ISSN (Print) : 2319-8613
ISSN (Online) : 0975-4024

COMPARISON OF ROTATING PICCOLO TUBE WITH FIXED PICCOLO TUBE BY USING CFD

[1] Rohini D, [2] Lokesgarun D, [3] Naveen R and [4] Samiyappan P
[1] Department of Aeronautical Engineering,
Bannari Amman institute of Technology, Erode, India-638401
rohinidharmaraj90@gmail.com
[2] Department of Aeronautical Engineering,
PARK College of Engineering and Technology, Coimbatore
lokeshwaran90@gmail.com
[3] Department of Aeronautical Engineering,
Bannari Amman institute of Technology, Erode, India-638401
aeror.naveen@gmail.com
[4] Department of Aeronautical Engineering,
PARK College of Engineering and Technology, Coimbatore
samiyappanaero91@gmail.com

ABSTRACT - The main objective of this paper is to design and analysis and also compared the results with anti-icing piccolo tube concept with the rotation piccolo tube concept for performance improvement. Ice build-u on aircraft affects the efficiency of aircraft during in flight in general ice formation on a wing produces significant reduction of lift and increase in drag and results an increased stall speed. It occurs in both cases like in ground or in flight. In ground condition, due to the accumulation of moisture on the surface of wing and forms the ice. In flight conditions, the aircraft is flying at an altitude of above 6km and which passes through the cloud and ice formation occurs over the wing .to avoid this, anti-icing should be done over the wing for increasing the performance of aircraft. The final results described the comparison of contour diagrams of both configurations.

Keywords: Anti-Icing, Piccolo Tube, Aircraft performance, Drag

1. INTRODUCTION

Ice formation on an aircraft wing is one of the major problems encountered in aircraft industries. Ice formed on the wing both onflight and inflight conditions we can encounter the ice formation in onflight condition by visual inspection before flight. But inflight condition we can't solve ice formation by simple information. Two different methods are there Anti-icing and Di- icing.

Anti-icing systems are used to breaks the ice formation from structures. This system has mechanical systems to break the already formed ice. For this anti icing process mechanically deformable membranes or electro-thermal devices, are designed to prevent ice formation by evaporating the impinging droplets [2].

One of the most widely used anti-icing devices for wing is high temperature bleed air anti-icing system, commonly called piccolo tube. This system will circulate the high velocity hot air, from the engine compressor. This system has staggered holes around the tube [3]. These holes will eject the high velocity hot air to inner surface of the wing structure and heat conduction from inner to surface to outer surface (fig:1). With this process water droplet will be evaporated for wing structure.

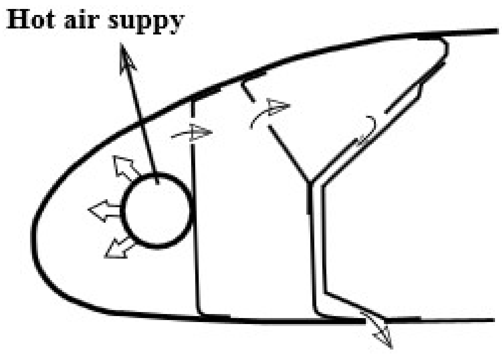

Fig: 1 Thermal Anti-icing system using piccolo tube. [1]

2. ANTI-ICING

The prevention of the formation of ice upon any object, especially aircraft, through wind shield sprayer, in addition of antifreeze materials to the carburetor, or heating the wings and tail. These methods used to protect the formation of ice over the critical areas of aircraft. Anti-icing is understood to be the uses of chemicals that not only de-ice, but also remain on a surface and continue to delay the reformation of ice for a certain period of time, or prevent from this to make mechanical removal easier. In-flight icing requires rigorous procedures and systems coupled to ice protection systems (de-icing or anti-icing systems), usually located at the leading edge of the exposed surfaces [4]. De-icing systems are reactive and commonly consist in mechanically deformable membranes or electro-impulse devices. Such systems are used periodically to remove already accreted ice. Anti-icing systems, like hot –bleed-air circulation systems or electro thermal system are proactive and designed to protect ice growth by dispersing the impinging droplets. Thermal heating systems are most frequently used on windshields and propellers. Large, turbine powered aircraft typically are equipped with anti-ice systems that use hot compressed air (called bleed air) that is tapped of the compressor section of the engines to prevent ice formation on critical engine components for example, air inlet lip and the turbine engine inlet guide vanes. This heated air is also ducted to airframe parts, the wing and leading edges. But, piston and turboprop-powered general aviation airplanes do not have bleed air anti-ice systems [5].

