Investigation of fluid flow and heat transfer characteristics for a thermal anti-icing system of a high-altitude and long-endurance UAV

Jen-Chieh Cheng1,* and You-Ming Chen1

1Department of Aeronautical Engineering, National Formosa University, Huwei, Yunlin, Taiwan
*Corresponding author: jcheng@nfu.edu.tw

ABSTRACT

This research performs a three-dimensional simulation to investigate the fluid flow and heat transfer characteristics for hot-air jets impinging on the wing leading-edge surface. Both the periodic model and the whole model are proposed to examine the thermal anti-icing performance for hot air ejecting from a piccolo tube onto the impinging surface. The results show that, for the periodic model, the enhancement of the average Nusselt number can be up to 94.4%, and the enhancement of the average heat flux is up to 29.7% for 100 \( \leq u_i \leq 350 \) m/s and 300 \( \leq T_j \leq 550 \) K when compared with the results of the basic case of \( u_i = 200 \) m/s and \( T_j = 450 \) K. The maximum enhancement of the \( \overline{Nu} \) is 62.3% as the spacing decreases from \( S_i = 8 \) to \( S_i = 4 \) and the optimum \( \overline{Nu}_{max} \) and \( \overline{Nu} \) occur at \( S_i = 5 \) and \( S_i = 6 \) for the single-array holes with \( 3 \leq S_i \leq 7 \) and \( 4 \leq S_i \leq 8 \). In addition, the \( \theta_h \) for maximum \( \overline{Nu} \) is 10° and the maximum enhancement of the \( \overline{Nu} \) is \( 15.7% \) for double-array holes and staggered-array holes as compared with single-array holes. In addition, the nonuniformity of Nusselt number and heat flux distributions are significantly improved. For the whole model, the maximum enhancement of the average Nusselt number is \( 7.5% \) and the optimum configuration is \( \theta_h = 40° \), for cases with \( L_a = 60, D_p = 8, m = 0.15 \) kg/s, \( S_i = 6, 1 \leq N_i \leq 5, 10 \leq S_i \leq 30 \) and \( 10 \leq \theta_h \leq 60° \).

KEYWORDS: thermal anti-icing system, jet impingement, heat transfer, UAV

1. INTRODUCTION

Ice accretion occurs on a high-altitude and long-endurance (HALE) unmanned aerial vehicle (UAV) during the climb stage of flight to an altitude below 25 000 feet. It usually passes through areas with high humidity or sufficient water content in the clouds, which are naturally icing high-risk regions or strong icing conditions. The wing aerodynamic characteristics and endurance time of the UAV could seriously deteriorate if ice forms on the wing surfaces. The manned aircraft are usually equipped with deicing and/or anti-icing devices, which are activated by the pilots according to flight manuals or their own experience. For the unmanned aircraft, adding anti-icing and deicing devices incurs weight and power penalties and hence reduces the useful payload and endurance. How to ensure sensors can correctly detect icing and when to activate the anti-icing and deicing devices are also difficult technical problems. In general, the deicing and anti-icing devices of aircraft can be divided into the pneumatic-mechanical deicing system, the thermal anti-icing system and the electrothermal deicing system. Among the above three types of ice protection systems, pneumatic deicing devices require more complicated mechanisms and are less applicable for low-energy UAVs. The latter two have the advantages of relatively simple system and easy control. The thermal anti-icing device uses hot air (such as bleed air in the compression section) developed from the engine and guided to the leading edge of the wing through a piccolo tube with slots to transfer the hot air in the inner surface of the wing to increase its temperature and prevent the wing from ice accretion. It has the advantage of saving additional energy needed for a large fuel-powered HALE UAV.

A lot of researchers have dealt with the flow and heat transfer characteristics between a surface and jets of air impinging on it. Gardon and Cobonpue [1] experimentally study the heat transfer performance of arrays of jets impinging on a heating flat plate. Results show that the heat transfer rate of the plate is related to the nozzle mechanism and the jet flow rate. An experimental study has been carried out by Choi et al. [2] for jet impingement cooling on a semicircular concave surface. It finds that the distance between the nozzle exit and target surface affects the thickness of the wall jet as well as the stagnation heat transfer rate. Lee et al. [3] investigate the effect of nozzle diameter on the heat transfer and fluid flow for jet impingement on a flat plate. The experimental results show that the momentum and turbulent intensity, as well as the local Nusselt number, increase as the nozzle diameter increases. Sharif and Mothe [4] used numerical methods to investigate the heat transfer characteristics of jet flowing through a narrow hole and impinging on a circular surface during turbulent flow. They found that the Reynolds number and the curvature of the impinging surface at the nozzle outlet affect the heat transfer enhancement. Singh et al. [5] investigated numerically the fluid flow and heat transfer...
characteristics on circular jet impingement heat transfer from a constant temperature circular cylinder with several turbulence models. It is found that the Re-Normalisation Group $k-\epsilon$ model predicts accurately heat transfer characteristics compared to all other turbulence models considered in the study. Results show that the stagnation point Nusselt number increases as the ratio of the distance between the nozzle exit and the cylinder surface to the diameter of the jet ($h/d$) and the ratio of nozzle diameter to cylinder diameter ($d/D$) decrease, and only near the stagnation region, the effects of change in $h/d$ and $d/D$ are significant. Isman et al. [6] study the heat transfer characteristics in the single slot jet impingement cooling process of constant heat flux surface with two-dimensional (2D) turbulent flow in steady state. The effects of inlet turbulence intensity on the heat transfer characteristics of stagnation region are analyzed for several turbulent models. Heo et al. [7] investigate numerically the characteristics of the fluid flow and heat transfer of staggered inclined impinging jets on a concave surface. Results show that the heat transfer on the concave surface is enhanced by the staggered inclination of jet nozzles, and the pitch of vertical jet nozzles improves the overall heat transfer. Baghel et al. [8] experimentally study the heat transfer properties of a heated convex semicylindrical curved surface impinged by a free surface water jet. Several cases with different curvature ratios, jet to surface distances and Reynolds numbers are analyzed in the study. It is found that the Nusselt number for the curved surface case is higher than that for the flat surface and is affected by curvature away from the impact.

