Effect of Hot Air Jets from a Piccolo Tube in Aircraft Wing Anti-Icing Unit

C. Sreedharan, *Q. H. Nagpurwala, S. Subbaramu

Faculty of Engineering and Technology, M. S. Ramaiah School of Advanced Studies, Bangalore 560058

*Contact Author e-mail: nagpurwala.aae.et@msruas.ac.in

Abstract

Aircraft industries are constantly facing big challenges to improve customer safety as well as aircraft performance. One of the challenges is the accumulation of ice on the aircraft wings which may cause a reduction in aerodynamic lift up to about 30% and an increase in drag up to about 40%. A widely used method in aviation industry to overcome this problem is to employ a hot-air anti-icing system owing to its simplicity, efficiency and reliability. High temperature, high pressure air from the engine compressor is extracted / bled and passed through a piccolo tube mounted inside the leading edge of the wing. The angle of impingement of hot air ensuing from piccolo tube, and the distance between piccolo tube and wing inner surface have strong influence on keeping the external surface of the wing leading edge sufficiently hot to avoid ice formation. In the present work, an anti-icing scheme for a typical aircraft wing of NACA 2412 airfoil shape, involving the effect of hot air jets from a piccolo tube, is investigated numerically. The CAD model of the wing-piccolo tube assembly was generated using CATIA software and discretisation of the flow domain was done using ANSYS ICEM CFD software. Steady state CFD analysis of the internal and external flow field of the wing was carried out using ANSYS FLUENT software for different angles of hot air jet impingement and for different distances between piccolo tube and wing inner surface. Both 2D and 3D simulations were performed to obtain the temperature and velocity distribution around the piccolo tube and on the wing external surface for a constant aircraft speed of 0.3 Mach, and bleed air temperature and pressure of 453 K and 90000 Pa respectively. It is concluded that a piccolo tube-wing surface spacing of 9 mm gives desired temperature distribution on the outer and inner surfaces of the wing. The outer surface temperature is maintained close to 300K (27º C), ensuring that there is no ice accretion on the wing.

Keywords: Ice Accretion, Wing Performance, Anti Icing System, Jet Impingement

1. INTRODUCTION

Ice formation on an aircraft structure occurs over the leading edges of aircraft wings, lip of the intake duct (diffuser), and also on the aircraft nose and fuselage [1]. Even a small ice formation on the wing leading edge would cause decrease in lift and increase in drag force. There are various systems in use to avoid ice formation, and these are mostly based on mechanical, chemical or thermal techniques. The thermal bleed technique is more popular in aircraft industry. A piccolo tube, with a series of in-line or staggered holes, is placed inside the wing leading end near to its inner surface (Fig. 1). The hot air, bled out from the engine compressor, is passed through the piccolo tube, and it ejects from the piccolo tube holes in the form of high velocity jets that impinge on to the inner surface of the wing leading edge. Subsequent heat conduction from wing inner surface to outer surface results in maintaining the outer surface of the wing leading edge hot enough to avoid accretion of ice.

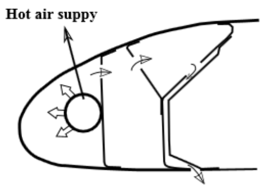


Fig. 1 A typical thermal anti-icing system using piccolo tube

There is an ever growing demand in aircraft industry for passenger and flight safety, leading to interest in hot air bleed anti-icing system. In an early study, Smith and Taylor [2] predicted the heat transfer phenomenon of water jet impinging on a solid surface, using a computational model. The predicted results showed close agreement with experimental data. Croce et al. [3] used FENSAP-ICE, an in-flight comprehensive simulation code, capable of modeling conjugate heat transfer, calculating droplet impingement and ice accretion. A wing slat element and a nacelle inlet with hot air anti-icing systems were studied in dry air conditions. The results were satisfactory, but no validation against experimental data was available. Croce et al [4] further studied the usage of FENSAP-ICE for dry and wet simulation of an engine nacelle and validated temperature behaviour for the dry simulation. Temperature behaviour for wet simulation had no validation. Habashi et al. [5] have reported the development of wet simulation for aircraft wing hot air anti-icing system. The external flow around the aircraft and the internal hot air flow were simulated separately. Al-Khalil [6] has described a scheme capable of simulating electrical and convective heating. The tempe-rature field inside the water runback flow was analysed with the energy equations, taking the temperature gradient parameter across the water film thickness also in the flow direction. De Mattos and Oliveira [7] investigated the effect of various piccolo tube hole configurations on the overall efficiency of the system. The results illustrated the effect of hot air mass flow rate from the piccolo tube jet holes on the surface heat transfer and wing skin temperature. Liu and Hua [8] analysed an aircraft wing anti-icing system in dry conditions using ANSYS FLUENT software for conjugate heat transfer through wing skin. The surface temperature matched reasonably well with the flight test data. Later, Hua et al. [9] also modeled the unsteady development of bleed airflow for the same system. The comparison of the time-accurate temperature distribution at half-span showed satisfactory agreement with the flight test data.

