Numerical Investigation on Anti-Icing Performance of Heating Surface for NACA0012 Airfoil

K. Fukudome*, S. Uranai*, H. Mamori† and M. Yamamoto*

* Department of Mechanical Engineering, Faculty of Engineering
Tokyo University of Science
6-3-1 Niijuku, Katsushika-ku, Tokyo 125-8585, Japan
Email: kfukudome@rs.tus.ac.jp

†Graduate School of Informatics and Engineering, School of Informatics and Engineering,
The University of Electro-Communications
1-5-1, Chofugaoka, Chofu, Tokyo 182-8585, Japan

ABSTRACT

Icing is a phenomenon that super-cooled water droplets collides a solid surface and it forms an ice layer. The icing phenomena occurs on trees, power lines, buildings, cars, boats, aircraft etc. Especially, in aircraft wings, the icing becomes roughness and it leads a drastic decrease of aerodynamic performance. Therefore, prevention and prediction of icing are important from the viewpoints of engineering. There are some anti-icing techniques, e.g., bleed air, anti-freezing liquid, and an electric heater. The present study focuses on the electric heater, since it is often employed for wings due to its simplicity. As previous studies, for example, Al-Khalil et al. [1] made an experimental study of icing on the wing with the different heating temperatures of the electric heater and Bu et al. [2] carried out the icing simulation including the heat flux on the wing. In contrast, we focus on the heating area of the electric heater, because the heating area is also important in order to minimize the energy consumption of heater.

The numerical procedure consists of four steps: generation of the computational grids; computation of the flow field around an airfoil; computation of the droplet trajectories to obtain the distribution of the impinged droplets; computation of the thermodynamics to obtain ice shapes. The compressible turbulent flow was assumed. The governing equations are two-dimensional continuity, Navier-Stokes, and energy equations. The Kato-Launder k-ε model was used as a turbulence model. The droplet trajectories were tracked in the Lagrangian approach. We made two assumptions: the droplets follow the flow field, while they do not affect the flow field; the force acting on droplets is only aerodynamic drag. For the thermodynamic calculation, an "extended Messinger model" [3] was used to calculate the amount of ice. Although the model imposes the adiabatic condition on the surface of the airfoil, in the present study we modified this model to satisfy the Dirichlet condition (i.e., constant surface temperature).

As a result, we obtained ice formation in the leading edge side, and confirmed significant increase of the drag accompanied by flow separation near the leading edge. The icing volume decreased by using heater and the separation was attenuated even though the heating surface is only set from reading edge to 2% cord length location. Moreover, we confirmed the energy consumption by the heater is small enough in comparison to propulsive power of the wing estimated by the drag.

REFERENCES

[1] K. M. Al-Khalil et al., "Validation of NASA thermal ice protection computer codes part 3: the validation of ANTICE", *NASA TM-2001-210907,* pp. 1-12, (2001).

[2] X. Bu et al., "Numerical simulation of an airfoil electrothermal anti-icing system", *Aerospace Eng*. Vol. 227, No. 10, pp. 1608-1622, (2013).

[3] S. Özgen and M. Canibek, "Ice accretion simulation on multi-element airfoils using extended Messinger model", J. Heat Mass Transfer, Vol. 45, No. 3, pp. 305-322, (2009).