Rama Kumar and Prasad [9] used numerical method to study the flow field and heat transfer characteristics of a single row of jets impinging on a concave surface and found that a pair of vortices that rotate in opposite directions between the jets is affected by the upwash airflow and mutual attraction. In addition, the local pressure coefficient and the Nusselt number on the concave surface are $\sim$12% lower than the experimental results. Craft et al. [10] investigate numerically the flow field and heat transfer characteristics of the three-dimensional (3D) array jet stream impinging on the concave surface and used two models of linear and nonlinear eddy viscosity for analysis and experimentation. By comparison, it is found that the wall jet was generated by the jet impact on the concave surface and the downflow phenomenon was caused by the collision of the wall jet. However, the nonlinear vortex viscosity model is better than the linear one, and the results are similar to the experimental data. The flow and heat transfer characteristics in the cooling of a heated surface by impinging slot jets have been investigated numerically by Sahoo and Sharif [11]. Computations are done for vertically downward-directed 2D slot jets impinging on a hot isothermal surface at the bottom and confined by a parallel adiabatic...
surface on top. It is observed that for a given domain aspect ratio and Richardson number, the average Nusselt number at the hot surface increases with increasing jet exit Reynolds number. However, the average Nusselt number does not change significantly with the Richardson number and the jet position. Guan et al. [12] numerically investigate the conjugated convective heat transfer on the leading edge of a conical wall subjected to external cold flow and internal hot-jet impingement by a single chevron nozzle. The effects of the chevron length \( l/d \) and chevron penetration depth \( p/d \) on the hot-jet impingement heat transfer performance are analyzed. It is found that the averaged heating effectiveness under dimensionless jet-to-leading edge distance \( H/d = 4 \) is greater than that under \( H/d = 2 \) at the same chordwise location once \( s/d \) is beyond 6 due to a large curvature of the conical surface.

Some studies in the literature have analyzed convective heat transfer characteristics of a thermal anti-icing system from the hot-air jets impinging on the cooled surfaces. Brown et al. [13] experimentally investigate the heat transfer in an aircraft nacelle anti-icing system. A correlation based on the heat transfer impingement area is developed. The correlation is independent
of the distance between the jet and the impingement surface. Fregeau et al. [14] numerically study the heat transfer distribution along a 3D curved surface from an array of hot-air jets impinging on the surface. The general trends have shown that the increase in Mach number results in an increase in heat transfer. For small nozzle-to-surface height, the maximum Nusselt number is found to decrease as the jet nozzles are spaced farther apart. For large heights, the opposite trend is observed. The heat transfer distribution is seen to decrease along the surface in all directions away from the stagnation region. Some 2D and 3D wing leading-edge models are simulated by Hua and Liu [15] for a thermal anti-icing system. Most of the estimated 2D flow characteristics are over but within 15% accuracy compared with their 3D results. Besides, the temperature over the entire upper skin increases correspondingly, and the temperature of the lower surface decreases as the slot is located 15° up position to the chord direction. Liu and Hua [16] perform a 3D simulation of a wing segment including the piccolo-type thermal anti-icing bays inside the leading edge. Simulation results visually reveal the hot/cold flow interactions and heat conductivity through the fluid and solid zones. The calculated leading-edge surface temperature is compared with flight-test data of a similar configuration. Fregeau et al. [17] study a 3D hot-air jet array impinging on the circular surface. The numerical correlations for the average and maximum Nusselt number for different nozzle-to-nozzle spacing, nozzle-to-surface height and hot-air jet Mach numbers have been established. A numerical study was conducted by Saeed [18] to simulate the heat transfer from an array of jets onto an impingement surface typical of those found on aircraft wing/SLT surfaces. The single array and the array with a 20° stagger yield better surface heat transfer than the 10° stagger. The etched surface or inner liner yields almost two to three times better surface heat transfer than the others.

Fregeau et al. [19] numerically study the heat transfer characteristics of a jet array impinging on the circular surface. It is found that the effect of heat transfer is better due to the lower flow momentum between the jets. Mu et al. [20] study the in-flight electrothermal deicing process to simulate the conjugate mass and heat transfer phenomena of water film runback, phase change and solid heat conduction. Mathematical models of water film runback and phase change are established and solved by means of a loosely coupled method. A deicing process is numerically simulated following an icing tunnel experiment, and the results match well with those in the literature. Khalil et al. [21] study numerically an anti-icing structure for a usual aircraft wing of NACA 23014 airfoil shape. Fregeau et al. [19] numerically study the heat transfer characteristics of a jet array impinging on the circular surface. It is found that the effect of heat transfer is better due to the lower flow momentum between the jets. Mu et al. [20] study the in-flight electrothermal deicing process to simulate the conjugate mass and heat transfer phenomena of water film runback, phase change and solid heat conduction. Mathematical models of water film runback and phase change are established and solved by means of a loosely coupled method. A deicing process is numerically simulated following an icing tunnel experiment, and the results match well with those in the literature. Khalil et al. [21] study numerically an anti-icing structure for a usual aircraft wing of NACA 23014 airfoil shape.

Table 2 The fixed and variable parameters for cases investigated in this study.

| Fluid | Partial segment of wing: || Piccolo tube: | Hot air at inlet of piccolo tube (for whole model): | Temperature of impingement surface: | Temperature at exhaust holes: | Pressure at exhaust holes: |
|-------|-----------------------------|-------------|-------------------------------|----------------------------------|-------------------------------|-------------------------------|
| air, $k_i = 0.0263 \text{ W/(m K)}$ | $H_i = 20$, $W_s = 10$, $L_s = 15$, $L_s = 1.25$, $S_i = W_{ev} = 2.560$ (for whole model) | $D_p = 8, 20$; $0^\circ \leq \theta_h \leq 60^\circ$; $4 \leq S_h \leq 12$; $1 \leq N_h \leq 5$ (for whole model) | $m = 0.15 \text{ kg/s}$, $T_i = 450 \text{ K}$ | $T_{aw} = 260 \text{ K}$ | $T_{aw} = 260 \text{ K}$ | $P_{aw} = 60 \text{ kPa}$ |

$u_i = 200 \text{ m/s}$ and $T_{aw} = 260 \text{ K}$. 

$T_{aw} = 260 \text{ K}$. 