Types of Anti-icing method a) Propeller Anti-Icers

b) Windshield Anti-Icers

c) Thermal Anti-icing d) Piccolo tube

e) Impinging Jet Flow

2.1 Piccolo Tubes

An integrated system mainly composed of a tube located inside wing leading edge. Guiding the bleed air collected from the engine's first compressor the piccolo tube function is primarily to heat up the skin so as to evaporate most of the impinging water droplets, prevent dangerous ice accumulation on the wings [6].

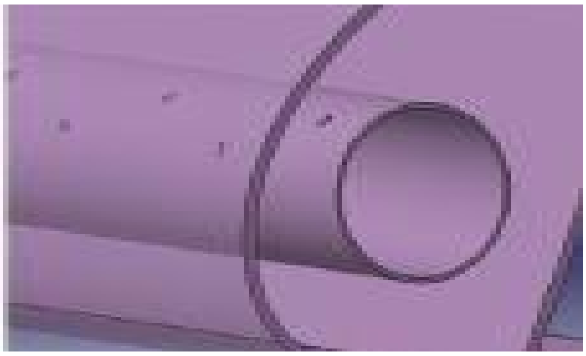

Fig: 2 piccolo tube

2.2 Benefits of Anti-Icing

The benefits of anti-icing are considerable. Pre-treating surfaces with ice melter before a storm arrives can increase winter safety, make subsequent snow and ice removal easier and less costly, minimize de-ice usage and reduce the potential for impact on properties and the environment.

- Increased safety
- Reduced de-icier use
- Labor and cost savings
- A sound strategy

3. DESIGN CONCEPT

In the present study, a NACA 2412 airfoil was chosen for the wing section as shown in fig (3).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 1.5m was selected, conforming to the wing dimensions of a typical aircraft.

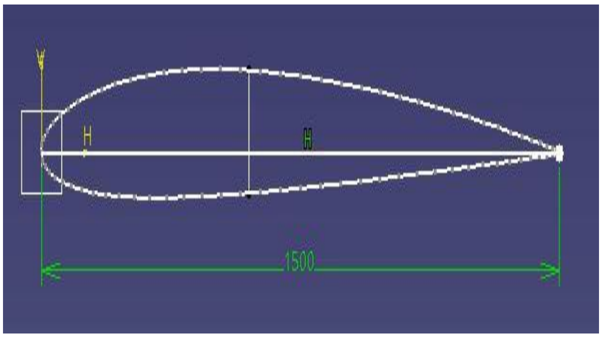

Fig: 3 schematic diagram of NACA 2412 Airfoil section

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 may 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 wing models were created using CATIA V5 design software. The 2D wing model shown in fig4.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 36mm and hot air injection jet diameter of 1.5mm.The model also comprised two vent holes/passages for the hot air to escape on the upper and lower side of the wing. Three holes of hot air jets fig (5) were located on the piccolo tube at 0^0 (center), $+45^0$ and -45^0

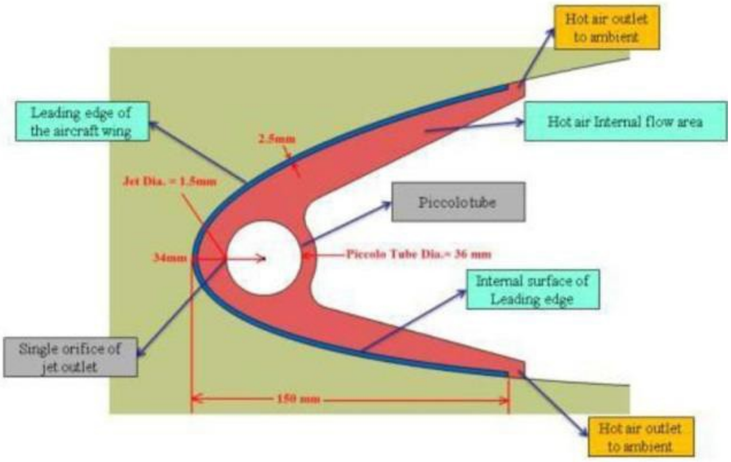

Fig: 4 2D wing model [1]

