Indian Journal of Science and Technology, Vol 8(30), DOI: 10.17485/ijst/2015/v8i30/86058, November 2015
ISSN (Print) : 0974-6846
ISSN (Online) : 0974-5645

Applying Computational Fluid Dynamic to Predict the Thermal Performance of the Nacelle Anti-Icing System in Real Flight Scenarios

M. A. Ismail* and M. Z. Abdullah

Universiti Sains Malaysia, Advanced Packaging and SMT Group, School of Mechanical Engineering, Malaysia; azmi_meche@usm.my

Abstract

Ice accumulation on the inner surface of a nacelle lip-skin can be hazardous. Therefore, it is necessary that the aircraft be protected against the effect of ice accumulation. The Piccolo Tube Anti-Icing (PTAI) is one of the most reliable ice protection systems for nacelle lip-skins and aircraft wings, and is commonly used in commercial aviation. Thus, many researchers studied and developed the empirical correlation of the PTAI system, including Jeffry Brown of Queens University Belfast. He focused on the nacelle lip-skin PTAI system. However, the Brown correlation was developed at room temperature. In present study, on the other hand, the Brown correlation was modified with the help of CFD, in order to make it suitable at different ambient temperatures. CFD analysis can help to save time and effort, as well lower the cost of the nacelle design, by reducing the number of experiments required during the development. The present research executed CFD in 5 different conditions, i.e. Laboratory, Taxi, Climb, Hold, and Descent. Laboratory data closely agree with the Brown correlation, having been run in similar conditions. The average Nusselt number increases as the Reynolds number increases. In conclusion, the research demonstrated that CFD is not only capable of predicting the thermal performance of the PTAI, but it can also be used to investigate some parameters/characteristics that are difficult to obtain from experiments, e.g. the ambient temperature effects and the hot air velocity inside the D-chamber.

Keywords: Anti-Icing, CFD, Lip-Skin and Aircraft Crash, Nacelle, Piccolo Tube Anti-Icing

Nomenclature

$A_{impingement}$ = Effective impingement area
A_{nozzle} = Nozzle cross sectional area
C = Equation gradient
c_p = Heat capacity at constant pressure
Cx = Nozzle spacing
d = The nozzle diameter
G = Hot air mass flow per unit area of impingement surface
h_{ave} = Average heat transfer coefficient
k = Air thermal conductivity
$m_{hot\ air}$ = Hot air mass flow rate
N_{nozzle} = Number of nozzles
Nu_{ave} = Average Nusselt number
Pr = Prandtl number
$q_{lip\text{-}skin}$ = Heat transfer rate at the impingement area
Re_G = Reynolds number based on the surface area of impingement.
Re_{PTAI} = Reynolds number based on the nozzle diameter
$T_{ambient}$ = Ambient temperature
$T_{ave\ exhaust}$ = Average exhaust temperature
$T_{ave\ impingement}$ = Average temperature at the effective impinging area
T_y = Local temperature
$T_{piccolo}$ = Air temperature within the piccolo tube
y^+ = First cell spacing normal to the nacelle surface
μ = Air dynamic viscosity

* Author for correspondence

1. Introduction

A large number of aircraft accidents had happen during 6 distinct phases: taxi, takeoff and initial climb, climb, hold, descent and initial approach, and final approach and landing. Many of accidents occurred during the final approach and landing phase. One of main contributors to aircraft crashes was ice accretion on the aircraft critical surfaces: the leading edge of the tail, the wing, and the nacelle[1]. According to the report released by the National Transport Safety Board, the probable cause of the February 2005 Cessana Citation 560 crash in the U.S. was the formation of rime ice on the wing leading edge during descent[2]. As the consequence, the left wing suddenly stalled, causing the aircraft to roll towards the left, resulting in the crash.