$P_{aw} = 60 \text{ kPa}$.
Fluid flow and heat transfer characteristics for a thermal anti-icing system

2. ANALYSIS

The physical system under consideration, as shown in Fig. 1, is a segment of the thermal anti-icing system in which heated air is ducted spanwise through a piccolo tube along the inside of the wing leading edge. For 3D investigation, this study considers the system with a length of \( l_a \) consisting of the leading edge of an NACA 4415 with a partial semicircular shape and the piccolo tube with diameter \( d_p \). The holes on the piccolo tube surface are arranged in line or in a staggered configuration. Hot air enters the piccolo tube with uniform temperature and uniform velocity. The hot-air jets flow out through the holes and impinge upon the segment of the wing leading edge to heat up the skin and prevent ice accretion. Thereafter, the air exhausts to ambient through exhaust holes. The operation modes including the effects of various velocities and temperatures of the hot-air jets are discussed. The mechanism arrangement of the system that includes (1) the design of the piccolo tube (diameter of piccolo tube \( d_p \), diameter of hole \( d_h \), horizontal spacing of holes \( S_h \), number of holes \( N_h \) and angle between the holes and \( X-Y \) plane \( \theta_h \)) and (2) the design of the position of the piccolo tube (distance between the hole and impingement surface \( S_i \)) is investigated.

In this study, the fluid flow and heat transfer from hot-air jets impinging on the wing leading-edge surface are governed by the 3D Reynolds-averaged Navier–Stokes equations with the eddy viscosity-based turbulent assumption. The governing equations describing the steady turbulent convection in this thermal anti-icing system domain include continuity equation, momentum equations, energy equation and equation of state by adopting ideal gas approximation. The governing equations are described as follows:

**Continuity equation:**

\[
\frac{\partial}{\partial x_j} (\rho u_j) = 0. \tag{1}
\]
Figure 5 The temperature distributions on the $X$–$Y$ plane and $X$–$Z$ plane for cases with single-array holes and $T_j = 450$ K, $S_i = 5$ and $S_n = 6$: (a) $u_j = 100$ m/s; (b) $u_j = 200$ m/s; and (c) $u_j = 300$ m/s.

Figure 6 The local Nusselt number distributions on the impingement surface for various cases with single-array holes and $T_j = 450$ K, $S_i = 5$ and $S_n = 6$: (a) $u_j = 100$ m/s; (b) $u_j = 200$ m/s; and (c) $u_j = 300$ m/s.
Fluid flow and heat transfer characteristics for a thermal anti-icing system

Figure 7 The wall heat flux distributions on the impingement surface for cases with single-array holes and \( u_i = 200 \text{ m/s}, S_i = 5 \) and \( S_n = 6 \): (a) \( T_j = 350 \text{ K} \); (b) \( T_j = 450 \text{ K} \) and (c) \( T_j = 550 \text{ K} \).

Momentum equations:

\[
\frac{\partial}{\partial x_j} \left( \rho \bar{u}_j \bar{u}_i \right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \tau_{ij} - \rho \bar{u}_i \bar{u}_j \delta_{ij} \right),
\]

Energy equation:

\[
\frac{\partial}{\partial x_j} \rho \bar{u}_j (\bar{E} + p) = \frac{\partial}{\partial x_j} \left[ \bar{u}_i \left( \tau_{ij} - \rho \bar{u}_i \bar{u}_j \right) \right] - \frac{\partial \bar{q}_j}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ -C_p \rho \bar{u}_j \bar{T} \right].
\]

Equation of state:

\[
p = \rho R \bar{T},
\]

where \( \rho, p, \bar{u}_i, \bar{T} \) and \( \bar{E} \), respectively, are the mean density, pressure, velocity, temperature and total energy represented in time- and mass-averaged values. \( \tau_{ij} \) and \( \bar{q}_j \), respectively, are the mean viscous stress and heat flux in mass-averaged values. \( \mu \) is the dynamic viscosity, \( C_p \) is the specific heat at constant pressure, \( R \) is the gas constant and \( Pr \) is the Prandtl number. In addition, \(-\rho \bar{u}_i \bar{u}_j \delta_{ij}\) is the turbulent Reynolds stress and \(C_p \rho u_j \bar{T}\) is the turbulent heat flux. For the turbulent parameters, \( \mu_t \) is the turbulent viscosity, \( k \) is the turbulent kinetic energy and \( Pr_t \) is the turbulent Prandtl number. In this study, the standard \( k-\varepsilon \) model proposal by Sarkar [23] is used to calculate the above turbulent terms to close this whole partial differential equation system.

The governing equations are subjected to the boundary conditions that the no-slip condition is assumed on all the solid walls, and the walls are assumed to be insulated, except at the impingement surface where a uniform outside temperature along the surfaces \( T = T_w \) is imposed. Two models under consideration in the present study are the periodic model and the whole model. The former model focuses on the portion close to the leading edge that can be assumed as concentric circles of 180° and the physical phenomena appeared periodically between adjacent holes along the piccolo tube. The latter model includes the whole thermal anti-icing system in the computation. The hot-air stream is assumed to have uniform velocity and temperature at holes on the piccolo tube for the periodic model, and is assumed to have constant mass flow rate and uniform temperature on the inlet of
The local Nusselt number distributions on the impingement surface for cases with single-array holes and $u_i = 200 \text{ m/s}$, $S_i = 5$ and $S_n = 6$: (a) $T_j = 350 \text{ K}$; (b) $T_j = 450 \text{ K}$; and (c) $T_j = 550 \text{ K}$.

Figure 8 The local Nusselt number distributions on the impingement surface for cases with single-array holes and $u_i = 200 \text{ m/s}$, $S_i = 5$ and $S_n = 6$: (a) $T_j = 350 \text{ K}$; (b) $T_j = 450 \text{ K}$; and (c) $T_j = 550 \text{ K}$.

The local Nusselt number along the impingement surface is of interest to thermal system design. It is defined as

$$Nu = \frac{h_c d_h}{k_a} = \frac{q'' d_h}{k_a (T_w - T_j)},$$

where $h_c$ is the air heat transfer coefficient, $k_a$ is the air thermal conductivity and $q''$ is the local heat flux of the impingement surface.

In addition, the average Nusselt number for the impingement surface is calculated by

$$\overline{Nu} = \frac{1}{A_s} \int_A Nu dA,$$

where $A_s$ is the area of the impingement surface.

