A numerical study of the anti-icing heat load for a three-dimensional aircraft wing

Zhangsong Ni 1, Senyun Liu 2, Jun Zhang 2, *, Mao Wang 2, Zixu Wang 2

1 School of Aeronautics, Northwestern Polytechnical University, Xi’an 710072, China
2 State Key Laboratory of Aerodynamics, China Aerodynamics Research and Development Center, Mianyang 621050, China

*Corresponding author e-mail: jzhang@nudt.edu.cn

Abstract. In the design stage of an anti-icing system for an aircraft, the heat load on a protection surface must be correctly calculated. In this study, the heat load calculation method for an aircraft wing model was analyzed. The calculation process was presented and the correctness of the method was validated using existing published experimental data. And the effect of variation of incoming flow speeds, flight altitudes, and temperatures on the surface heat-load distribution was given. The results show that the heat load is mainly located at the leading edge of the wing. The maximum value of the heat load can reach 3500w/m² and the closer to the tip of the wing, the larger the heat load value will be. The heat load curve has a trough in the middle and two shoulders at each side. The range of the non-zero region and the amplitude of the heat load is in proportion to the flow speed. However, the variation of altitude has a non-obvious effect on the heat load results. As the temperature rises, the amplitude of the heat load increases, but the distribution characteristic of the curve remains the same.

1. Introduction

Ice will be growing at the leading edge of the wing, at the lip of the engine inlet of an aircraft when the aircraft passes through the cloud that contains super cold liquid droplets (SLD). As a result, the aerodynamics, the manipulation, and the stability of an aircraft can be greatly affected [1-3]. Therefore, protecting the aircraft from icing is greatly concerned by aircraft manufactures and research institutes. To protect the aircraft from icing, the large civil aircraft usually installs anti-icing systems. Based on the strategy, the anti-icing method for aircraft can be divided as mechanical, hydro-dynamic, gas thermal and electrical thermal, etc. [4-6]. Currently, commercial aircraft mainly use gas thermal or electrical thermal anti-icing systems. In the design stage of the gas thermal system, the protection area and the energy needed to reach a certain temperature must be firstly determined [7-9]. So precisely choosing a reasonable condition and calculating the surface anti-icing heat load is very important.

Wu [10] etc. numerically simulated the jet using three dimensional Navier-Stokes equation, and the surface temperature distribution at different jet angles was obtained. Liu [11] studied the 3D numerical model for the anti-icing cavity for an aircraft wing. He presented a completed gas thermal anti-icing model for the cavity and calculated the coupled flow field of the inside and the outside cavity. The variation of the surface temperature and the convective heat-exchange coefficient (CHC) under different outer environment conditions was calculated and analyzed. Planauart [12] studied the gas thermal anti-
icing system for the leading edge of a wing experimentally. The hot gas was ejected on the wing surface through holes, and the surface temperature was measured using infrared detectors. The distribution of the surface CHC was obtained. Later on, a new calculation method was presented by Planauart. And his method was validated by comparing the experimental results with the calculations. Chang [13] calculated the protection range of the anti-icing and de-icing system of a helicopter rotor, and the relationship between surface temperature and control law was revealed.

In this study, the anti-icing heat load for an aircraft wing model was calculated. Firstly, the calculation method and the procedure were introduced. Secondly, based on the real flight condition and the wing model, the effect of environmental parameters, including the incoming flow speed, the flight altitude, and the temperature on the distribution and the amplitude of the heat load was presented. And the selection principle of the calculation condition point of anti-icing heat load was given. The study is meaningful for the design of a gas thermal anti-icing system for aircraft.

2. Calculation method for anti-icing heat load

Based on the operation condition of the anti-icing system, the physical process of SLD colliding with the wing surface varies. Under the dry-ice condition, the liquid droplets evaporate immediately. However, under wet-ice condition, the liquid droplets are in liquid form after a collision. And the liquid water evaporates during the process of flowing. To simulate the physical process after the collision of SLD with the wing surface, the surface was divided into multiple control volumes. P represents the objective grid, and N, E, W, S are the adjacent grids. And n, e, w, s represents the corresponding front edge.