Ice accretion on leading edge surfaces occurs upon flying through clouds containing Super Cooled Water Droplets (SWD), where the surface temperature is 0°C and below[3]. Ice, if allowed to accumulate on the nacelle lip-skin surface, can result in a reduction or the loss of aerodynamic performance due to the formation of non-aerodynamic shapes on the nacelle lip-skin[4], which results in a non-aerodynamic profile. When this occurs, the fuel consumption increases, due to the higher drag force generated when these non-aerodynamic shapes are present. Ice accumulation on the inner surface portion of the nacelle lip-skin can restrict the airflow through the fan, leading to the loss in performance (thrust), and possible malfunction. The worst scenario would be when broken ice particles are absorbed, which ingestion of would damage the engine.

Protection against icing is required, as icing on these critical surfaces causes the loss in aerodynamic and engine performances, damages the engine and the noise abatement device, and is a potential cause of aircraft crashes. The Federal Aviation Administration therefore necessitates that all aircraft manufacturers install ice protection systems on their aircraft. Currently, there are two types of ice protection systems widely employed in commercial aviation: the anti-icing system (*AI*) and the de-icing system (*DI*).

DI and *AI* systems have been developed to address the problem of ice formation on leading edge surfaces. A *DI* system allows ice to form and then removes or sheds the ice from the surface by means of some thermal effects or mechanical motions. Usually, it operates periodically. Examples of the existing *DI* systems include Pneumatic Inflatable Boots (*PIB*), Electro Impulse De-Icing (*EIDI*), Electro Magnetic Expulsion De-Icing (*EMEDS*), the Electric Thermal Heater (*ETH*), and the Thermo-Mechanical Expulsion De-Icing System (*TMEDS*).

A *PIB* device consists of rubber strips positioned on the outer surface of the structure to be protected. The ducts inside the rubber boot function as an air conduit. When they are inflated using air bled from the engine, the external surface of the rubber distorts. The distortion causes the ice to break up, due to a combination of shearing and bending. The flow of air over the external surface then acts as an aerodynamic peel force to remove the ice from the structure. Unfortunately, the commercial *PIB* system has a low efficiency when it comes to shedding ice less than 0.03" thick[5,6].

The *EIDI* and *EMEDS* are also known as Low Power De-Icing (*LPDI*) devices. These devices consume one hundred times less the amount of electric energy that an electric heater or a hot bleed air system does. Therefore, these *DI* systems are mostly suited for providing protection for small aircraft[7,8]. An *EIDI* system functions while a capacitor is charged and discharged through an electric coil. When the *EIDI* system is switched on, an electric coil is energized. This produces a magnetic field and a large amplitude impulse, which acts upon a nearby electrical conductive plate. The impulse force affects on an aluminium nacelle surface, resulting in the expansion and the contraction of the metallic surface. This rapid, repeated motion generates mechanical vibration on the leading edge. As a result, this mechanical vibration breaks the bond between the ice and the surface, so that broken ice is shed[9]. The *EIDI* system is limited in its capability to remove ice with thickness less than 0.2"[7].

An Electro Magnetic Expulsion De-icing Systems (*EMEDS*) is usually used in conjunction with an Electro Thermal Heater [*ETH*]. This combination is known as a hybrid system. Conductive strips are wound onto coils with actuators shaped into flattened elongated tubes. Both the axis of the winding coils and the longitudinal axis of the elongated tube are coincident with each other. These tubes are situated in close proximity to the surface to be protected. When a current is passed through the elongated coils they want to become circular, striking the internal surface of the protected structure. The mechanical motion of these bending actuators sheds the ice from the surface and the external air flow washes them away. As a rule, *EMEDS* is used in small horizontal aviation components, and has the capacity remove ice with thicknesses in the range of 0.05” to 0.2”[10].