The past and recent studies show that a continuous improvement in anti-icing techniques is mandatory for high performance wing designs. Although, there is published information on various aspects of thermal bleed icing system, but there is still a need to understand the underlying flow physics and thermal aspects that will help in designing more effective and efficient systems. In the present investigations, a NACA 2412 wing airfoil section is chosen and an in-depth parametric study is performed through numerical simulations to obtain a comfortable temperature distribution on the outer surface of the wing in order to minimise or avoid ice formation during in-flight condition of aircraft.

2. WING SECTION AEROFOIL

In the present study, a NACA 2412 aerofoil [10] was chosen for the wing section, as shown in Fig. 2. The profile is defined with maximum camber of 2% chord length, maximum camber position at 40% chord length and maximum thickness equal to 12% chord length. A chord length of 1.5 m was selected, conforming to the wing dimensions of a typical aircraft.

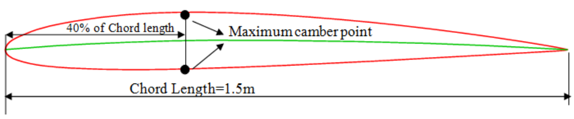
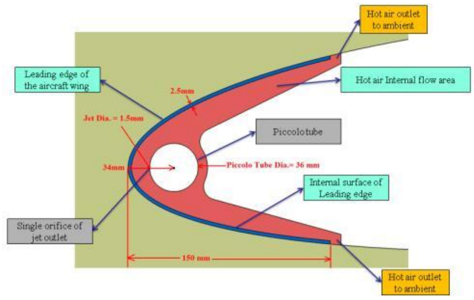


Fig. 2 NACA 2412 aerofoil section [10]

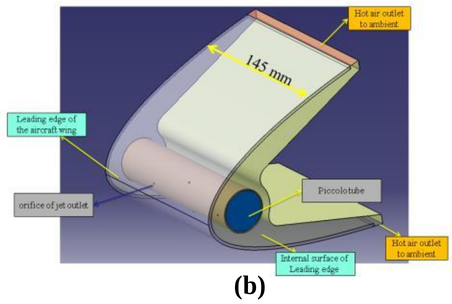
3. GEOMTRIC MODEL OF WING WITH PICCOLO TUBE

The hot-air anti-icing problem was treated with a modular approach where the physical domain was divided into three regions: (1) the external flow, composed of the gas phase (atmospheric air), (2) the hot air internal flow, including the diffuser bay and relatively narrow upper and lower exit air passages, and (3) the wing wall or skin, which conducts heat from the interior of the leading edge to the runback flow on the wing external surface.

Based on the above modular approach, the 2D and 3D wing models were created using CATIAV5 R14 software. The 2D model of the wing cross-section is shown in Fig. 3a, and the 3D wing model in Fig. 3b. The internal domain with piccolo tube was modeled for 10% of the chord length of the aircraft wing section. The piccolo tube had a diameter of 36 mm and hot air injection jet diameter of 1.5 mm. The model also comprised two vent holes / passages for the hot air to escape -- one on the upper side and the other on the lower side of the wing. Three rows of staggered hot air jets (Fig. 4) were located on the piccolo tube at 0º (centre), +45º and at -45º. The numerical simulation was done with each jet injecting hot air individually and with a combination of all three jets injecting together. Apart from these jet configurations, the distance *z* between the outer surface of the piccolo tube and the inner surface of the wing leading edge was set at 4.5 mm, 9mm and 13.5 mm. Further, the 3D analysis was performed with the three rows of staggered jet configuration along a portion of the length of the piccolo tube for three different values of *z*, as indicated above.



