Temperature and Runback Ice Prediction Method for Three-Dimensional Hot Air Anti-Icing System

Ying Zhou¹, Guiping Lin¹, Xueqin Bu¹,³,⁴, Zuodong Mu¹, Rui Pan¹, Qimo Ge² and Xudong Qiao²

¹School of Aeronautic Science and Engineering, Beihang University, Beijing 100191, China
²Jiangxi Hongdu Aviation Industry Group Corporation Limited, AVIC, Nanchang, Jiangxi, 330024, China
³bxueqin@buaa.edu.cn

Abstract. A prediction method of surface temperature and runback ice for a three-dimensional hot air anti-icing system was proposed. Computational approach to realize this method was introduced. Both the external and internal flows were separately calculated, results of which were set as boundary conditions of heat conduction computation in airfoil skin. The results of external and internal flow calculations show that the effect of surface temperature on convective heat transfer coefficients and local droplet collection efficiency is negligible and the calculations can be decoupled. The prediction method based on heat flux was used to calculate surface temperature and runback ice results. The results show that, the effects of LWC and Mach number are much more significant than the effect of external flow temperature. The surface temperature at impinging interaction point is more sensitive to the change of external conditions than that at stagnation point. The surface temperature changes significantly with changing Mach number because both the mass rate of droplet and the impact limit are changed.

1. Introduction

Ice accretion on the aircraft surface often leads to both weight and drag increasing, lift reduction and a degradation of the aerodynamic and handling performance, therefore, is a safety hazard for flight. To protect aircraft from such a hazard, anti-icing systems are employed. One of the most widely used anti-icing systems on commercial aircrafts is the hot air anti-icing system, which uses the high-pressured hot air, bleeding from the compressor of the engine, to heat the wing leading edge. The hot air is transported through small round holes in a piccolo tube installed inside the anti-icing cavity.

In order to comprehensively understand the hot air anti-icing process, a lot of valuable researches has been done for the last decades. Those studies were mainly concentrated in the following aspects: 1, droplet collection characteristics and water film models; 2, parametric investigation of the piccolo tube and anti-icing cavity; 3 simulation method to predict the surface temperature and runback ice.

Both Lagrange [1] and Euler [2] method were utilized to calculate the water droplet trajectory. Water film models were investigated by Al-Khalil [3] and Myers [4] to obtain more accurate and physical descriptions of the water film motion process in the latter case.

Saeed [5, 6] simulated the thermal performance of 3-D hot-air array jetting on concave surfaces by using normal semi-circular concave surfaces instead of anti-icing cavity. Extensive experiments were conducted by Papadakis and See-Ho [7] to investigate the system performance and efficiency.
Three-dimensional hot air anti-icing system was studied computationally under dry air flow condition by Hugh H. T. Liu [8, 9] and See-Ho Wong [10]. The external and internal flows in their researches were coupled by heat conduction through the solid skin. Two-dimensional system with wet surface was computed by Francois Morency [11] and Rodrigo H. Domingos [12] using decoupled computations of external and internal flows. Lee [13] developed a procedure to integrate the internal and external solutions in Three-dimensional system.

A prediction method of the surface temperature and runback ice for three-dimensional hot air anti-icing system was proposed in this paper. Following the method applied in previous researches, both the external and internal flows were separately calculated, and the calculation results were used as boundary conditions of heat conduction computation of airfoil skin. By calculating the heat conduction iteratively, the results of temperature and runback ice were obtained.

2. Geometric model

Figure 1 shows a geometry model of the present CFD simulation. The three-dimensional wingspan segment is composed from a 90-mm span, 1800-mm chord NACA 23012 airfoil. The hot air anti-icing system is inserted into the first 160-mm chord of the wing. The thickness of the leading edge skin is 1.6 mm.