ETH systems are mainly used in turbo propeller engine aircraft. The main construction of an *ETH* system consists of resistive heaters installed on the external skin of the nacelle intake. These resistive heater devices consist of strips of conductors sandwiched between layers of neoprene, or a glass fibre cloth impregnated with epoxy. To protect the pad against rain erosion, the electrical heater elements are coated with polyurethane-based paint[11]. The disadvantage of *ETH* is polyurethane's low thermal conductivity (0.02 W/mK), along with the degradation in the thermal efficiency of conventional *ETH* systems[12]. *TMEDS* uses a similar concept to that of the hybrid system, and can remove ice accretions less than 0.05” thick very well, with very low power consumption[13]. However, this type of anti-icing system is ineffective at removing ice accretions more than 0.05” thick.

Some examples of existing *AI* systems are Electric Anti-Icing (*EAI*), a combination of *EAI* and *LPDI* devices named Hybrid Anti-Icing, and Piccolo Tube Anti-Icing (*PTAI*)[9]. *EAI* systems are similar to *ETH* systems. Unlike, ETH system, they utilise electro thermal heater elements that are switched on constantly, rather than cyclically[10]. Hybrid Anti-Icing systems are a combination of *EAI* and *LPDI*, and are used as an alternative to hot bleed air systems. They are suitable for small aircraft with limited power supply[14].

PTAI is one of the most popular hot bleed air anti-icing devices utilized in wing[15] and nacelle anti-icing systems[16]. *PTAI* is an extremely efficient and reliable anti-icing system[17]. The parameters affecting the thermal performance of the *PTAI* include the hot air mass flow rate, jet spacing, the distance from the holes to the surface, the impingement angle, etc[18]. This high performance preventive system is better than other ice protection systems with respect to efficiency, but has limitations in terms of performance losses, energy cost, complexity, weight, and hotspot issues.

Computational Fluid Dynamic (*CFD)* is a proven, useful, and reliable tool, used to predict an assortment of issues, including heat transfer, fluid flow, and combustion. Currently, a great number of researchers are using CFD codes as a first step in designing or studying anti-icing systems. By using *CFD*, researchers can reduce the number of tests, as well as the cost associated with physical experiments and, at the same time, study some parameters that cannot be obtained from the experiment. Saeed and Al-Garni used *CFD* codes to study the effect of an array of jets on the aircraft wing/slat surface[19]. Hua et al. applied *CFD* to study the thermal flow and the temperature of a wing skin under different hot air conditions, created using control valve adjustments[20]. Domingos et al. developed a 2D simulation tools to predict the thermal performance on a wing leading edge, and compared their results with experimental data for dry and wet conditions[21]. Recently, Pellissier et al.[22] performed a design optimization of the *PTAI* by using a *CFD* code.

First, this study uses CFD to predict thermal characteristics at various altitudes and ambient temperatures. Then, this paper investigates the effects of the Reynolds number, based on the surface area of impingement, $Re_{G,}$ on the average Nusselts number, Nu_{ave}, for five different conditions, in order to better understand the system's mechanisms. Later on, a modified Brown empirical correlation, which is suitable for use with different ambient temperatures, is developed and presented at the end of this paper.

2. Model Description

In the present study, *CFD* is applied in order to investigate the thermal performance of a *PTAI* system at various free stream Mach numbers and ambient temperatures. The holes on the tube were developed such that they formed two staggered rows, named Nozzle 1 and Nozzle 2 for row 1 and row 2, respectively, as shown in Figure 1. Briefly, an engine nacelle *PTAI* system works as follows: hot air is bled from the engine compressor at high pressure and high temperature, and conducted forward to the fixed leading edge D-chamber using a supply pipe. Within the D-chamber resides a circumferential perforated tube (or piccolo tube), which is held at a certain distance from the surface to be protected, in this case, the leading edge lip-skin. The high-pressure and high temperature air exits the piccolo tube via a series of holes or nozzles strategically drilled to generate a series of high-velocity jets of hot air, which impinge on the nacelle lip-skin's inner surface. The placement of the piccolo holes can be tailored to provide a very precise area of jet impingement, thus allowing the designer to efficiently control the region to be heated. Heat from the hot air jets keeps the external nacelle lip-skin surface temperature at sufficiently high to fully evaporate any impinging moisture, thus ensuring that the surface is kept free of ice. The D-chamber contains an exhaust on the bulkhead, to ensure that spent air dumped overboard.