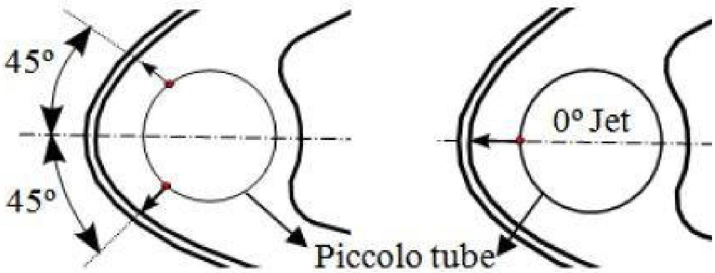

Fig: 5 Angular positions of hot air impinging jets [1]

3.1 Model Description

The 2D wing segment comprises the following component. The hot air, which is introduced by piccolo tube into the leading edge anti-icing bay Many number of small holes present on the front side of the piccolo tube from that the hot jet enter the inner surface of the leading edge.

The anti-icing operation, aluminium skin is heated by the hot air. Lower side of the exhaust holes allow the hot air to exit to the external flow. The heat shield act as the back wall of the bay Meshes are generated for both internal and external flow fields. The interior mesh and exterior mesh are connected through the mesh in the exhaust holes. A mesh is developed inside the aluminium skin for the heat conductivity calculation. By creating high efficiency mesh 2D steady flow simulation has been carried out in the Piccolo tube. A 2D model wing segment is generated by using CatiaV5

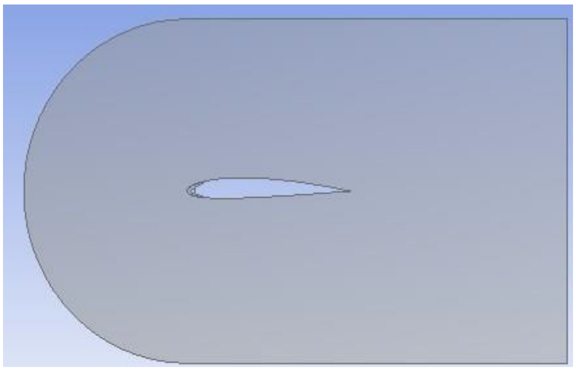

Fig: 6 Model with domain

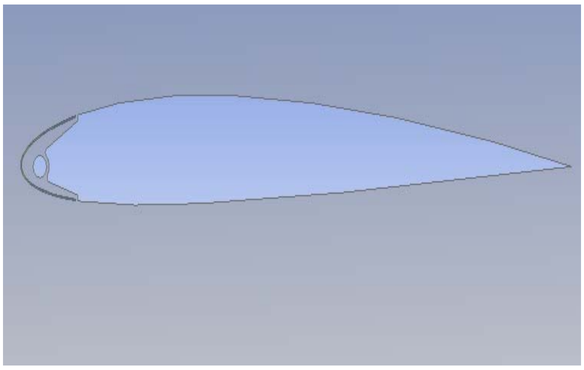

Fig.7 Detail view of aero foil

The flow domain was meshed with mapped elements for 2D simulation (Fig.6) and with hexahedral elements for using ICEM CFD software. The overall computational domain for 2D case, along with a few blown up sectional views, is shown in Fig.8. The numerical analysis was carried out using ANSYS FLUENT SOFTWARE, using standard k-ε turbulence model.

The wing external condition was set for a typical flight Mach number of 0.3. The corresponding ambient temperature was taken as 300 K. the ambient conditions of temperature and pressure also prevailed at the inner domain of wing containing piccolo tube. The hot air pressure and temperature at piccolo tube inlet were $97500P_a$ and 453K respectively. The above cases of hot air jet(s) coming out of piccolo tube were considered in the 2D model.