3. NUMERICAL METHODOLOGY

In this study, a 3D turbulent numerical simulation is performed by the commercial ESI CFD-ACE software to investigate the flow and heat transfer characteristics of the thermal anti-icing system of UAV. The physical phenomena are governed by the Reynolds-averaged Navier–Stokes equations. Besides, the turbulence is modeled using the standard $k$–$\varepsilon$ model. The finite volume method is adopted to discretize the governing equations using the pressure-based algorithm to deal with the pressure–velocity coupling calculation. The SIMPLE (semi-implicit method of pressure link equations) algorithm proposed by Patankar [24] and the SIMPLEC (semi-implicit method of pressure link equations—consistent) algorithm proposed by Van Doormaal and Raithby [25] are used in this work. The software adopts the above two algorithms to solve the momentum equations. The convection terms are discretized by the second-order upwind method, and the diffusion terms are discretized by the central interpolation method. The line-by-line method with iteration is employed to solve the systems of algebraic equations obtained from discretizing the governing equations. The solution is considered to be converged when the relative differences of variables at each node between two consecutive iterations of the system are less than a prescribed value of $10^{-4}$. The grid system of the periodic model (symmetric about the $X$–$Z$ plane) and the integrated model (cutaway) are shown in Fig. 2. The isothermal cold wall is set on the impingement surface to simulate the ice accretion behaviors of a UAV at a high altitude. To obtain enhanced accuracy, nonuniform grids are arranged in the computational domain. The grid density is higher in the vicinity
Figure 9 The local Nusselt number distributions on the impingement surface for cases with single-array holes and $u_j = 200 \text{ m/s}$ and $S_n = 6$: (a) $S_i = 3$; (b) $S_i = 5$; and (c) $S_i = 6$.

of the wall and holes of the piccolo tube to capture the drastic variations of the flow and thermal fields. The proposed numerical algorithm is validated rigorously in this study. First, for the 3D periodic model with $k_b = 0.0263$, $D_p = 20$, $S_i = S$, $S_n = 6$, single array, $u_i = 200 \text{ m/s}$, $T_j = 450 \text{ K}$ and $T_w = 260 \text{ K}$, different numbers of grid are employed to ensure that the solution is grid independent. The calculated grid points used were 61 750, 88 000, 165 500 and 268 000 for testing, and the $Nu_{\text{max}}$ on the impact surface was compared with different grid numbers. Table 1 shows the $Nu_{\text{max}}$ and comparison of different grid systems on the impact surface. It can be seen that the test results are quite close, and the error of $Nu_{\text{max}}$ between 165 500 and 268 000 cases is $\sim 0.13\%$. Therefore, the 165 500 grid system is chosen in this computation for the periodic model. The similar grid arrangement is also used in the whole model. Besides, several first-cell heights of the wall boundary grid are applied on the computation for the periodic model of the hot-air jet. When the first-cell height is $1.5 \times 10^{-5}$, the difference in $Nu_{\text{max}}$ is $<1.5\%$ compared with results computed by Saeed [18]. Second, results for the case of a cold jet flow impinging on a hot flat plate with $k_b = 0.0263$, $S_i = 6$, $S_n = 20$, single line of holes, $u_i = 200 \text{ m/s}$ and $T_w - T_j = 20 \text{ K}$ are compared to the work of relevant literature. Figure 3 shows the Nusselt number distribution of the present prediction and the experiment by Gardon and Cobonpue [1] and computed results by Fregeau et al. [17]. Good agreements are found between the present prediction and the experiment and computed results presented, respectively. Through these program tests, the proposed numerical scheme is considered to be appropriate for the present problem under investigation.

4. RESULTS AND DISCUSSION

This study intends to perform a 3D numerical simulation to thoroughly investigate the flow and convective heat transfer characteristics for the thermal anti-icing system of a UAV. Efforts are devoted to investigating the influences of geometric configurations of the piccolo tube and operating conditions on the flow structures, temperature distributions and the heat transfer characteristics of the thermal anti-icing system. Besides, effective ways to enhance the heating performance of the system are also searched. Finally, the effects of the operation procedures in this system are also integrated into the investigation.

In this study, the periodic and whole models are simultaneously proposed to study the phenomena of the thermal anti-icing system. An inspection of the foregoing analysis indicates that the flow and thermal characteristics in the present system depend on nine parameters for the periodic model. These are the velocity of hot jet on the holes of piccolo tube $u_j$, the temperature
temperature of hot jet on the holes of piccolo tube $T_j$, the temperature of impingement surface $T_w$, the dimensionless diameter of piccolo tube $D_p$, the dimensionless distance between the hole and impingement surface $S_i$, the dimensionless horizontal spacing of holes $S_h$, the angle between the holes and the $X$–$Y$ plane $\theta_h$, the temperature at the exhaust $T_{\text{out}}$ and the pressure at the exhaust $P_{\text{out}}$. Meanwhile, for the whole model, in addition to the above-mentioned parameters for the periodic model, there are other parameters required in the whole numerical computation to govern the transfer processes for the whole model. These include the mass flow rate at the entrance of the piccolo tube $m$, the number of holes $N_h$, the dimensionless length, width and height of the segment of the leading edge $L_{\text{se}}$, $W_{\text{se}}$, $H_{\text{se}}$, the dimensionless length and width of the exhaust hole $S_e$, $W_e$, the dimensionless distance between the exhaust hole and rib of the segment $L_{\text{se}}$, and the dimensionless distance between the exhaust hole and inner liner skin $L_w$. It is noted that the $u_t$, which is not known a priori for the whole model, should be obtained in the solution procedures. Since a vast number of the governing parameters are required to characterize the system, a comprehensive analysis of all combinations of problems is not practical. While computations can be performed by any combination of these parameters, the objective here is to illustrate the effects of $u_t$, $T_j$, $S_i$, $S_h$, $\theta_h$ and $N_h$ on the heating characteristics of the cold surface of the wing leading edge by using hot-air jet impingement. Table 2 represents the settings of fixed and variable parameters for the numerical computations in this study. It is noticed that the setting values of parameters are based on practical engineering applications.