3. Geometry and Mesh Generation

First, the 3D geometry and the mesh of the *PTAI* system were developed by using a Gambit pre-processor. Because of the periodical nature of the arrangement of the holes, only 2 holes are included in the *CFD* model of *PTAI* system. In the present study, Nozzle 1 and Nozzle 2, as shown in Figures 1 and 2, representing row 1 and row 2 holes, respectively. This is done in order to reduce mesh consumption, hence saving time and CFD model memory. Moreover, the circumferential piccolo tube and the double curvature nacelle lip-skin were replaced with a straight tube and a simplified straight section of nacelle lip-skin, with a single curvature. The predicted results of this simplified geometry were expected to deviate little from that of the actual design. Some past research concurs with this assumption, e.g. the work done by Brown[16,18], and Raghunathan[17]. A schematic drawing of the perforated piccolo tube used for the simulation is shown in Figure 2, where d is the diameter of the nozzles.

The structured mesh, shown in Figure 3, was developed and employed to predict the thermal performance of the *PTAI* system. More than 2.9×10^6 elements were used in this model. A very fine mesh was set up in areas containing Nozzle 1 and Nozzle 2, in order to obtain accurate simulation results. Additionally, the grid was refined/adapted by the Fluent CFD code and compared to the original model to ensure that the original results were reliable. The first cell spacing normal to the nacelle surface was set at $y^+ < 1$, as shown in Figure 4.

As for the boundary conditions, a constant pressure at the far-field was employed on the front curve surface (the blue surface) and a pressure outlet was utilized at the rear ambient surface (the red surface). At the same time, periodic boundary conditions were used on the right and the left side surfaces of the nacelle and the ambient domain (magenta colour). All the boundary conditions are shown in Figures 5 and 6. The nacelle lip-skin and the piccolo tube were declared to be the solid domain (light blue), while the hot air inside D-chamber, and the ambient air volumes were declared to be the fluid domain. The mass flow rate boundary condition was given at the piccolo tube entrance surface (purple surfaces), and a pressure outlet surface at the bottom bulkhead area (red dots) was used as exhaust surface (red dots).

The FLUENT *CFD* code was utilized to obtain predicted results. K-ω SST was selected as the turbulence model, as it accounts for the transport of the turbulent shear stress, and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients[23]. In addition, the k-ω SST model offers better prediction results, compared to the k-ε and spalart-allmaras models, especially inside complex boundary layers with adverse pressure gradient[24]. Ideal gas was assumed in the computational model, as the air velocity within the simulation was higher than 0.3 Mach number (Mn). Second upper upwind was applied to help obtain more precise results.

The following assumptions have been made within this study. First, the turbulent intensity of the free stream is constant (2%) for all test conditions. All the cases analysed have the same angle of attack as the Brown case (0°). The study also assumes that most of the heat from the hot air was transferred directly to the effective impingement area. Therefore, the interaction between the hot air and the outside effective impingement area was assumed to be negligible, in order to perform the Average Nusselt number calculation. The same approach has been used in the past by researchers such as Brown[16,18].

Table 1. Summary of all altitude conditions

Condition	Altitude (m)	Free stream Mach number
Lab	0	0.12
Taxi	396	0.1
Climb	3200	0.42
Hold	3048	0.32
Descent	2438	0.44

Table 2. Summary of all altitude conditions

Condition	T_∞ (K)	$T_{piccolo}$ (K)	P_∞ (Pa)
Lab	294	413	101300
Taxi	290	383	96526
Climb	276	545	68051
Hold	279	458	69774
Descent	282	403	75153

The study covered five different conditions, with a different altitude, free stream Mach number, hot air temperature ($T_{piccolo}$), ambient temperature (T_∞), and ambient pressure (P_∞) for each one. The operating conditions for all the cases are shown in Tables 1 and 2.