(a)



(b)

Fig. 3 2D and 3D models of wing and piccolo tube assembly

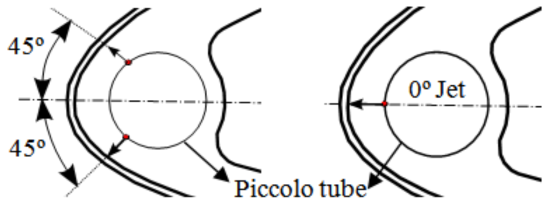


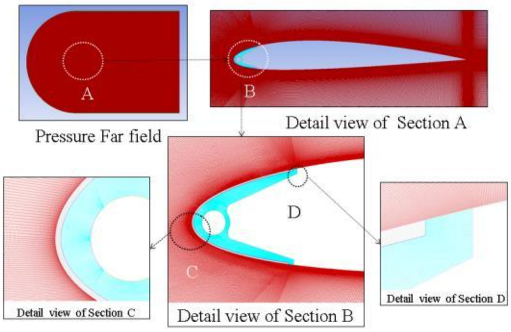
Fig. 4 Angular positions of hot air impinging jets

4. FLOW DOMAIN DISCRETISATION AND SOLVER SETTINGS

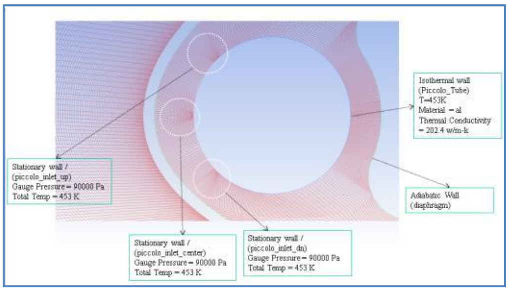
The flow domain was meshed with quad elements for 2D simulation (Fig. 5) and with hexahedral elements for 3D simulation (Fig. 6) using ICEM CFD software. The overall computational domain for 2D case, along with a few blown up sectional views, is shown in Fig. 5a. The mesh for internal domain and the boundary conditions are shown in Fig. 5b. The discretised 3D flow domain is shown in Fig. 6. An optimum grid size of 3.5 lakh elements was chosen for 2D case based on grid independence study.

The numerical analysis was carried out using ANSYS FLUENT software, using standard *k*-*ε* turbulence model. The residuals were reduced to a minimum level of 10^{-5} indicating solution convergence. The wing external condition was set for a typical flight Mach number of 0.3 at an altitude of about 5 km. The corresponding ambient temperature was taken as 258 K. The ambient conditions of temperature and pressure also prevailed at the inner domain of the wing containing piccolo tube. The hot air pressure and temperature at piccolo tube inlet were 90000 Pa and 453 K respectively. The following cases of hot air jet(s) coming out of piccolo tube were considered both in the 2D and the 3D models:

(1) Centre jet at 0º, (2) jet at +45º, (3) jet at -45º, and (4) all three jets together. In addition, the distance, *z*, between piccolo tube and wing leading edge was also varied (Fig. 7) and was kept at 4.5 mm, 9 mm and 13.5 mm.



(a)



(b)

Fig. 5 Discretised 2D flow domain and boundary conditions for NACA 2412 aerofoil

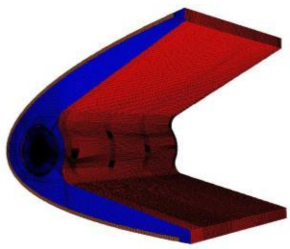


Fig. 6 Discretised 3D flow domain of aircraft wing leading end with piccolo tube

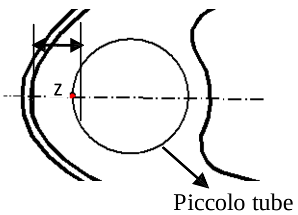


Fig. 7 Distance between piccolo tube and wing inner surface

5. RESULTS AND DISCUSSIONS

5.1 Influence of Distance *z* and Jet Impingement Angle --- 2D Case

The three parameters, viz. distance *z* between piccolo tube and wing inner surface, jet impingement angle, and jet mass flow rate are the ones that influence the heat transfer from hot air to the wing inner surface and then heat conduction to the wing outer surface for a given wing skin thickness and material. In the present investigations only first two parameters are considered.