### 4.1 The periodic model

Initially, the effects of various $u_t$ and $T_j$ on the flow structures and heat transfer characteristics are investigated for the cases with single-array holes on the piccolo tube and $S_i = S_h = 6$ and $\theta_h = 0^\circ$ for the periodic model. The 3D velocity vector and the 2D velocity vector projected on the $X$–$Y$ plane ($Z = 0$) and $X$–$Z$ plane ($Y = 0$) for the case with $u_t = 300$ m/s and $T_j = 450$ K are shown in Fig. 4. The results indicate that the hot-air jets impinge directly on the cold wall in the region close to the stagnation point, which would result in high heat transfer performance in this region. The flow then expands and slows down along the surface and a local recirculation occurs induced by the jet flow. In addition, the flow interaction between the adjacent hot-air jets is evident. The neighboring flows collide at the centerline of the surface and induce two recirculations with opposite rotating direction. Figure 5 shows the temperature distributions on the $X$–$Y$ plane and $X$–$Z$ plane for the cases of single-array holes with various $u_t$ and $T_j = 450$ K, $S_i = 5$ and $S_h = 6$. The temperature depends on the speed and direction of the airflow. When the hot air injects toward the impingent surface, the higher fluid speed with the direction approximately perpendicular to the surface can enhance the heat transfer efficiency. The temperature is higher in the region around the stagnation point. The overall temperature increases as the injection velocity increases. Figure 6 illustrates the influences of $u_t$ on the local Nusselt number distributions on the impingement surface with $T_j = 450$ K. It is found in this figure that the maximum Nusselt number appears at the stagnation point. The Nusselt numbers decay apparently in the region away from the stagnation point. This is due to the fact that the flow speed decreases with the flow expansion and the flow direction approximately parallel to the surface. Besides, the $Nu$ at the region between the stagnation points is higher due to the jet flow interaction. As the jet flow speed increases, the maximum and average Nusselt numbers both increase, especially in the region near the stagnation point. It is pointed out that the enhancement of the average Nusselt number $Nu$ is $\sim$94.4% as the velocity increases from 200 to 350 m/s.

Considering the effects of jet flow temperature on the heat transfer characteristics for the impingement surface, Figs 7 and 8 illustrate the local heat flux and Nusselt number distributions for the system with a variation of $T_j$ and fixed $u_t = 200$ m/s. The results show that the overall heat flux on the surface increases with the increase in the jet flow temperature. However, the
Figure 11 The local Nusselt number distributions on the impingement surface for cases with single-array holes and \( u_j = 200 \text{ m/s}, T_j = 450 \text{ K} \) and \( S_i = 5 \): (a) \( S_n = 4 \); (b) \( S_n = 6 \); and (c) \( S_n = 8 \).

Figure 12 The local Nusselt number distributions on the impingement surface for cases with double-array holes and \( u_j = 100 \text{ m/s}, T_j = 450 \text{ K}, S_i = 5 \) and \( S_n = 6 \): (a) \( \theta_h = 5^\circ \); (b) \( \theta_h = 15^\circ \); (c) \( \theta_h = 20^\circ \); and (d) \( \theta_h = 35^\circ \).
Figure 13 The local Nusselt number distributions on the impingement surface for cases with staggered-array holes and \( u_j = 200 \text{ m/s}, T_j = 450 \text{ K}, S_i = 5 \) and \( S_n = 6 \): (a) \( \theta_h = 5^\circ \); (b) \( \theta_h = 15^\circ \); (c) \( \theta_h = 25^\circ \); and (d) \( \theta_h = 35^\circ \).

Figure 14 The variations of maximum Nusselt number and average Nusselt number versus \( \theta_h \) for the case with staggered-array holes and \( u_j = 200 \text{ m/s}, T_j = 450 \text{ K}, S_i = 5 \) and \( S_n = 6 \).

local Nusselt number becomes smaller for higher \( T_j \). It can be explained that the high-temperature air jet can transfer more heat to the cold wing surface, and enhances the anti-icing effects. On the contrary, the heat transfer coefficient decreases for the cases with higher \( T_j \). This is due to the higher temperature difference, \( T_j - T_w \), which appears in the definition of \( Nu \). When the temperature difference increases from 190 to 290 K, the enhancement of the average heat flux is \( \sim 29.7\% \) and the reduction of the average Nusselt number is \( \sim 15.1\% \).

Attention is now turned to investigate the effects of geometric configuration of the piccolo tube for the cases with various \( S_i, S_n \) and \( \theta_h \) at fixed \( u_j = 200 \text{ m/s} \) and \( T_j = 450 \text{ K} \) for the periodic model. The effects of \( S_i \) on the distributions of local Nusselt number are represented in Fig. 9. It is observed that the \( Nu \) for the region around the stagnation point is larger as the distance between the hole and impingement surface, \( S_i \), is shorter. This is due to the fact that the air jet impingement is more concentrated on the cold surface for smaller \( S_i \). On the contrary, the \( Nu \) distribution is more uniform as \( S_i \) increases for the sake of a broader range of jet spray on the surface. Besides, the tendency of maximum \( Nu_{\text{max}} \) does not appear in a monotone way with the variation of \( S_i \). The interactions between the adjacent jets play an important role in the enhancement of heat transfer. It is expected that there exists an optimum distance between the piccolo tube and the impingement surface. Figure 10a and b plots the variations of \( Nu_{\text{max}} \) and \( \overline{Nu} \) versus \( S_i \) with various \( u_j \) and \( T_j \). It is found that the \( Nu_{\text{max}} \) increases initially as \( 3 \leq S_i \leq S_r \) and then turn to decrease evidently as \( S_i > 5 \). The maximum \( Nu_{\text{max}} \) occurs at \( S_i = 5 \). A similar result of the variation of average Nusselt number \( \overline{Nu} \) versus \( S_i \) can be observed in Fig. 10b where the distance of maximum \( \overline{Nu} \) increases with \( S_i \) until \( S_i = 6 \). In addition, the optimum \( S_i \) for \( Nu_{\text{max}} \) and \( \overline{Nu} \) are both independent of the velocity and temperature on the holes of the piccolo tube.

Furthermore, it is interesting to explore the effect of the horizontal spacing between holes, \( S_n \), on the heat transfer performance. Figure 11 plots the local Nusselt number distribution on the impingement surface for various \( S_n \) at fixed \( u_j = 200 \text{ m/s}, T_j = 450 \text{ K} \) and \( S_i = 5 \). With the reduction of the area and enhancement of the interaction between the hot-air jets, the \( Nu_{\text{max}} \) and \( \overline{Nu} \) rise with the decrease in \( S_n \). The enhancement
Table 3 Comparison of average Nusselt numbers for cases with double-array and staggered-array holes with single-array holes various \( \theta_h \) at \( T_j = 450 \text{ K} \), \( S_i = 5 \) and \( S_n = 6 \).

| \( \theta_h \) | Double array \( u_i = 100 \text{ m/s} \) | Staggered array \( u_i = 200 \text{ m/s} \) |
|-------------|-----------------|-----------------|
| 5           | 0.806           | 0.914           |
| 10          | 0.839           | 1.157           |
| 15          | 0.871           | 1.111           |
| 20          | 0.875           | 1.120           |
| 25          | 0.909           | 1.123           |
| 30          | 0.873           | 1.092           |
| 35          | 0.837           | 1.083           |

\( \overline{Nu}_A \): the average Nusselt number of single-array holes for various \( \theta_h \) at \( u_i = 200 \text{ m/s} \).