4. Grid Independence Test

Before each run, a grid dependency test was conducted, to ensure that the simulation results obtained were reliable. Figure 7 shows the correlation for the results of the dimensionless temperature, the Ť distribution on the nacelle lip-skin, for five different grid densities: very coarse mesh ($1.5x10^6$ grid elements), coarse mesh ($2.0x10^6$ grid elements), current mesh ($2.9x10^6$ grid elements), fine mesh ($4.0x10^6$ grid elements), and very fine mesh ($1x10^7$ grid elements). The dimensionless temperature distribution in the present study was defined as:

$$\check{T} = (T_y - T_{ambient})/(T_{piccolo} - T_{ambient}) \quad (1)$$

The wrap location represents the measured surface distance from the top point to the bottom of the nacelle lip-skin, and is expressed as a percentage of the total lip-skin length, as shown in Figure 8.

The plot clearly shows that the structural temperature profiles for all the meshes are similar. As shown in the figures, the temperatures for the current mesh, the fine mesh, and the very fine mesh, are similar. However, the temperatures of very coarse and the coarse meshes are significantly lower than that of the current mesh.

The contours of the Mach numbers from the nozzles to the impingement surface for the coarse mesh, the current mesh, and the fine mesh, are shown in Figures 9, 10, and 11, respectively. It can be seen from these contours that the Mach numbers for the fine mesh and the current mesh are very similar, while the Mach number of the coarse mesh is lower than that of the current mesh. Since the very coarse mesh and the coarse mesh were ruled out as choices, the current mesh was chosen for all computational models, in order to save computational time.

5. Result and Discussion

Figure 12 illustrates the temperature contour on the nacelle lip-skin for the climb condition at the Reynolds number based on the surface area of Impingement (Re_G) of 56.2. In the present study, the Reynolds number based on the nozzle diameter is not suitable for use in the empirical correlation with the average Nusselt number, Nu_{ave}, since the Nu_{ave} is based on the heated area in the nacelle D-chamber. Therefore, a new Reynolds number based on the surface area of impingement (Re_G), was defined as in Equation 2 which is defined as:

$$Re_G = (4\, G\, d)/(\mu\, \pi) \quad (2)$$

The figure shows that the highest dimensionless temperatures occur at the lower portion of the nacelle lip-skin, which must remain ice-free in order to avoid ice shedding and subsequent absorption by the engine. Then, the structural temperature of the lower half of the lip-skin should be higher than that of the upper half when the system is in operation.

The figure also reveals that the Points A and B have higher temperatures that their surrounding regions. These points are the location of the highest intensity of the impinging jets for Nozzle arrays 1 and 2, respectively. The figure also illustrates that the temperature at Point A is a little higher that that at point B. This is because the distance between the nacelle lip-skin surface and the nozzles themselves is slightly shorter at Point A than it is at Point B.

The dimensionless temperature (Equation 1) profiles along the nacelle lip-skin in the plane hotspot A for all conditions except Laboratory, are shown in Figure 13. All profiles show similar trends, with the dimensionless temperature increasing robustly from wrap location of 0% to hotspot A, then decreasing moderately from hotspot A to wrap location 100%. Of the four cases analysed, it is the Taxi case that provides the highest lip-skin structural dimensionless temperature, followed by Descent, Climb, and Hold, except between points 1 and 2. In fact, the Taxi case has the lowest temperature difference, $T_{piccolo}-T_{ambient}$, and an intermediate temperature difference, $T_y-T_{ambient}$. According to Equation 1, the dimensionless is proportional to $T_y-T_{ambient}$ and inversely related to $T_{piccolo}-T_{ambient}$. From Figure 13, it is evident that the Hold produces the lowest lip-skin dimensionless temperature since it has a high temperature difference, $T_{piccolo}-T_{ambinet}$ and a low temperature difference, $T_y-T_{ambient}$, except between points 1 and 2, where the hold temperature is similar to the climb temperature. This phenomenon occurred because the hold had a higher $T_{ambient}$ than that of the climb, as shown in Tables 1 and 2.