Figure 2 shows details of the piccolo tube in the anti-icing cavity, which has two horizontal rows of 1.32mm diameter round holes. The adjacent spacing between two holes along spanwise is 30 mm. The rows are placed at 45 degrees circumferentially above and below the piccolo tube axis. Two thin channels (1.27mm) at both upper and lower sides of the leading edge skin allow hot air to exit the cavity.

3. Methodology

The surface temperature prediction process was divided into three regions: the external flow region including dry air convection heat transfer and water droplets impingement, the internal flow region of piccolo tube heat transfer with the inner surface, and the airfoil skin region conducting heat from internal flow to external.

The external flow and internal flow were computed separately by a commercial CFD flow solver. The results of the convective heat transfer coefficient of both external and internal skin surface and the local collection efficiency of droplet impingement were stored respectively to provide boundary conditions to airfoil skin thermodynamic equilibrium. Detailed introduction of the external and internal flow computations were given in [14, 15]. The present work was mainly focused on the airfoil skin thermodynamic equilibrium consisting of the heat conduction in the skin and the heat transfer of the phase-changing water film over it.

The conservation equations for thermodynamic equilibrium of phase-changing water film considering runback ice are as follows.

3.1. Mass conservation equation
For a water film control volume (CV) on the leading edge skin shown in Figure 3, the mass flow of water flowing into the CV is equal to the quality of the water flowing out. The mass conservation equation can be established as equation (1):

$$\dot{m}_{\text{in}} + \dot{m}_{\text{imp}} \cdot \Delta s = \dot{m}_{\text{evap}} \cdot \Delta s + \dot{m}_{\text{out}} + \dot{m}_{\text{ice}} \cdot \Delta s$$

\(\dot{m}_{\text{in}}, \dot{m}_{\text{out}}\) represent the water mass flow into and out of the control volume respectively, \(\text{kg/s}\). \(\dot{m}_{\text{evap}}\) is the rate of water evaporating, \(\text{kg/s} \cdot \text{m}^2\). \(\dot{m}_{\text{imp}}\) is the rate of droplets impinging on the leading edge surface, \(\text{kg/s} \cdot \text{m}^2\). \(\dot{m}_{\text{ice}}\) represents the mass rate of water freezing, \(\text{kg/s} \cdot \text{m}^2\).

![Figure 3 Mass and energy fluxes](image)

3.2. Energy conservation equation

According to the steady state energy balance for the water film control volume on the external surface, the energy gets in and energy gets out of the control volume are equal. So the conductive heat flux \(q_s\) conducted from the leading edge skin keeping the control volume thermodynamic equilibrium can be calculated by equation (2):

$$\dot{q}_{\text{in}} + \dot{q}_{\text{ic}} + \dot{q}_{\text{c}} + \dot{q}_{\text{evap}} + \dot{q}_{\text{w}} = \dot{q}_{\text{out}} + \dot{q}_{\text{w}} + \dot{q}_{\text{evap}} + \dot{q}_{\text{c}}$$

\(\dot{q}_{\text{in}}\) and \(\dot{q}_{\text{out}}\) are the energy flow into and out of the CV. \(\dot{q}_{\text{ic}}\) is the kinetic energy released by the droplets at impingement. \(\dot{q}_{\text{c}}\) is the latent solidification heat flux of the water. \(\dot{q}_{\text{evap}}\) is the heat exchange released when the ice temperature is below freezing point. \(\dot{q}_{\text{w}}\) is the convective heat flux to the external flow. \(\dot{q}_{\text{evap}}\) is the latent evaporation heat flux of the water. \(\dot{q}_{\text{c}}\) is the heat exchange absorbed when the water temperature is above freezing point. The calculation formulae of each heat flux term were elaborately recommended in [15].