3.2 Phase of Analysis

STEP 1: PRE-PROCESSING OR MODELLING THE STRUCTURE

- Modeling
- Meshing
- Finalizing boundary condition
- Applying boundary condition
- Analysis

STEP 2: POST – PROCESSING

- Visualizing the temperature distribution over the wing by using temperature distribution over the wing by using temperature contour
- Comparing the result of two models

4. DESIGN AND CFD ANALYSIS

4.1 Design of the Anti-icing system:

The 2D wing model piccolo tube has been created by using CATIA V5 software as shown in fig (8), are generic models provided by commercial aircrafts which experimental data has been obtained. A piccolo tube inside the wing and close-up of the anti-icing system shown in fig (9)

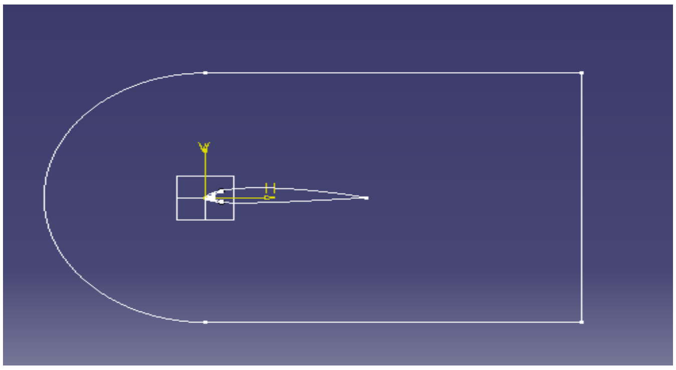

Fig: 8 Design of NACA 2412 Anti-icing system

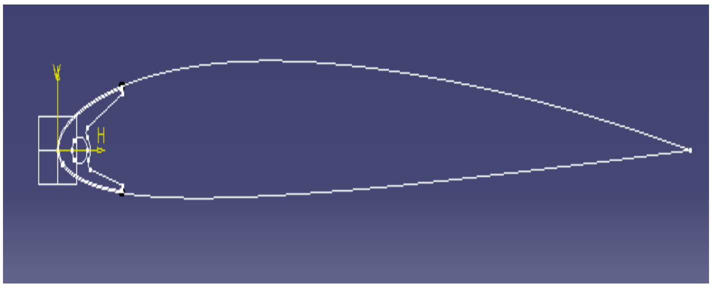

Fig: 9 wing and piccolo tube geometric configuration

The piccolo tube is an integrated system mainly composed of a tube located inside the leading edge of the wing, guiding the bleed air collected from the engine's first compressor. This tube is perforated with a number of holes (usually two or three, in this case uses of hole three) of small holes, the blow the hot air into the wing's leading edge inner skin surface, the function of the piccolo tube is primarily to heat up the skin so as to evaporate most of the impinging water droplets, and there by prevent dangerous ice accumulation on the wings.

Fixing and arranging the geometry are important steps in order to generate the CFD mesh properly and smoothly run the CFD computation, and get accurate results. The adjustments on the geometry aim to keep only the relevant features while discarding unnecessary complexities.

Also, the quality of the surfaces when importing from the virtual design process to the mesh process is crucial. Indeed, if the mesh is projected to the cad (computer Aided Design) surface, this case will become more critical so, in this specific case where heat fluxes and flow variables, that are to be extracted at the wall. As a proof of concept, geometric parameters were considered as design variables, that a piccolo tube-wing surface spacing of 10mm gives desired temperature distribution on the outer and inner surface of the wing. Piccolo tube diameter is 36mm.The jet orientation angle for each holes $\{45^0 < \Theta < 0^0$ and $0^0 < \Theta < 45^0\}$.

