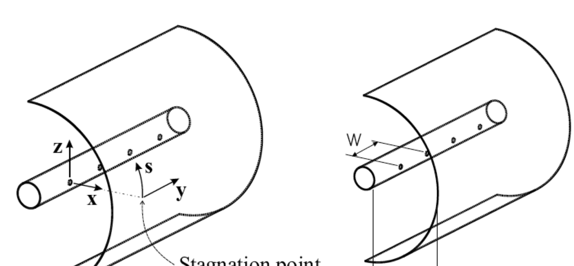
Table 2 Geometric characteristics and operating conditions used in this study

Variables	Values		
Jet Mach number M_{jet}	0.4	0.6	0.8
Jet height-to-diameter ratio H/d	5	10	15
Jet spacing-to-diameter ratio W/d	7.5	15	22.5

CFD Modeling

The state-of-the-art commercial CFD software FLUENT has been used in this study. The software comes with the mesh generator GAMBIT for modeling and meshing geometry. Fluent is extensively used for research and development by educational and research institutions and industry and has proven to be reliable in predicting CFD results. Figure 2a shows the coordinate reference frame used for the modeling, and Fig. 2b outlines the geometric parameters used in the anti-icing model of this study. No complex geometries such as baffles and reduced outlet area were taken into account. A more detailed analysis should, however, include such details for a more accurate assessment of heat transfer. A single array of hot-air jet holes on the piccolo tube was selected for the choice of geometry.

To take advantage of the simplicity of the model and to reduce the size of computational domain, one-quarter of the jet along with the domain was modeled and symmetrical boundary conditions were applied to simulate an array of the jets. Table 3 outlines the different boundary conditions shown in Fig. 3. A boundary-layer grid with a first cell dimension of 3×10^{-5} was used along the impingement surface, as well as the piccolo-tube surface, to reduce numerical error and to capture large gradients near the walls. Figure 3 also shows the structured mesh generated for use in the study. The mesh within the jet core was refined to ensure a good capture of gradients and numerical stability. Figure 4 shows the details of the mesh refinement. A grid dependency study was carried out to find the best



a) b)

Fig. 2 Illustrations of the a) coordinate system used for the anti-icing system modeling and b) geometric parameters used for the singular array of the jets configuration.

Table 3 Boundary conditions

Boundary name	Boundary-type specification
Outlet (refer to Fig. 3)	Pressure outlet
Inlet (refer to Fig. 3)	Velocity/mass flow rate
Impact surface (refer to Fig. 3)	Wall
Piccolo tube (refer to Fig. 3)	Wall
Symmetrical surfaces (refer to Fig. 3)	Symmetry
Internal surfaces (refer to Fig. 4)	Interior

geometry that would give reliable and stable results. The smallest geometry consisted of 25,000 hexahedral elements and the largest consisted of 60,000 elements. The coarse mesh demonstrated superior stability in all cases.

An energy equation was used to include heat-transfer effects. The one-equation Spalart-Allmaras (S-A) turbulence model was used along with the wall-functions approach. A weakness of the S-A model is its sensitivity to pressure gradients, which could affect the accuracy of results [17]. However, this model is generally consistent with the flow. A coupled solver was preferred, as it solves equations in a vector form. An ideal-gas law was used to take into account compressibility effects. Some computation models using a second-order discretization scheme were found to be unstable at high jet Mach numbers. All cases were thus performed using a first-order discretization scheme. Figure 5 compares results from first- and second-order spatial discretization schemes. Although the peak Nusselt number of the first-order model is higher than that of the second-order model by about 5%, a similar trend was observed for all cases. Many problems had been encountered regarding numerical stability at high Mach numbers. To avoid misrepresenting the physical problem by using a constant velocity profile at the inlet and to ensure numerical consistency, a turbulent velocity profile model was used at the inlet following Munson and et al. [18] formulation, given by