![Figure 1. A sketch of the surface grid](image)

Taking a control volume as an example, the mass equation of the liquid can be given as:

$$\dot{m}_{in} + \sum \dot{m}_{in} - \dot{m}_{exp} - \sum \dot{m}_{out} = 0$$  \hspace{1cm} (1)

Using the entropy to represent the heat flux, and for each control volume, the energy conservation equation is written:

$$\dot{q}_{\text{required}} + \dot{q}_{\text{clt}} + \sum \dot{q}_{\text{in}} - \dot{q}_{\text{exp}} - \sum \dot{q}_{\text{out}} = 0$$  \hspace{1cm} (2)

Here, \( \dot{m} \) stands for mass flux, and the footnotes clt represents collection term, in for inflow water term, evp for evaporation term, out for outflow water term, cnv for the convective heat-exchange term. \( \dot{q} \) stands for the heat flow density, and required represents anti-icing term inside the control volume.

The CHC at the stagnation point can be calculated using the approximation method given by Smith and Spalding [14]:

$$Nu_{stag} = \left[ \frac{C^2}{9.89} \frac{Re_{\text{air}}}{Re_{\text{critical}}} \frac{d(u_{stag}/u_{stag})}{d(s/c)} \right]^{0.15}$$  \hspace{1cm} (3)
$$N_{u_{stag}} = \frac{h_{stag} \cdot c}{k_a}$$  \hspace{1cm} (4)$$

Where $h_{stag}$ stands for the CHC at the stagnation point, $N_{u_{stag}}$ is the Nusselt number, $C_m$ is a constant, which equals 1.56. $Re$ is the Reynolds number in air, and $u_e$ is the flow speed outside of the boundary layer. $s$ is the span-wise position, $c$ is the chord length, and $k_a$ is the heat convection ratio in air.

Under a dry-ice condition, the liquid water evaporates immediately, and no water flow happens. Therefore, $\sum m_{in}$ and $\sum m_{out}$ both equal zero, and the mass equation can be simplified as

$$\dot{m}_{clt} - \dot{m}_{exp} = 0$$  \hspace{1cm} (5)$$

Using the above equation, the surface evaporation content, and the temperature can be obtained directly. And the surface heat load can be solved using the energy conservation equation.

For wet ice, the solution steps for the above control equations are summarized:

Step 1, set the objective temperature on the heating surface;

Step 2, assume the initial value of $\sum m_{in}$ equals zero and $\sum m_{out}$ can be solved using the mass equation. Then calculate $\dot{m}_{out_{water}}$ and $\dot{m}_{out_{water}}$ respectively based on the mass allocation principle of the overflow, and the residual value of the overflow.

Step 3, according to the relationship of the mass of water that flows into the control volume and that floats out of its adjacent control volume, and re-allocate the mass that flows into the control volume.

Step 4, repeat step 2 and step 3 until the residual value of the overflow water satisfactorily converges.

Step 5, substitute the necessary terms obtained in the mass equation into the energy equation and solve the anti-icing heat load directly.

3. Validation of the method

To validate the method for the surface heat-load calculation, the electric-thermal anti-icing experiment conducted in the icing wind tunnel (IWT) of NASA Glenn research center by Al-Khalil [15, 16], etc. was taken as a reference. To compare the experimental data and calculation results given by Al-Khalil and the numerical results in this study, the presented method in section 2 can be validated. In this calculation, a NACA0012 airfoil is selected. The chord length is 0.9144m, and the span length is 1.8288m. At the leading edge of the airfoil, an electric-thermal anti-icing system is installed. The electric units can be divided into 7 heating areas and the power of heating can be controlled to change the surface temperature distribution. The inflow flow speed is 89.408m/s, the temperature of the air is -21.8oC, the angle of attack is 0 oC, the liquid water content is 0.55g/m3, and the diameter of liquid water is 20μm.

![Figure 2. Calculation results of temperature distribution on the surface of a NACA0012 airfoil (left) and the heat load curve comparison (right).](image)

The temperature distribution on the wing surface is shown in Fig.2. It’s obvious that the calculation results using the present method agree well with the experimental data and the reference results. Thus,
the calculation method in this study is validated. In the heat load curve, the impingement of SLD on the surface of the wing is within ±0.01s/c. Due to the high power used by the heating units in this region, one part of the liquid droplets evaporate and the other part floats towards the downstream. Therefore, the temperature in this region is relatively low. Within -0.15s/c and -0.01s/c, the water content reduces as the liquid water evaporates, and the temperature increases gradually. And from 0.10s/c to 0.15s/c, the surface temperature decreases as the electrical power drops. Larger than 0.15s/c, the surface temperature is the same as the environmental temperature since there’re no heating units and liquid water. Meanwhile, as the heating units distribute unevenly on the upper and the lower side of the wing, the temperate varies accordingly. Under the current condition, the heating power is small. As a result, the whole temperature distribution is relatively small. The temperature value is between 275k and 285k, which prevents the growth of ice on the wing and helps save on-board energy.