After impinging on the leading edge, water droplet is partially vaporized in the protected area, while the rest droplets flow along the airflow on the surface and may freeze downstream. Therefore, the calculation area can be divided into three cases: temperature above freezing point, freezing point and temperature below freezing point:

1. \(T_s > 273.15\text{K}\)

When the leading edge external surface temperature is higher than freezing point, there is no frozen water. \(\dot{m}_{\text{ice}} = 0, \dot{q}_{\text{ice}} = 0\).

$$\dot{q}_{\text{in}} + \dot{q}_{\text{ic}} = \dot{q}_{\text{c}} + \dot{q}_{\text{evap}} + \dot{q}_{\text{w}} + \dot{q}_{\text{out}}$$

(3)

2. \(T_s = 273.15\text{K}\)

When the leading edge external surface temperature is equal to freezing point, both water and ice exist on the surface. \(\dot{q}_{\text{ic}} = \dot{q}_{\text{out}} = 0\).

$$\dot{q}_{\text{in}} + \dot{q}_{\text{ic}} + \dot{q}_{\text{c}} = \dot{q}_{\text{evap}} + \dot{q}_{\text{w}}$$

(4)

3. \(T_s < 273.15\text{K}\)
When the leading edge external surface temperature below freezing point, the heat release by water condensation and ice cooling must be considered. Due to completely freezing on the surface, there is no outflow of water and energy. \( n_{\text{out}} = 0 \), \( q_{\text{out}} = 0 \).

\[ q''_{\text{n}} + q''_{\text{c}} + q''_{\text{in}} + q''_{\text{ice}} + q''_{\text{w}} = q''_{\text{c}} + q''_{\text{evap}} + q''_{w} \]  \tag{5}

4. Numerical solution method

According to Section 3, the water state on the anti-icing surface can be determined by the conductive heat flux \( q''_{\text{n}} \) required for thermodynamic equilibrium of the water film independently. A numerical method based on heat flux \( q''_{\text{n}} \) was proposed in the present study. The method is that 1) use the conductive heat flux \( q''_{\text{n}} \) obtained from the heat conduction of the leading edge skin to calculate the surface temperature via the mass and energy conservation equations of the water film, 2) set the temperature as the Dirichlet boundary condition for the heat conduction calculation to update \( q''_{\text{n}} \), 3) calculate iteratively until the result converges.

It is necessary to evaluate the water state on the anti-icing surface before calculating. Assuming \( T_s = 273.15 \text{K} \), the required \( q''_{\text{n}} \) for \( f = 1 \) and \( f = 0 \) are determined respectively.

For \( f = 1 \):

\[ q''_{n1} = q''_{c} + q''_{\text{evap}} + q''_{w} - q''_{k} - q''_{\text{in}} - q''_{\text{ice}} \]

For \( f = 0 \):

\[ q''_{n2} = q''_{c} + q''_{\text{evap}} + q''_{w} - q''_{k} - q''_{\text{in}} \]

By comparing the known conductive heat flux \( q''_{n} \) with \( q''_{n1} \) and \( q''_{n2} \), the surface temperature on a control volume can be divided into three cases:

1. \( q''_{n} < q''_{n1} \), the conductive heat flux is so small that the water is completely frozen. \( f = 1 \), \( T_s < 273.15 \text{K} \);
2. \( q''_{n} > q''_{n2} \), the conductive heat flux is large enough to dry up the surface. \( f = 0 \), \( T_s > 273.15 \text{K} \);
3. \( q''_{n1} < q''_{n} < q''_{n2} \), \( T_s = 273.15 \text{K} \).

5. Results

5.1. External flow region

The Reynolds-averaged Navier-Stokes equations and the Transition SST turbulence model were applied to compute the external air flow to gain the convection heat transfer on the airfoil skin. Structured meshes involving hexahedral elements were used to model the external flow by Pointwise as shown in Figure 4. Pointwise is a commercial software to generate high quality orthogonal grids. The wall-adjacent cell distance (cell center-to-wall distance) in terms of the non-dimensional distance \( y^+ \) was kept below 1. Pressure-far-field boundary condition was used for the air flow with an air pressure of 101,325\,\text{Pa}, a temperature of 263\,\text{K} and a Mach number of 0.2. The airfoil wall was modeled by using the isothermal wall boundary condition with \( T_{\text{wall,ex}} = 270\,\text{K}, 280\,\text{K}, 290\,\text{K}, 300\,\text{K}, 310\,\text{K}, 320\,\text{K}, 330\,\text{K}, 340\,\text{K} \). The convective heat transfer coefficient \( h_{\text{ex}} \) can be obtained by equation(6):