The relationship between the average Nusselt number (Nu_{ave}) and average Reynolds number, Re_G, is investigated as well as Brown empirical correlation. This correlation helps the designer to predict the *PTAI* performance for all nacelle designs. Hence, the cost of ice protection for nacelle can be reduced by reducing the number of experiments.

In Figure 15, $Re_G^{0.922} (Cx/d)^{0.064} Pr^{0.3}$ in the range from 0 to 100 have been plotted against average Nu_{ave} for the five scenarios. A line has been added to the same plot, to show the results obtained from Brown's study[18], to compare directly. Brown's empirical correlation is used for comparison because his model is similar to the nacelle D-chamber model in this study. Brown used the following equation to estimate Nu_{ave}:

$$Nu_{ave} = C\, Re_G^{a}\, (Cx/d)^{b}\, Pr^{0.3} \quad (3)$$

From his study, he obtained $C = 0.577$, $a = 0.922$ and $b = 0.064$

The corresponding Nu_{ave} is expressed as:

$$Nu_{ave} = (h_{ave}\, d)/k \quad (4)$$

where h_{ave} was determined using the following equation:

$$h_{ave} = q_{lip\text{-}skin} / (A_{impingement}\, (T_{piccolo} - T_{ave\ impingement})) \quad (5)$$

The heat transfer rate at the impingement area, $q_{lip\text{-}skin}$, can be determined using the energy balance equation below:

$$q_{lip\text{-}skin} = m_{hot\ air}\, c_p\, (T_{piccolo} - T_{ave\ exhaust}) \quad (6)$$

Since the geometry of the nacelle D-chamber model and the position of the piccolo tube in this study were similar to those used by Brown[16], the same method of estimating the effective impinging area was used. Brown estimated that the effective impinging area started from the Highlight point to 152.4 mm inside the highlight. The locations of hotspots 1, 2, and 3, were approximately 25.4 mm, 76.2 mm, and 127 mm from the highlight, respectively, as shown in Figure 14. The average effective impinging length was estimated at 25.4 mm from impinging point and on both sides of it. However, only two rows piccolo holes have been used in the present study. The distance of the impinging effect was estimated at 25.4 mm from the impinging point. Therefore, the mean effective impinging area has been determined from the highlight to be 101.6 mm inside the highlight,

The structural temperature increases as Re_G increases, because Re_G is proportional to the mass flow rate of the hot air, $m_{hot\ air}$. Figure 15 demonstrates a proportional relationship between Nu_{ave} and Re_G. It also shows the laboratory condition has the highest Nu_{ave} gradient, followed by the taxi, the descent, the hold, and then the climb condition.

From Figure 15, it is clear that laboratory condition CFD results show closely follow Brown's empirical correlation. As shown in same figure, the climb condition shows the largest deviation from the results obtained by Brown, followed by the holdhold and the descent conditions. This phenomenon happens because ambient temperature is the main parameter affecting Nu_{ave} in the *PTAI* system. As the ambient temperature decreases, more heat from the nacelle lip-skin is drawn out to the outside environment. Consequently, the average temperature of the lower half of the lip-skin decreased. As Equation 5 shows, h_{ave} is inversely proportional to $T_{ave\ impingement}$.

The ambient temperature appears to be the parameter affecting the Nu_{ave} in the *PTAI* system the most. As the ambient temperature decreases, more heat from the nacelle lip-skin is drawn to the outside environment. Consequently, the average temperature of the lower half of the lip-skin decreased. In accordance with Equation 5, h_{ave} is inversely proportional to $T_{piccolo} - T_{ave\ impingement}$. Therefore, if $T_{piccolo}$ is constant, $T_{piccolo} - T_{ave\ impingement}$ increases as the ambient air temperature is reduced. As the result, h_{ave} decreases. Hence, Nu_{ave} also decreases as the ambient air temperature decreases. In the present study, the laboratory condition temperature was the room temperature, while the Climb test has the lowest temperature (see Tables 1 and 2). As a result, the laboratory and the climb conditions produce the highest and the lowest Nu_{ave} gradients, respectively.