4. Numerical calculations

4.1. Calculation of the heat load on an aircraft wing
In this study, an aircraft wing model is chosen to calculate the heat load using the method proposed in section 2. The calculation conditions are as follows: the angle of attack 5°, the flight height 1.524km, the flight speed 302.4km/h, the environmental temperature -5°C, the liquid water content (LWC) 0.531g/m³, and the mean liquid water diameter (MVD) 20μm. The heat load calculation process includes the grid generation, the flow field calculation, the trajectory calculation, and the heat-load calculation, etc. The geometric model of the wing is given in Fig.3, and the trailing edge of the wing is swept. ICEM software is used to generate the grid, and the flow field calculation uses FLUENT. The pressure-based solver is selected and k-ω SST turbulent model is used. The trajectory of the water droplet is calculated using the UDF function in FLUENT [17], which is developed based on the movement control equations. Finally, the heat load on the wing surface is calculated using an in-house developed code [18].

![Figure 3. Computation grid for a 3D aircraft wing](image)

The wet condition is chosen in the computation, and the surface temperature is 300K. In Fig.4 and Fig.5, the distribution map and the curve of the surface heat load is shown. It can be seen that the heat load is mainly located at the leading edge of the wing. And the maximum value of the heat load is 35000w/m². The closer to the tip of the wing, the larger the heat load value is. The heat load distribution can be clearly seen from the curve. The curve has a trough in the middle and two shoulders at each side, which can be explained by the fact that the SLD doesn’t evaporate immediately after impinging on the wing surface. At this location, the heat load value can be kept low enough, at which the water remains in the liquid state. This part of liquid water floats downstream with the flow. Behind the stagnation point, the heat units not only need to vaporize the liquid water coming from upstream but also need to make sure that the SLD won’t freeze. Therefore, the heat load value (the energy input) is large. As the liquid water keeps moving downward, the mass reduces. Thus, the heat load value decreases. It’s shown that
the heat load is large close to the tip of the wing. This trend agrees well with the water collection curve, which implies that more heat load power is needed for a large amount of water collection. The overall heat load is 21.3kw, which is integrated over the wing surface.

![Figure 4. Anti-icing heat load distribution map for a 3D wing](image1)

![Figure 5. Anti-icing curve at different cross-sections](image2)

4.2. The effect of environmental parameters on the heat load

The same aircraft wing model is used here, and the surface temperature remains 300K. The grid condition keeps unchanged and the boundary conditions vary. The computation state points are listed in table 1. The effect of environmental parameters, such as the incoming flow speed, the flight altitude, and the temperature, on the distribution of the heat load is presented.

| NO. | Flight Condition | Icing condition |
|-----|------------------|-----------------|
|     | Height, H (m)    | Speed, V (m/s)  | Temperature, T (°C) | LWC (g/m³) | MVD (μm) |
| 1   | 1500             | 84              | -5                | 0.531     | 20       |
| 2   | 1500             | 60              | -5                | 0.531     | 20       |
| 3   | 1500             | 124             | -5                | 0.531     | 20       |
| 4   | 4000             | 84              | -5                | 0.531     | 20       |
| 5   | 6500             | 84              | -5                | 0.531     | 20       |
| 6   | 1500             | 84              | -15               | 0.531     | 20       |
| 7   | 1500             | 84              | -25               | 0.531     | 20       |

The heat load vs. the non-dimensional distance parameter s/c is given at the location of 80% span length is shown in Fig.6. It can be seen from Fig.6a that the non-zero range and the amplitude of the heat load increases as the incoming flow speed increases. The reason is multiple: Firstly, the larger the incoming flow speed, the larger the collected mass of SLD is during a unit period of time. Therefore, more heat load is needed to vaporize the water droplet. Secondly, the larger the incoming flow speeds, the wider the range of the water collection coefficient, which means that the collision range of the water droplets is wider. To protect the wing surface from icing, the non-zero range and value of the surface heat load must be larger. Lastly, the larger the incoming flow speed, the faster the flow in the boundary layer. As a result, the convective heat-exchange coefficient increases, which raises the energy for evaporation and convection. The above three mechanisms generate the final shape of the heat load curve. From Fig.6b, the shape change of the heat load curve is non-obvious as the flight height varies. And from Fig.6c, the heat load at the same location increases as the temperature reduces. However, the range of the non-zero part of the curve remains unchanged. That’s because, the lower the environmental temperature, the larger the cold extent of SLD. As a result, more energy is needed for evaporation and heat exchange.
5. Conclusion