Comparison of average Nusselt numbers for cases with double-array and staggered-array holes with single-array holes various \( \theta_h \) at \( T_j = 450 \text{ K} \), \( S_i = 5 \) and \( S_n = 6 \).

The above results illustrate that the higher Nusselt number and heat flux are rather concentrated in the region close to the stagnation point for the case with single-array holes. The \( Nu \) and \( q'' \) drop sharply in the region away from the impingement point. It is reasonably recognized that the double arrays and the staggered arrays of jets could improve the nonuniformity of \( Nu \) and \( q'' \) distributions. Thus, it is interesting to examine the transfer characteristics for the system with double- and staggered-array holes on the piccolo tube. In order to maintain the same mass flow rate of hot-air jet, the \( u_i \) of double-array case is specified to be half of the single-array and staggered-array cases. The influence of \( \theta_h \) on the \( Nu \) distribution for the double-array case with fixed \( u_i = 100 \text{ m/s} \), \( T_i = 450 \text{ K} \), \( S_i = 5 \) and \( S_n = 6 \) is shown in Fig. 12. The \( Nu \) distribution is symmetric and the higher \( Nu \) is more uniform, as compared with the single-array case. The intensity of interaction between the upper and lower arrays of jets varies with \( \theta_h \). The maximum \( Nu_{max} \) and \( \overline{Nu} \) occur at \( \theta_h = 15^\circ \) and \( \theta_h = 25^\circ \), respectively. It is also found that the reduction of velocity on each hole affects the heat transfer performance apparently.

Figure 13 illustrates the influence of \( \theta_h \) on the \( Nu \) distribution for the case with staggered arrays of hot-air jets and \( u_i = 200 \text{ m/s} \), \( T_i = 450 \text{ K} \), \( S_i = 5 \) and \( S_n = 6 \). The distribution of \( Nu \) is asymmetric and the region of higher \( Nu \) is

![Image](https://example.com/image.png)

**Figure 15** The temperature distributions on the X–Y plane and X–Z plane for the whole model with single-array holes and \( m = 0.15 \text{ kg/s} \) and \( T_j = 450 \text{ K} \): (a) \( S_n = 30 \) and \( N_h = 1 \); (b) \( S_n = 15 \) and \( N_h = 3 \); and (c) \( S_n = 10 \) and \( N_h = 5 \).
broader than the single-array case. The interaction between the neighboring jets is weakened as the $\theta_h$ increases. The variations of $Nu_{\text{max}}$ and $\overline{Nu}$ versus $\theta_h$ are plotted in Fig. 14. It is observed that the $Nu_{\text{max}}$ is lower at $\theta_h < 25^\circ$ and attains a maximum at $\theta_h = 25^\circ$. The $\overline{Nu}$ increases drastically with $\theta_h$ at first, and the maximum value appears at $\theta_h$ between 10° and 12°. At $\theta h > 15^\circ$, the $\overline{Nu}$ reduces gradually. Table 3 shows the comparison of Nusselt number for cases with double-array, staggered-array and single-array holes at various $\theta_h$. Results show that the average Nusselt number for the double-array case is less than that for the single-array case. For the staggered-array case, the maximum enhancement of the average Nusselt number is $\sim 15.7\%$ at $\theta_h = 10^\circ$ as compared with the single-array case.

### 4.2 The whole model

Finally, the flow and heat transfer characteristics of a whole model for the thermal anti-icing system applied on a segment of the wing are investigated. The fore segment of the NACA 4415 is adopted for the whole model. The grid-independent test of the whole model is also performed, which is similar to that of the periodic model. It is found that the deviation of $Nu_{\text{max}}$ using grids between 1 125 500 and 1 574 600 is $\sim 1.47\%$. The total grids of the whole model are around 1 125 000. The hot air flows into the piccolo tube and spurs out through holes on the tube to prevent ice accretion on the wing. The hot air flowing through the $D_p = 20, L_a = 60, D_p = 8$ and $S_i = 6$ with $T_j = 450 \text{ K}, T_w = 260 \text{ K}$ and $m = 0.15 \text{ kg/s}$ is considered for the numerical computation. The fixed pressure set at the exhaust is 60 kPa, which refers to the atmospheric condition at an altitude of 5000 m. The results are presented for the cases with single-array and staggered-hole holes at $1 \leq N_h \leq 5 \ (10 \leq S_n \leq 30), 10^\circ \leq \theta_h \leq 60^\circ$. Figure 15 shows the effect of hole numbers on the temperature distribution. The total area of holes for each case is identical. It can be observed that the temperature distribution is significantly affected by the flow direction of the hot-air flow. The hot air flows into the piccolo tube and then ejects from the holes to the impinging surface. After being affected by the impact surface, the temperature of air jets gradually decreases. It is also found that the region near the middle of the section has higher temperature distribution. As the number of holes increases, the temperature distribution is more uniform for the sake of the multiple holes that make the hot-air flow easier to flow in the entire area. The influences of the number of holes on the $Nu$ distribution for the case with single-array holes are shown in Fig. 16. It is found that the $Nu_{\text{max}}$ is larger as the $N_h$ is reduced due to the higher jet velocity at the holes. However, the nonuniformity of $Nu$ is improved as the $N_h$ increases. It is expected that a proper arrangement of holes can enhance the overall heat transfer effects.

Furthermore, the effects of staggered arrays of jets are studied for the whole model. Figure 17 shows the effect of staggered holes in the front end of a segmental wing on temperature distribution with different angles of holes $\theta_h$ with fixed $N_h = 4$. It can be observed that when the positions of different holes are angled and staggered, the temperature will be affected by the flow direction of the hot-air flow. The appropriate angle of holes can allow the airflow to flow to the edge of the wing so that the temperature in this area increases. Figure 18 illustrates the influence on the...
The temperature distributions on the X–Y plane and X–Z plane for the whole model with staggered-array holes and \( m = 0.15 \text{ kg/s}, \ T_j = 450 \text{ K}, S_i = 6, N_h = 4 \) and \( S_n = 12 \): (a) \( \theta_h = 10^\circ \); (b) \( \theta_h = 20^\circ \); and (c) \( \theta_h = 30^\circ \).