$$\frac{u}{u_{\max}} = \left(1 - \frac{2y}{d}\right)^{\frac{1}{n}} \tag{5}$$

where

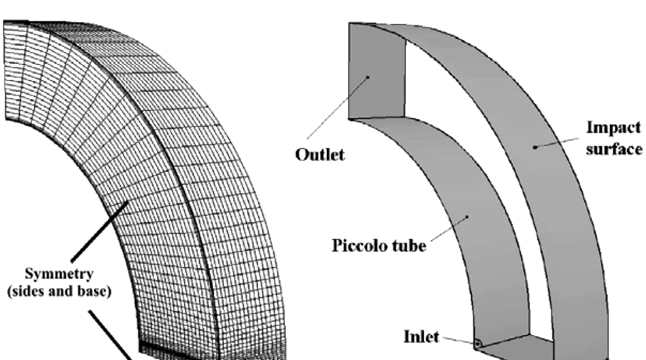
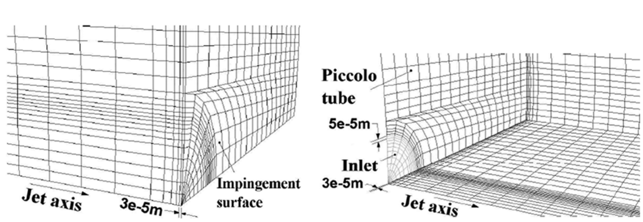


Fig. 3 Structured mesh and boundary conditions representing the CFD model.



a) b)

Fig. 4 Mesh refinement near the a) impingement surface (exterior view) and b) inlet (interior view).

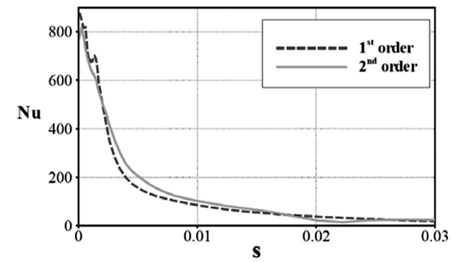


Fig. 5 Comparison of results of the first- and second-order discretization schemes.

$$\frac{u}{u_{\max}} = \frac{(1+n)(2n+1)}{2n^2}\bar{U}; \qquad n = 2.169 Re_{\mathrm{jet}}^{0.103} \tag{6}$$

The Reynolds number used in Eq. (6) is based on jet diameter and mean jet velocity, as $\bar{u}d/\nu$. The preceding relation is valid for $10^4 < Re_{\mathrm{jet}} < 2 \times 10^6$. Typical run time for each case was about 20 h on a 1 GHz RS6000 machine.

Validation Case

Literature dealing with hot-air jets impinging on a surface is relatively limited [7,9,10,19–22]. Most of those studies deal with flat plates as impingement surfaces, 2-D problems, or small-scale turbine-blade cooling. Consequently, those studies cannot be directly compared with larger geometries such as the leading edge of an aircraft wing. However, to establish the validity of our CFD model, the Gardon and Cobonpue [20] study has been used for validation of the CFD model. This experimental study uses a flat plate as the impingement surface. A simple 3-D model of the linear array of the jets impinging on a flat plate was examined. A round-type piccolo tube was considered to ensure similarity with their study. A summary of input conditions for the validation case is listed in Table 4. The boundary-layer mesh near the exit hole was built to have a maximum value of y^+ below 1 on all wall surfaces. Figure 6 shows the CFD model and mesh used for the validation case. A good agreement between the predicted local-surface Nusselt number distribution and the experimental results of Gardon and Cobonpue [20] is found, as shown in Fig. 7. The first 5% of s/d shows higher heat transfer predicted by the numerical results, which can be attributed to the use of a first-order spatial discretization scheme (see Fig. 5).

Conclusions

Figure 8 shows a significant increase in local Nusselt number distribution along s axis with a decrease in jet-to-surface distance H/d. An increase in local Nusselt number distribution along the s axis with jet-exit Mach number and jet-to-jet spacing W/d is observed (Fig. 9). Figure 10 shows a significant decrease in local Nusselt number distribution along the y axis, with an increase in jet-to-surface distance H/d. An increase in local Nusselt number distribution along the y axis with jet-exit Mach number and jet-to-jet spacing W/d is also observed (Fig. 11), which suggests that a larger spanwise spacing of the jets will tend to increase surface heat transfer. In the majority of cases, a trough in the local Nusselt number distribution near the jet centerline was observed. The size of this trough is found to decrease with an increase in M_{jet} and a decrease in