Figure 16 shows the plot of the Brown correlation coefficient (C) against the ambient temperature for all the conditions. A value of *C* for each case was obtained from Figure 15, i.e. from the CFD results. The Laboratory condition produced the highest *C*, followed by Taxi, Descent, Hold, and then Climb. The relationship between $T_{ambient}$ and *C* was then plotted (see Figure 16). The figure shows that *C* is directly proportional to $T_{ambient}$, a linear relationship.

A modified correlation is proposed in Equation (7), based on the findings from this study. Instead of *C* being treated as a constant, it is treated as a function of the ambient temperature.

$$Nu_{ave} = (0.016T_{ambient} - 4.418)\, Re_G^{0.922}\, (Cx/d)^{0.064} Pr^{0.3} \quad (7)$$

6. Conclusion

The predicted results show that the upper half of the lip-skin sees lower temperatures than the lower half. The highest temperature spots occur at the points where the high velocity jets form the nozzles impinge directly on the lower half of the lip-skin. The nozzles are purposely directed towards this area in order to avoid ice formation on this surface, therefore protecting the engine from absorbing ice. All cases show that Re_G is proportional to Nu_{ave}. Moreover, the simulation results for Laboratory and Taxi conditions closely matched the empirical correlation from Brown's study[16]. However, the other cases have results much lower than those predicted by Brown. It is found that the lower ambient air temperature of the flight phases analysed is a key factor behind this deviation. The gradient of the Nu_{ave} decreases as the ambient temperature, $T_{ambient}$, decreases. The gradient coefficient in Brown's empirical equation, C, was modified to take $T_{ambient}$ into account, in order to make it more accurate for all of the conditions analysed. It can be concluded that FLUENT CFD code has the ability to predict the thermal performance of an anti-icing system in turbulent flow regime. In addition, the correlation of the designated Reynolds number and the Nusselt number is proportional. The effect of the ambient temperature on this correlation is linear.

7. Acknowledgement

Many thanks to Grant code 304/Pmekanik/60313021 from USM due to sponsored this paper.