\[ h_{\text{ex}} = q_{\text{ex}} / (T_{\infty} - T_{\text{ad}}) \]  \tag{6}

\( T_{\text{ad}} \) is the local temperature when the airfoil is under adiabatic condition, \( K \). \( q_{\text{ex}} \) is the surface heat flux to the external flow. \( T_{\infty} \) is the external flow temperature, \( K \).

The distributions of \( h_{\text{ex}} \) for different \( T_{\text{wall,ex}} \) are presented in Figure 6. The results indicate that the effect of external surface temperature on the convective heat transfer coefficient under the same inlet air condition (air speed, temperature and pressure) is negligible. Thus, the convection heat transfer computation can be decoupled, and the convective heat flux to the external flow in equation (2) can be calculated by:

\[ q_{c} = (T_{\infty} - T_{\text{ad}}) \cdot h_{\text{ex}} \]  \tag{7}
The droplet collection characteristics were calculated by Euler method based on the previous external air flow field. The distributions of local droplet collection efficiency $\beta$ for different $T_{\text{wall}}$ with $Ma = 0.2, 0.3$ and MVD $= 20\mu m$ were shown in Figure 7. The effect of $T_{\text{wall}}$ on $\beta$ is also negligible, and the droplet collection characteristics computation can be decoupled as well seen from Figure 7.

Figure 4 Grids of external flow

Figure 5 Grids of internal flow

Figure 6 External convective heat transfer coefficient

Figure 7 Local droplet collection efficiency

5.2. Internal flow region

The Realizable $k$-$\varepsilon$ turbulence model with near-wall treatment method was conducted to calculate the convection heat transfer of jets impinging on the anti-icing cavity inner surface. Unstructured meshes were generated by ICEM software (in Figure 5) and the non-dimensional distance was $30 < y^+ < 300$. The hot air inlet used the mass-flow-inlet boundary with a 0.003 kg/s mass flow-rate and a 453K total temperature. The exit flow was defined as a pressure-outlet boundary with the pressure of 101,325 Pa. The impingement wall was modeled isothermal with $T_{\text{wall,in}} = 270K, 280K, 290K, 300K, 310K, 320K, 330K, 340K$, respectively. The convective heat transfer coefficient $h_{\text{jet}}$ can be calculated as follows:

$$h_{\text{jet}} = \frac{q_{\text{jet}}}{(T_{\text{jet,tol}} - T_{\text{wall,in}})}$$

$q_{\text{jet}}$ is the heat flux provided by the impinging jets, W/m$^2$. $T_{\text{jet,tol}}$ is the total temperature in the piccolo tube, K. $T_{\text{wall}}$ is the temperature of the inner surface, K.

The distributions of $h_{\text{jet}}$ for different $T_{\text{wall,in}}$ were shown in Figure 8. It is indicated that the effect of inner surface temperature on $h_{\text{jet}}$ is negligible. So $h_{\text{jet}}$ and $T_{\text{jet,tol}}$ can be used as the Robin boundary condition for a given hot air property in the piccolo tube (including the total temperature and mass flow rate). As a consequence, the internal flow computation can be decoupled.
5.3. Prediction of temperature and runback ice

The performance of the simulation method based on heat flux was evaluated by analyzing the temperature and runback ice results under different conditions. The external and internal conditions used in the validation cases are summarized in Table 1.