Ice will be growing at the leading edge, at the lip of the engine inlet of an aircraft when the aircraft passes through the cloud that contains super cold liquid droplets (SLD). As a result, aerodynamics, manipulation, and the stability of the aircraft can be greatly affected. Firstly, the heat load calculation method for the anti-icing system of an aircraft wing is established. Secondly, the present method is validated using typical experiment data performed in the NASA Glenn research center. Thirdly, the heat load for an aircraft wing is calculated, and the heat load distribution map and curve are given. Lastly, the effects of the incoming flow speed, the flight height, and the temperature on the map, and the amplitude of the heat load are studied. The conclusions are drawn as follows: The heat load is mainly located at the leading edge of the wing. The maximum value of the heat load can reach 3500w/m² and the closer to the tip of the wing, the larger the heat load value. The heat load curve has a trough in the middle and two shoulders at each side. The non-zero range of the heat load curve and the amplitude are in proportion to the flow speed. However, the variation of the flight altitude has a trivial influence on the heat load results. As the temperature rises, the amplitude of the heat load increases. However, the distribution of the curve remains the same.

References
[1] William O, Robert S, and James N. Ice shapes and the resulting drag increase for a NACA 0012 airfoil [R]. NASA TM-14427, 1984.
[2] Cole J, Sand W. Statistical study of aircraft icing accidents[R]. AIAA-91-0558, 1991.
[3] Kevin R P, Carol D J. A statistical review of aviation airframe icing accidents in the U.S[R]. NTSB, Washington, DC.
[4] Aviation Accident Statistics[R]. National Transportation Safety Board, 2008.
[5] Macklin W C. The density and structure of ice formed by accretion[J]. Quarterly Journal of the Royal Meteorological Society, 1962, 88(375): 30-50.
[6] Robert N, Kreeger R E. Analysis of a Hovering Rotor in Icing Conditions [R]. NASA/TM-2012-217126, 2012.

[7] Peterson A, Oldenburg J. Spray nozzle investigation for the Improved Helicopter Icing Spray System (IHISs) [R]. AIAA-1990-0666, 1990.

[8] Pastorbarsi C, Arrington A. Aero-Thermal Calibration of the NASA Glenn Icing Research Tunnel (2012 Test) [R]. AIAA-2012-2934, 2012.

[9] Chen J and Zhang D L. Numerical simulation on the icing process of an aircraft wing [J]. Journal of Aerospace Power, 2005, 20(6):1010-1017.

[10] Wu J, Tang L and Luke E A, et al. A Comprehensive Numerical Study of Jet Flow Impingement over Flat Plates at Varied Angles [R]. AIAA 2001-0745, 2001.

[11] Liu, H H T, Hua J. Three-Dimensional Integrated Thermodynamic Simulation for Wing Anti-Icing System [J]. Journal of Aircraft, Vol. 41, No. 6, 2004, pp. 1291-1297.

[12] Planquart P, Vanden Borre G, Buchlin J M. Experimental and Numerical Optimization of a Wing Leading Edge Hot Air Anti-icing System [R]. 43rd AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, USA, 2005.

[13] Chang S N, Hou Y Q, and Yuan X. Influence of periodic electro heating pulse on deicing surface temperature [J]. Journal of Aerospace Power, 2007, 22(8): 1247-1251. (in Chinese)

[14] Colin S. Bidwell, Mark G. Potapczuk. Users Manual for the NASA Lewis Three-Dimensional Ice Accretion Code (LEWICE3D) [R]. NASA Technical Memorandum 105974, 1993.

[15] Al-Khalil, K. M. et al. Validation of NASA thermal ice protection computer codes, Part 3- Validation of ANTICE [R]. AIAA 97-0051, 1997.

[16] G. A. L. da Silva, O. M. Silvares, E. J. G. J. Zerbini. Airfoil anti-ice system modeling and simulation [R]. AIAA 2003-734, 41st Aerospace Sciences Meeting and Exhibit, 2003.

[17] Sun Z G Research on numerical simulation of ice accretion and design for icing research tunnel parts [D]. Nanjing: Nanjing University of Aeronautics and Astronautics, 2012 (in Chinese)

[18] Liu S Y, Zhu C X and Zhu C L. et al. Effect of dynamic behavior on its impingement characteristics for droplets [J]. Journal of Aerospace Power, 2018, 35(2): 581-589. (in Chinese)