Table 4 Conditions used for the validation test case

Variable	Value
Jet Mach number, M_{jet}	0.4
Jet height-to-diameter ratio, H/d	6
Jet spacing-to-diameter ratio, W/d	20
Piccolo-hole diameter, d	6.35 mm
$\Delta T = T_{\mathrm{jet}} - T_s$	20 K

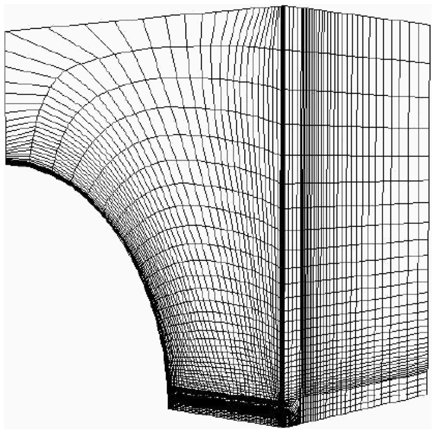


Fig. 6 Mesh used for the validation case.

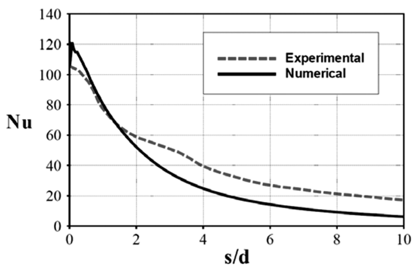


Fig. 7 Comparison of heat-transfer distribution from a rectangular array of the jets impinging on a flat plate: experiment vs numerical study for validation purposes.

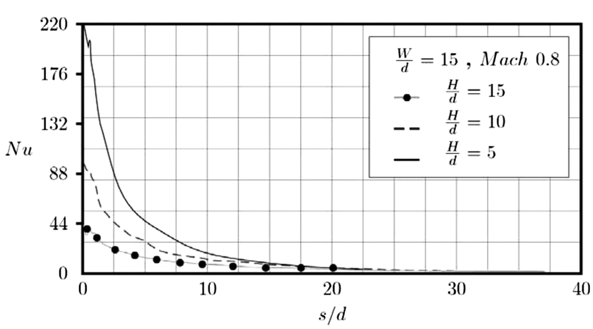


Fig. 8 Effect of H/d on Nusselt number along s.

H/d. The trough appears when the potential core of the jet is far from the impingement surface.

In all the cases studied, the Nusselt number decreases along the s axis much faster than along the y axis. The shape of a linear array will maintain higher energy between jets along the array, as the flow momentum tends to dissipate slowly. On the other hand, along the direction perpendicular to the array, the flow momentum is seen to dissipate quickly, as does the convection, leading to a fast decrease in heat transfer. The current study, however, did not attempt to find the optimum jet-to-surface H/d and jet-to-jet W/d spacing. It is speculated that the use of smaller H is likely to be more effective than increasing jet Mach number or the number of the jets (i.e., a smaller value of W). An ideal value of W/d would be the one that will preserve the flow at a high turbulence level between jets and sustain high level of convection.

Acknowledgments

The authors would like to acknowledge the support of the Natural Sciences and Engineering Research Council, Canada, through a cooperative research and development grant with Bombardier Aerospace. Helpful discussions with Fassi Kafyeke and his team at the Advanced Aerodynamics Department, Bombardier Aerospace, are gratefully acknowledged. The authors would also like to acknowledge the support of King Fahd University of Petroleum and Minerals for accomplishing this study.