**Figure 17** The temperature distributions on the X–Y plane and X–Z plane for the whole model with staggered-array holes and \( m = 0.15 \text{ kg/s}, \ T_j = 450 \text{ K}, S_i = 6, N_h = 4 \) and \( S_n = 12 \): (a) \( \theta_h = 10^\circ \); (b) \( \theta_h = 20^\circ \); and (c) \( \theta_h = 30^\circ \).

\( \text{Nu} \) distribution for the case with staggered-array holes and fixed \( N_h = 4 \) \( (S_n = 12) \). It reveals that the \( \text{Nu} \) distribution is slightly asymmetric, and the range of higher \( \text{Nu} \) is wider than the case with single-array holes. The interaction between the neighboring jets becomes weaker as the \( \theta_h \) increases. Table 4 represents the comparison of Nusselt number among the cases of staggered-array holes and single-array holes with various \( \theta_h \) and \( N_h = 4 \). The results indicate that the highest enhancement of the maximum and average Nusselt number can be up to 46.4\% and 7.5\%, respectively, as compared with the single-array holes. In addition, the maximum \( \text{Nu}_{\text{max}} \) and average \( \overline{\text{Nu}} \) occur at \( \theta_h = 40^\circ \), which are larger than those for the periodic model. The overall heat transfer enhancement is smaller than that of the periodic model, which can be due to the discrepancy of the configurations and boundary conditions in both the models. However, the physical phenomena of flow structure, temperature distribution and heat transfer are similar.

The thermal anti-icing protection systems are designed based on the balance of system performance and energy conservation. The relations for the practical design parameters with the ice protection performances have been proposed. The results obtained in this study would provide valuable information for designing such systems in engineering applications.

### 5. CONCLUSIONS

The objective of this study is to investigate the flow and heat transfer characteristics for hot-air jets impinging on the wing leading-edge surface of an aircraft. This study is based on practical engineering applications, especially for the UAV passing through the regions of strong icing conditions during the climbing stage of flight to mid–high altitude. Two models, periodic and whole models, are proposed to explore the thermal anti-icing performance. The results illustrate that, for the periodic model, the enhancement of the average Nusselt number can be up to 94.4\%, and the enhancement of the average heat flux is up to 29.7\% for \( 100 \leq u_j \leq 350 \text{ m/s} \) and \( 300 \leq T_j \leq 550 \text{ K} \) when
The local Nusselt number distributions on the impingement surface for the whole model with staggered-array holes and \( \dot{m} = 0.15 \text{ kg/s}, T_i = 450 \text{ K}, S_i = 6, N_h = 4 \) and \( S_n = 12 \): (a) \( \theta_h = 10^\circ \); (b) \( \theta_h = 20^\circ \); (c) \( \theta_h = 30^\circ \); and (d) \( \theta_h = 40^\circ \).

Table 4 Comparison of Nusselt number of staggered-array holes with single-array holes at various \( \theta_h \) for the whole model and \( \dot{m} = 0.15 \text{ kg/s}, T_i = 450 \text{ K}, S_i = 6, S_n = 12 \) and \( N_h = 4 \).

| \( \theta_h \) | \( \frac{Nu_{\text{max}}}{Nu_{\text{max},F}} \) | \( Nu/Nu_F \) |
|--------------|----------------|---------|
| 0            | 1              | 1       |
| 10           | 1.019          | 1.002   |
| 20           | 1.067          | 1.039   |
| 30           | 1.176          | 1.040   |
| 40           | 1.464          | 1.075   |
| 50           | 1.386          | 1.054   |
| 60           | 1.328          | 1.047   |

\( Nu_{\text{max},F}, Nu_F \): the maximum and average Nusselt numbers of single-array holes.

Table 4 shows the comparison of Nusselt number of staggered-array holes with single-array holes at various \( \theta_h \) for the whole model and \( \dot{m} = 0.15 \text{ kg/s}, T_i = 450 \text{ K}, S_i = 6, S_n = 12 \) and \( N_h = 4 \). The results indicate that the maximum enhancement of the Nusselt number is approximately \( 15.7\% \) for double-array holes and staggered-array holes with \( 5^\circ \leq \theta_h \leq 35^\circ \) as compared with single-array holes. Meanwhile, the nonuniformity of Nusselt number and heat flux distributions can be significantly improved. For the whole model, the maximum enhancements of \( Nu_{\text{max}} \) and \( \bar{Nu} \) are \( 46.4\% \) and \( 7.5\% \) occurring at \( \theta_h = 40^\circ \) as compared with the single-array holes for the case with \( L_a = 60, D_p = 8, \dot{m} = 0.15 \text{ kg/s}, S_i = 6, 1 \leq N_h \leq 5, 10 \leq S_n \leq 30 \) and \( 10^\circ \leq \theta_h \leq 60^\circ \). By comparing the results of the periodic model and the whole model, it is found that the physical phenomena and the trend in heat transfer effect are similar. Besides, the overall benefits are better for the periodic model.

NOMENCLATURE

\( d_h \) = diameter of holes on a piccolo tube
\( d_p \) = diameter of a piccolo tube
\( D_p \) = dimensionless diameter of a piccolo tube, \( d_p/d_h \)
\( h \) = heat transfer coefficient
\( h_s \) = height of the segment of leading edge
\( H_s \) = dimensionless height of the segment of leading edge, \( h_s/d_h \)
\( k_a \) = thermal conductivity of air
\( l_a \) = length of the segment of leading edge
\( l_e \) = distance between the exhaust hole and rib of the segment
\( l_e \) = distance between the exhaust hole and inner liner skin
\( L_a \) = dimensionless length of the segment of leading edge, \( l_a/d_h \)
\( L_e \) = dimensionless distance between the exhaust hole and rib of the segment, \( l_e/d_h \)
\( L_s \) = dimensionless distance between the exhaust hole and inner liner skin, \( l_s/d_h \)
\( \dot{m} \) = mass flow rate
\( N_h \) = number of holes on a piccolo tube
\( Nu \) = local Nusselt number, Eq. (9)
\( Nu_{\text{max}} \) = maximum \( Nu \) of the impingement surface
\( \bar{Nu} \) = average \( Nu \) of the impingement surface
\( Pr \) = Prandtl number, \( \nu/\alpha \)
\( q' \) = local wall heat flux, \( k_a(\partial T/\partial n)_w \)
\( s_e \) = length of the exhaust hole
\( s_i \) = distance between the hole and impingement surface
\( s_n \) = spacing of holes
\( S_i \) = dimensionless length of the exhaust hole, \( s_i/d_h \)
\( S_t \) = dimensionless distance between the hole and impingement surface, \( s_t/d_h \)
\( S_n \) = dimensionless spacing of holes, \( s_n/d_h \)
\( T \) = temperature
\( T_{\text{out}} \) = temperature at the exhaust
\( T_w \) = outside temperature of the impingement surface
\( u \) = velocity in the x direction
\( u_i \) = velocity of hot-air jets at the holes
\( v \) = velocity in the y direction
\( w \) = velocity in the z direction
\( w_x \) = width of the segment of leading edge
\( w_s \) = width of the square exhaust hole
\( W_s \) = dimensionless width of the segment of leading edge, \( w_s/d_h \)
\( W_e \) = dimensionless width of the square exhaust hole, \( w_e/d_h \)
\( X \) = dimensionless longitudinal coordinate, \( x/d_h \)
\( Y \) = dimensionless vertical coordinate, \( y/d_h \)
\( Z \) = dimensionless lateral coordinate, \( z/d_h \)
\( \alpha \) = thermal diffusivity of air
\( \rho \) = density
\( \theta_h \) = angle between the holes and \( X-Z \) plane
\( \mu \) = dynamic viscosity of air
\( \nu \) = kinematic viscosity of air