Fig. 8 shows total temperature distribution for all the impingement distances with +45º and -45º jets. The hot air from piccolo tube holes is injected at a temperature of 453 K. It is observed that for all three values of *z*, the temperature distribution is almost symmetrical about the chord line on both the sides. Higher temperature is found in the internal domain for z=4.5 mm as compared to the other two cases. However, for z=9.0 mm, a relatively favourable temperature distribution is observed compared to the other cases. Also, a higher, but concentrated, temperature zone is obtained near the wing surface for z=4.5 mm and 9.0 mm, while lower temperatures are observed at outlet for z=13.5mm, showing the effect of distance between piccolo tube and wing inner surface. In case of z=9 mm, there is smoother temperature distribution along the inner surface of the wing after the jets diverge at the upper and lower regions of the wing surface. Overall, for the two-jet case, the distance z=9.0 mm seems to give better temperature distribution. Similar favourable temperature distribution was also obtained at z=9.0 mm for a single centre jet impinging on the wing inner surface. Based on these parametric studies with various jet impingement angles and impingement distances, it is observed that an ideal location for piccolo tube is 9.0 mm away from the wing wall. A total temperature of 471.73 K is obtained at the inner surface of the wing near the point of impingement and then the temperature is gradually reduced to 303.19 K at the outlet of the inner domain on both upper and lower sides.

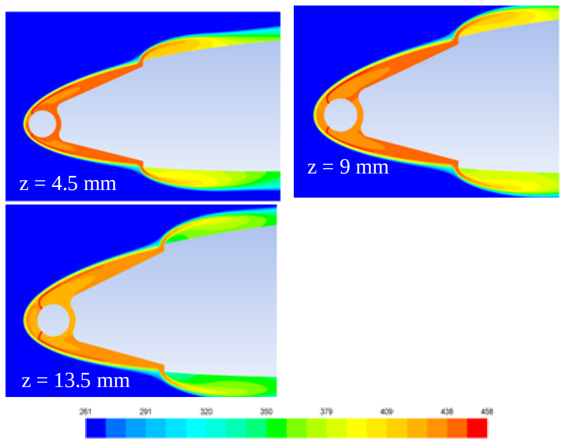
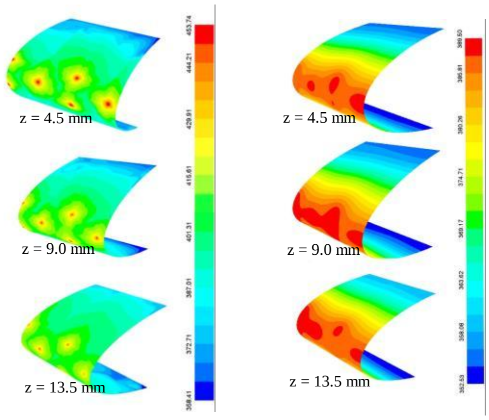


Fig. 8 Total temperature distribution with two jets at +45º and -45º and *z* = 4.5, 9 and 13.5 mm

5.2 Influence of Distance *z* and Jet Impingement Angle --- 3D Case

With an optimum piccolo tube location obtained from 2D simulations, the 3D analysis was performed by taking a portion of the wing containing four spanwise locations of air injection holes in staggered configuration, as shown in Fig. 6. The total temperature distribution on the inner and outer surfaces of the wing is shown in Fig. 9a and 9b respectively for three impingement distances with staggered jets on the piccolo tube. It may be preferable to plot static temperature distribution on the wing surfaces, but since the flow Mach numbers are small, the total temperature variation will also provide similar useful information. It is observed that a higher temperature of about 453 K occurs at the centre of each jet impingement location. The temperature gradually reduces around the respective impingement location and finally reduces to a low value of about 358 K near the outlet of inner flow domain. The area covered by high total temperature profile around the centre of impingement on the wing inner surface is more for z=4.5 mm and less for z=13.5 mm.