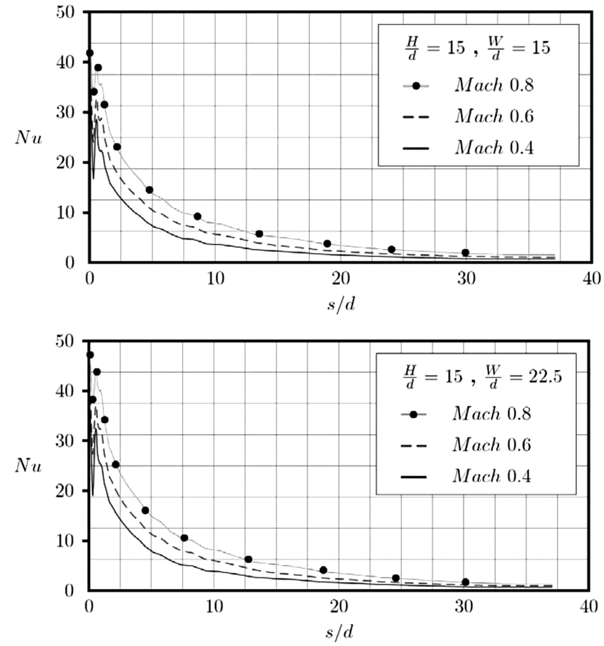


Fig. 9 Effect of Mach number and W/d on Nusselt number along s.

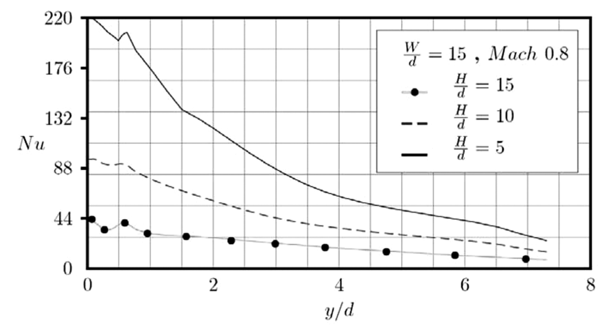


Fig. 10 Effect of H/d on Nusselt number along y.

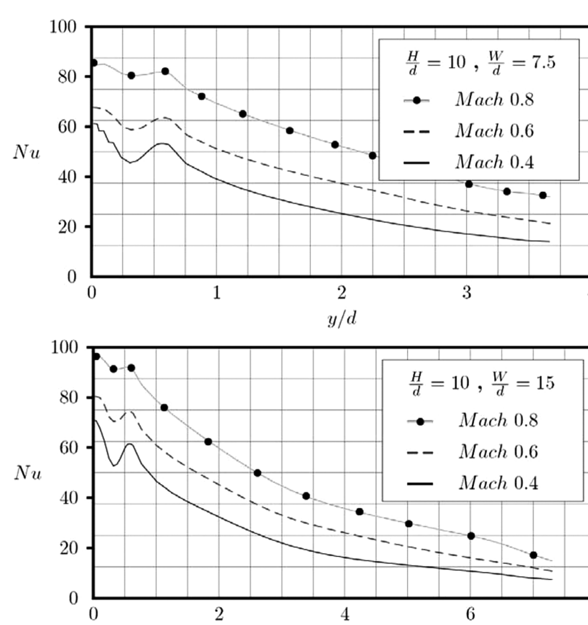


Fig. 11 Effect of Mach number and W/d on Nusselt number along y.

References

[1] "FAA Inflight Aircraft Icing Plan," U.S. Department of Transportation, Federal Aviation Administration, Apr. 1997, http://www.faa.gov/aircraft/air_cert/design_approvals/transport/media/Inflight_Icing_-Plan.pdf [retrieved Jan. 2008].

[2] Morency, F., Tezok, F., and Paraschivoiu, I., "Anti-Icing System Simulation Using Canice," *Journal of Aircraft*, Vol. 36, No. 6, Nov.–Dec. 1999, pp. 999–1006. doi:10.2514/2.2541

[3] Morency, F., Tezok, F., and Paraschivoiu, I., "Heat and Mass Transfer in the Case of an Anti-Icing System Modelisation," *Journal of Aircraft*, Vol. 37, No. 2, Mar.–Apr. 2000, pp. 245–252. doi:10.2514/2.2613

[4] Tran, P., Brahimi, M. T., Paraschivoiu, I., Pueyo, A., and Tezok, F., "Ice Accretion on Aircraft Wings with Thermodynamic Effects," *Journal of Aircraft*, Vol. 32, No. 2, 1995, pp. 444–446. doi:10.2514/3.46737

[5] Paraschivoiu, I., Tran, P., and Brahimi, M. T., "Prediction of the Ice Accretion with Viscous Effects on Aircraft Wings," *Journal of Aircraft*, Vol. 31, No. 4, July–Aug. 1994, pp. 855–861. doi:10.2514/3.46571

[6] Tran, P., Brahimi, M. T., and Paraschivoiu, I., "Ice Accretion on Aircraft Wings," *Canadian Aeronautics and Space Journal*, Vol. 40, No. 3, Sept. 1994, pp. 185–192.