| case | Mach number | Temperature (K) | MVD (μm) | LWD (g/kg) | Total temperature (K) | Mass flow rate (kg/s) |
|------|-------------|-----------------|----------|------------|-----------------------|----------------------|
| 1    | 0.2         | 263             | 20       | 1          | 453                   | 0.003                |
| 2    | 0.2         | 263             | 20       | 2          | 453                   | 0.003                |
| 3    | 0.3         | 263             | 20       | 1          | 453                   | 0.003                |
| 4    | 0.3         | 263             | 20       | 2          | 453                   | 0.003                |
| 5    | 0.3         | 263             | 20       | 3          | 453                   | 0.003                |
| 6    | 0.3         | 258             | 20       | 3          | 453                   | 0.003                |
| 7    | 0.3         | 268             | 20       | 3          | 453                   | 0.003                |

The surface temperature distribution is presented in Figure 9. The maximum temperature is observed at the stagnation point of the internal impingement contributed by its intense heat transfer performance. The runback water freezes on the surface out of the protected area as shown in Figure 10. $H_{\text{ice}}$ is the ice accretion rate calculated as follows:

$$H_{\text{ice}} = \frac{m''_{\text{ice}}}{\rho_{\text{ice}}}$$

where $\rho_{\text{ice}}$ is the density of ice, kg/m$^3$.
5.3.1. Effect of LWC. The temperature distributions for different LWC are shown in Figure 11. When LWC increases from 1.0 g/kg to 2.0 g/kg, temperature changes at the stagnation points are $\Delta T_1 = 3.04K$ and $\Delta T_2 = 4.04K$, at the impinging interaction point ($s = 0$) is $\Delta T_3 = 5.36K$. When LWC increases to 3.0 g/kg, temperature changes are $\Delta T_1 = 6.05K$, $\Delta T_2 = 8.06K$ and $\Delta T_3 = 10.08K$. It is indicated that the effect of LWC on temperature at the interaction point is greater than that at the stagnation point.

![Figure 11](image1.png)  
**Figure 11** $T_s$ distributions for varying LWC

![Figure 12](image2.png)  
**Figure 12** $T_s$ distributions for varying external flow temperature

5.3.2. Effect of external flow temperature. Figure 12 shows the temperature distributions under different external flow temperatures. Compared with LWC, the effect of external flow temperature is much smaller. The temperature at impinging interaction point increases by $\Delta T_3 = 2.98K$ and 6K respectively with the external flow temperature rising from 258K to 263K and 268K.

5.3.3. Effect of Mach number. Figure 13 presents the temperature distributions for different Mach number. The effect of Mach number on surface temperature is significant seen from the figure. The surface temperature drops by $\Delta T_1 = 4.7K$, $\Delta T_2 = 4.75K$ and $\Delta T_3 = 5.2K$ when Mach number increases from 0.2 to 0.3 for LWC = 1.0 g/kg. And the temperature drops by $\Delta T_1 = 5.89K$, $\Delta T_2 = 6.18K$ and $\Delta T_3 = 6.8K$ for LWC = 2.0 g/kg. The maximum change of temperature occurs at $s = -0.14mm$ with a change of 18.01K and 18.12K for LWC = 1.0 g/kg and 2.0 g/kg. It is because both the mass rate of droplet and impact limit are changed with Mach number as shown in Figure 7. So does the heat transfer coefficient of the external surface.

![Figure 13](image3.png)  
**Figure 13** Distributions of temperature for different Mach number

6. Conclusion
A prediction method of surface temperature and runback ice for a three-dimensional hot air anti-icing system was proposed and was implemented based on the thermodynamic equilibrium of the water film and airfoil skin heat conduction.

To evaluate the performance of the simulation method presented in this paper, external and internal flow computations were conducted under different skin temperature. The results show that the effect of skin temperature is negligible so that the calculations can be decoupled. Temperature and runback ice results under different external conditions were obtained and analyzed to verify the effectiveness and rationality of the method.