8. References

1. Airplane crashes [Internet] July 2014 [updated January 2015; cited March 2015] Available from: http://callsam.com/research-topic-item/other-case-types/airplanecrashes/
2. National Transportation Safety Board, Crash During Approach to Landing, Circuit City Stores, Inc., Cessna Citation 560, N500AT, Pueblo, Colorado, February 16, 2005, Aircraft Accident Report NTSB/AAR-07/02, PB2007-910403,Washington, D .C. 2007.
3. Papadakis M, Breer M, Craig N, Liu X. Experimental Water Droplet Impingement Data on Airfoils, Simulated Ice Shapes, an Engine Inlet and a Finite Wing. NASA Contractor Report. 1994; 4636.
4. Bidwell CS, Mohler SR Jr. Collection efficiency and ice accretion calculation for a sphere, a Swept MS (1)-317 Wing, a Swept NACA-0012 Wing Tip, an Axisymmetric Inlet and Boeing 737-300 Inlet. 33rd Aerospace Science Meeting and Exhibit sponsored by the AIAA, Reno, NV, USA. 1995 Jan 9-12; 95–0755.
5. Ramamurthy S, Keith TG Jr, De Witt KJ, Putt JC, Martin CA, Leffel KL. Numerical Modelling of an Advanced Pneumatic Impulse Ice Protection System (PIIP) for Aircraft. AIAA 29th Aerospace and Science Meeting, Rena, NV, USA, AIAA. 1991 Jan 7-10; 91–0555.
6. Martin CA, Putt JC. An Advanced Pneumatic Impulse Ice Protection System (PIIP) for Aircraft. AIAA 28th Aerospace Science Meeting, Rena, NV, USA, AIAA. 1990 Jan 8-11; 90–0492.
7. Zumwalt GW. Electromagnetic impulse de-icing applied to a nacelle nose lip. 21st Joint Propulsion Conference, Monterey, California, USA, AIAA. 1985 Jul 8-10; 85–1118
8. Labeas GN, Diamantakos ID, Sunaric MM. Simulation of electroimpulse de-icing process of aircraft wings. Journal of Aircraft. 2006; 43(6):1876–85.
9. Egbert RI, Schrag RL, Bernhart WD, Zumwalt GW, Kendrew TJ. An investigation of power line de-icing by electro-impulse methods. Journal of IEEE Transactions on Power Delivery. 1989; 4(3):1855–61.
10. Dryden JJG, Aurora RBI. Electro-magnetic expulsion de-icing system. US Pattern 6102333A.
11. The jet engine Rolls-Royce 5th edition. 1996; 147–51.
12. Botura GC, Dave Sweet MS, Flosdroft D. Development and demonstration of low power electrothermal de-icing system. 43rd AIAA Aerospace Science and Exhibit, Reno, NV, USA, AIAA. 2005 Jan 10-13; 2005–1460.
13. Al-Khalid K. Thermo-Mechanical Expulsive De-icing System – TMEDS. AIAA 45th Aerospace Science Meeting and Exhibit, AIAA. 2007 Jan 8-11; 2007–692.
14. Al-Khalil K, Ferguson T, Philips D. A hybrid anti-icing ice protection system. 35th Aerospace Sciences Meeting and Exhibit, Reno, NV, USA, AIAA. 1997 Jan 6-9; 97–0302.
15. Hua J, Liu HT. Fluid Flow and thermodynamic analysis of a wing anti-icing system. Canadian Aeronautics and Space Journal. 2005; 51(1):35–40.
16. Brown JM, Raghunathan S, Watterson JK. Heat transfer correlation for anti-icing system. Journal of Aircraft. 2002 Jan-Feb; 39(1):65–70.
17. Raghunathan S, Benard E, Watterson JK, Cooper RK, Curran R, Price M, Yao H, Devine R, Crawford B, Riordan D, Linton A, Richardson J, Tweedle J. Key aerodynamic technologies for aircraft engine nacelles. The Aeronautical Journal. 2006 May; 110(17):265–88.
18. Brown JM. Investigation of Heat Transfer between a Staggered Array of Hot Air Jets and a Jet Engine Nacelle Lipskin Surface. [PhD Thesis]. Queens University Belfast, 1999.
19. Saeed F, Al-Garni AZ. Numerical simulation of surface heat transfer from an array of hot air jets. 25thApplied Aerodynamics Conference, Miami, FL, AIAA. 2007 Jun 25-28. p. 2007–4287.

20. Hua J, Kong F, Liu HT. Unsteady thermodynamic CFD simulation of aircraft wing anti-icing operation. Journal of Aircraft. 2007; 44(4):1113–7.
21. Domingos RH, Papadakis M, Zamora A. Computational methodology for bleed air ice protection system parametric analysis. Atmospheric and Space Environments Conference, Toronto, Ontario Canada, AIAA. 2010 Aug 2-5. p. 2010–7834.
22. Pellissier M, Habashi WG, Pueyo A. Design optimization of hot air anti-icing by FENSAP-ICE. 48th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition. Orlando, Florida. 2010 Jan 4-7.
23. Ansys Fluent Teory Guide. [Internet] July 2014 [updated January 2015; cited March 2015] Available from: http://148.204.81.206/Ansys/150/ANSYS%20Fluent%20Theory%20Guide.pdf
24. Wong SH, Papadakis M, Zamora A. Computational investigation of bleed air ice protection system. 1st Atmospheric and Space Environments Conference, San Antonio, Texas, AIAA. 2009 Jun 22-25; 2009–3966.