Fig. 9b shows the total temperature distribution on the outer surface of the wing for the three impingement distances. It is observed that a high total temperature of about 389 K occurs along the leading edge of the wing and then it gradually reduced to a lower value of about 356 K away from the leading edge both on suction and pressure surfaces. Hence, a sufficiently high temperature obtained in the vicinity of the leading edge will prevent ice formation on the outer surface of the selected wing under the conditions studied.



(a) Wing inner surface (b) Wing outer surface

Fig. 9 Temperature distribution on inner and outer surfaces of the wing for z=4.5, 9.0 and 13.5 mm --- 3D simulations

6. CONCLUSIONS

A thermal bleed anti-icing system, involving piccolo tube in a representative aircraft wing made of NACA 2412 airfoil section, has been studied through numerical simulations in both 2D and 3D environment. The simulations were done for a constant aircraft speed of 0.3 Mach and bleed air temperature and pressure of 453 K and 90000 Pa respectively. The parameters varied included number and spacing of jets and the distance, z, between piccolo tube and inner surface of the wing leading end. From the results of CFD analysis, the following conclusions are drawn:

- In 2D case for z=4.5 mm, the jet impingement distance is small. The nearness of the jet with the wing inner surface creates lot of turbulence inside the domain with three jets. Also, the temperature distribution is not as effective as in other cases. In 3D case with z=4.5 mm, the flow behaviour is better as compared to that in 2D case.
- In 2D case for z=13.5 mm, the impingement distance is relatively larger. Hence, the temperature distribution along the inner surface of the wing is not effective. This is true even in 3D case with z=13.5 mm. Impingement is not effective and the ensuing jets sag before they actually strike the wing inner surface.
- In 2D case for z=9.0 mm, the temperature distribution along either side of the chord line is more uniform and the results appear to be optimised when compared to the other two cases of z=4.5 mm and z=13.5 mm. In case of the 3D model with z=9.0 mm, the temperature distribution on outer surface of the leading edge is relatively better than the other two cases with z=4.5mm and z=13.5mm. This is also supported by local Nusselt number distribution for the case with z=9 mm (not presented here).
- From 2D and 3D analysis results it is evident that a distance of z=9.0 mm between piccolo tube and wing inner surface gives favourable temperature distribution on the outer and inner surfaces of the wing, and the temperature is sufficiently high to avoid ice formation under selected in-flight conditions.

REFERENCES

[1] Lynch F.T., Khodadoust A., (2001) Effects of ice Accretions on Aircraft Aerodynamics, *Progress in Aerospace Sciences*, 37, pp. 669-767.

[2] Smith A.G., Taylor K., (1997) Simulation of an Aircraft Engine Intake Anti-Icing System, *PHOENICS Journal of CFD and its Applications*, 10(2), pp. 150-166.

[3] Croce G., Habashi W.G., Guevremont G., Tezok F., (1998) 3D Thermal Analysis of an Anti-Icing Device Using FENSAP-ICE, AIAA paper 1998-0193.

[4] Croce G., Beaugendre H., Habashi W.G., (2002) CHT3D: FENSAP-ICE Conjugate Heat Transfer Computations with Droplet Impingement and Runback Effects, AIAA paper 2002-0386.

[5] Wang H., Tran P., Habashi W.G., Chen Y., Zhang M., Feng L., (2007) Anti-Icing Simulation in Wet Air of a Piccolo System Using FENSAP-ICE, SAE Paper 2007-01-3357

[6] Al-Khalil, K.M., (1991) *Numerical Simulation of an Aircraft Anti-Icing System Incorporating a Rivulet Model for the Runback Water*, Ph.D. Thesis, University of Toledo, Toledo, OH.

[7] De Mattos B., Oliveira G., (2000) Three-dimensional Coupled Analysis of a Wing Slice Slat with a Piccolo Tube, AIAA paper 2000-39841.

[8] Liu H., Hua J. (2004) Three-Dimensional Integrated Thermodynamic Simulation for Wing Anti-icing System, *Journal of Aircraft*, 44(6), pp. 1291-1297.

[9] Hua J., Kong F., Liu H., (2007) Unsteady Thermodynamic Computational Fluid Dynamics Simulations of Aircraft Wing Anti-Icing Operation, *Journal of Aircraft*, 44(4), pp. 1113-1117.

[10] Aerospace Engineering (2014) Retrieved Feb 23, 2014, from http://aerospace.illinois.edu/m-selig/ads/aircraft.html.