The results indicate that the effects of LWC and Mach number are much more significant than external flow temperature. The surface temperature at impinging interaction point is more sensitive to the change of external conditions than that at stagnation point. The surface temperature changes a lot with changing Mach number because both the mass rate of droplets and impact limit are changed.

7. References
[1] Wright W. B. Users Manual for the Improved NASA Lewis Ice Accretion Code LEWICE 1.6[C]. June 01. 1995. NASA-Contractor Report-198355
[2] Bourgault Y., Boutanios Z., Habashi W.G. Three-Dimensional Eulerian Approach to Droplet Impingement Simulation Using FENSAP-ICE, Part 1: Model, Algorithm, and Validation[J]. Journal of Aircraft. 2000. 37(1): 95-103
[3] K. M. Al-Khalil, T. G. Keith Jr., K.J. De Witt. Development of an Anti-Icing Runback Model[C]. 28th Aerospace Sciences Meeting. 1990. AIAA 90-0759
[4] Tim G. Myers, Chris P. Thompson. Modeling the Flow of Water on Aircraft in Icing Conditions[J]. AIAA Journal, Vol. 36. No. 6 (1998), pp. 1010-1013
[5] Mathieu Fregeau. F. Saeed. I. Paraschivoiu. Surface Heat Transfer Study for Ice Accretion and Anti-Icing Prediction in Three Dimension[C]. 42nd AIAA Aerospace Sciences Meeting and Exhibit. 2004. AIAA 2004-0063
[6] Mathieu Fregeau. F. Saeed. I. Paraschivoiu. Numerical Heat Transfer Correlation for Array of Hot-Air Jets Impinging on 3-Dimensional Concave Surface[J]. Journal of Aircraft. 2005. Vol. 42 No. 3: 665-670
[7] M. Papadakis, S.J. Wong, H.W. Yeong, S.C. Wong, Icing tunnel experiments with a hot air anti-icing system[J], AIAA paper 2008-444, 2008
[8] Hugh H. T. Liu and Jun Hua. Three-Dimensional Integrated Thermodynamic Simulation for Wing Anti-Icing System[J]. Journal of Aircraft. Vol. 41. No. 6, November–December, 2004
[9] Jun Hua and Hugh H. T. Fluid Flow and Thermodynamic Analysis of a Wing Anti-Icing System[J]. Canadian Aeronautics and Space Journal. Vol. 51. No. 1, March 2005
[10] See-Ho Wong, Michael Papadakis, Alonso Zamora. Computational Investigation of a Bleed Air Ice Protection System[C]. 1st AIAA Atmospheric and Space Environments Conference. 22 - 25 June 2009, San Antonio, Texas. AIAA 2009-3966
[11] Francois Morency, Fatih Tezok, Ion Paraschivoiu. Anti-Icing System Simulation Using CANICE[J]. Vol. 36, No. 6, November–December 1999
[12] Rodrigo H. Domingos, Michael Papadakis, Alonso O. Zamora. Computational Methodology for Bleed Air Ice Protection System Parametric Analysis[J]. AIAA Atmospheric and Space Environments Conference. 2 - 5 August 2010, Toronto, Ontario Canada. AIAA 2010-7834
[13] Lee, J., Rigby, D., Wright W. and Choo, Y. Analysis of Thermal Ice Protection System (TIPS) with Piccolo Tube using State-of-the-Art Software[C]. 44th AIAA Aerospace Sciences Meeting and Exhibit. 9 - 12 January 2006, Reno, Nevada. AIAA 2006-1011
[14] Bu Xue-qin, Lin Qui-ping, Peng You-xin, Yu Jia. New Method for Calculation of Anti-icing Heat Loads[J]. Chinese Journal of Aeronautics. Vol. 127 No. 12. March 2006
[15] Bu Xue-qin, Lin Qui-ping, Yu Jia. Three-dimensional conjugate heat transfer simulation for the surface temperature of wing hot-air anti-icing system[J]. Journal of Aerospace Power. Vol .24 No .11. Nov .2009