[7] Saeed, F., and Paraschivoiu, I., "Numerical Correlation for Local Nusselt Number Distribution for Hot-Air Jet Impingement on Concave Surfaces," *Proceedings of the 8th Annual Conference of the CFD Society of Canada (CFD2K)*, Vol. 2, June 2000, pp. 897–904.

[8] Saeed, F., Morency, F., and Paraschivoiu, I., "Numerical Simulation of a Hot-Air Anti-Icing Simulation," *38th Aerospace Sciences Meeting and Exhibit*, Vol. 2, AIAA Paper 2000-0630, Jan. 2000, pp. 897–904.

[9] Martin, H., "Heat and Mass Transfer between Impinging Gas Jets and Solid Surfaces," *Advances in Heat Transfer*, Vol. 13, Academic Press, New York, 1977, pp. 1–60.

[10] Brown, J. M., Raghunathan, S., Watterson, J. K., Linton, A. J., and Riordon, D., "Heat Transfer Correlation for Anti-Icing Systems," *Journal of Aircraft*, Vol. 39, No. 1, Jan.–Feb. 2002, pp. 65–70. doi:10.2514/2.2896

[11] Meola, C., Carlomagno, G. M., Riegel, E., and Salvato, F., "An Experimental Study of an Anti-Icing Hot-Air Spray-Tube System," *ICAS Proceedings 1994: 19th Congress of the International Council of the Aeronautical Sciences*, AIAA, Washington, D.C., Sept. 1994, p. 63.

[12] Croce, G., Habashi, W. G., Guevremont, G., and Tezok, F., "3-D Thermal Analysis of an Anti-Icing Device Using FENSAP-ICE," AIAA Paper 98-0198, Jan. 1998.

[13] Tawfek, A. A., "Heat Transfer Studies of the Oblique Impingement of Round Jets upon a Curved Surface," *Heat and Mass Transfer*, Vol. 38, No. 6, 2002, pp. 467–75. doi:10.1007/s002310100221

[14] Wright, W. B., "An Evaluation of the Jet Impingement Heat Transfer Correlations for Piccolo Tube Application," NASA CR 212917, Apr. 2004; also AIAA Paper 2004-0062, Jan. 2004.

[15] Fregeau, M., Saeed, F., and Paraschivoiu, I., "Numerical Heat Transfer Correlation for Array of Hot-Air Jets Impinging on 3-Dimensional Concave Surface," *Journal of Aircraft*, Vol. 42, No. 3, 2005, pp. 665—670. doi:10.2514/1.3856

[16] Papadakis, M., and Wong, S. J., "Parametric Investigation of a Bleed Air Ice Protection System," AIAA Paper 2006-1013, Jan. 2006.

[17] Spalart, P. R., and Allmaras, S. R., "A One-Equation Turbulence Model for Aerodynamic flows," AIAA Paper 92-0439, 1992.

[18] Munson, B. R., Young, D. F., and Okiishi, T. H., *Fundamentals of Fluid Mechanics*, 3rd ed., Wiley, Hoboken, NJ, 1998.

[19] Dyban, E. P., and Mazur, A. I., "Heat Transfer for a Planar Air Jet Striking on a Concave Surface," *Inzhenerno Fizicheskii Zhurnal*, Vol. 17, No. 5, Nov. 1969, pp. 785–790.

[20] Gardon, R., and Cobonpue, J., "Heat Transfer Between a Flat Plate and Jets of Air Impinging on It," *International Developments in Heat Transfer*, American Society of Mechanical Engineers, New York, 1962, pp. 454–460.

[21] Behbahani, A. I., and Goldstein, R. J., "Local Heat Transfer from Staggered Arrays of Impinging Circular Air Jets," American Society of Mechanical Engineers Paper 82-GT-211, 1982.

[22] Schlichting, H., and Gersten, K., *Boundary Layer Theory*, 8th ed., Springer, New York, 2000.