ACKNOWLEDGEMENTS

The financial support of this study by the Engineering Division of Ministry of Science and Technology, Taiwan, R.O.C., and the assistance by Prof. Y.L. Tsay are greatly appreciated.

REFERENCES

1. Gordon R, Cobonpue J. Heat transfer between a flat plate and jets of air impinging on it. In: International Developments in Heat Transfer: Part 2. New York: ASME, 1961, 454–460.
2. Choi M, Yoo HS, Yang G, Lee JS, Sohn DK. Measurements of impinging jet flow and heat transfer on a semi-circular concave surface. International Journal of Heat and Mass Transfer 2000;43(10):1811–1822.
3. Lee DH, Song J, Jo MC. The effect of nozzle diameter on impinging jet heat transfer and fluid flow. Journal of Heat Transfer 2004;126(4):554–557.
4. Sharif MAR, Mothe KK. Parametric study of turbulent slot-jet impingement heat transfer from concave cylindrical surfaces. International Journal of Thermal Sciences 2010;49(2):428–442.
5. Singh D, Premachandran B, Hohi S. Numerical simulation of the jet impingement cooling of a circular cylinder. Numerical Heat Transfer, Part A: Applications 2013;64(2):153–185.
6. Isman MK, Pulat E, Etemoglu AB, Can M. Numerical investigation of turbulent impinging jet cooling of a constant heat flux surface. Numerical Heat Transfer, Part A: Applications 2008;53(10):1109–1132.
7. Heo MW, Lee KD, Kim KY. Parametric study and optimization of staggered inclined impinging jets on a concave surface for heat transfer augmentation. Numerical Heat Transfer, Part A: Applications 2012;61(6):442–462.
8. Baghel K, Sridharan A, Murallidharan JS. Heat transfer characteristics of free surface water jet impingement on a curved surface. International Journal of Heat and Mass Transfer 2021;164:120487.
9. Ram Kumar BVN, Prasad BVSS. Computational flow and heat transfer of a row of circular jets impinging on a concave surface. Heat and Mass Transfer 2008;44(6):667–678.
10. Craft TJ, Iacovides H, Mostafa NA. Modeling of three-dimensional jet array impingement and heat transfer on a concave surface. International Journal of Heat and Mass Transfer 2008;29:687–702.
11. Sahoo D, Sharif MAR. Mixed-convective cooling of an isothermal hot surface by confined slot jet impingement. Numerical Heat Transfer, Part A: Applications 2012;65(9):887–909.
12. Guan T, Zhang JZ, Shan Y. Conjugate heat transfer on leading edge of a conical wall subjected to external cold flow and internal hot jet impingement from chevron nozzle—part 2: numerical analysis. International Journal of Heat and Fluid Flow 2017;106:339–355.
13. Brown JM, Raghunathan S, Watterson JK, Linton AJ, Riordon D. Heat transfer correlation for anti-icing systems. Journal of Aircraft 2002;39(1):65–70.
14. Fregeau M, Gabr M, Saeed F, Paraschivioiu I. Numerical simulation of heat transfer from an array of hot-air jets impinging on 3D concave surface. In: 50th CASI Annual General Meeting & Conference, Montreal, April 2003.
15. Hua J, Liu HHT. Fluid flow and thermodynamic analysis of a wing anti-icing system. Canadian Aeronautics and Space Journal 2005;51(1):35–40.
16. Liu HHT, Hua J. Three-dimensional integrated thermodynamic simulation for wing anti-icing system. Journal of Aircraft 2004;41(6):1291–1297.
17. Fregeau M, Saeed F, Paraschivioiu I. Numerical heat transfer correlation for array of hot-air jets impinging on 3-dimensional concave surface. Journal of Aircraft 2003;40(3):665–670.
18. Saeed F. Numerical simulation of surface heat transfer from an array of hot-air jets. Journal of Aircraft 2008;45(2):700–714.
19. Fregeau M, Gabr M, Paraschivioiu I, Saeed F. Simulation of heat transfer from hot-air jets impinging a three-dimensional concave surface. Journal of Aircraft 2009;46(2):721–726.
20. Mu Z, Lin G, Shen X, Bu X, Zhou Y. Numerical simulation of unsteady conjugate heat transfer of electrothermal deicing process. International Journal of Aerospace Engineering 2018;2018: article ID 5362541.
21. Khalil EE, Said E, AlSaheh A, ElHarriry G. Effect of hot air jet arrangement from a piccolo tube in aircraft wing anti-icing system. In: AIAA Propulsion and Energy 2020 Forum. Reston, VA: AIAA, 2020, 3952–3952.
22. Yang B, Chang S, Wub H, Zhao Y, Leng M. Experimental and numerical investigation of heat transfer in an array of impingement jets on a concave surface. Applied Thermal Engineering 2017;27:473–483.
23. Sarkar S, Balakrishnan L. Application of a Reynolds-stress turbulence model to the compressible shear layer. ICASE Report 90-18, NASA CR 182002, 1990.
24. Patankar SV. Numerical Heat Transfer and Fluid Flow. New York: McGraw-Hill, 1980.
25. Van Doormaal JP, Raithby GD. Enhancements of the SIMPLE method for predicting incompressible fluid flows. Numerical Heat Transfer 1982;7(2):147–163.