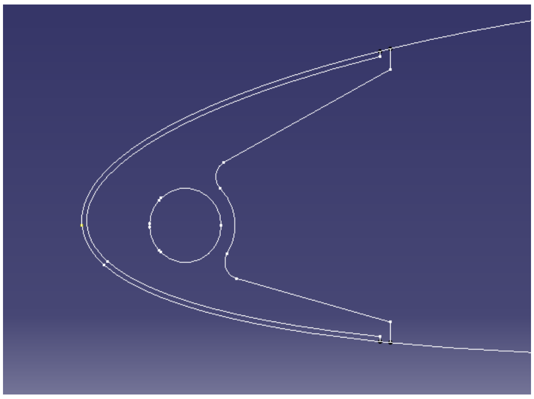

Fig 10: The jet orientation

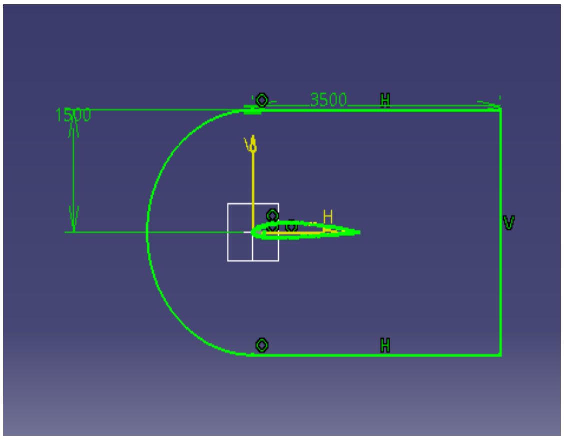

Fig11: Geometry module

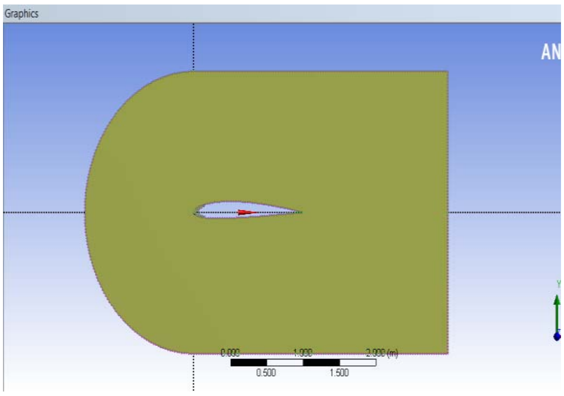

Fig12: Domain surface in the geometry module

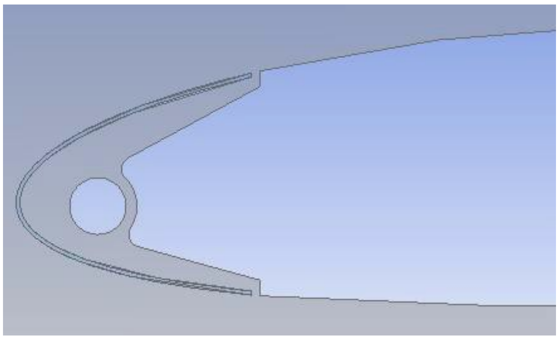

Fig13: Bay model

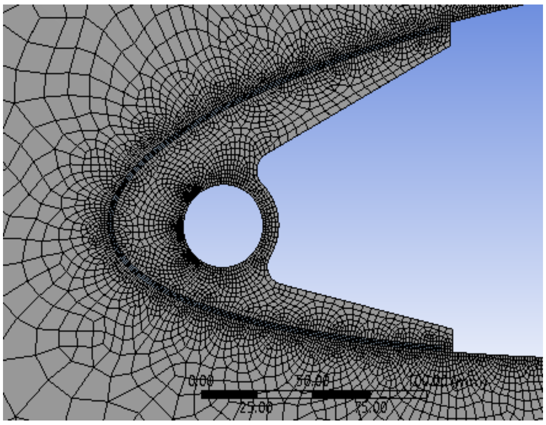

Fig.14 Meshed model

4.2 Boundary Conditions

To perform CFD analysis for fixed Piccolo tube model and rotating piccolo tube model has following specifications is given in fluent15.0 which is referred in the journal.

- Fixed piccolo tube model: a. Solver: pressure based. b. Time: steady
  - c. Turbulence Model: SST,K-ε (2- Equation) is used
  - d. Pressure velocity coupling : coupled scheme.
  - e. Boundary conditions are,

1. Inlet : Velocity inlet = 96.5m/s,thermal = 253K
2. Outlet: pressure outlet
3. Secondary inlet: Pressure-inlet
4. Gauge total pressure=90Kpa,thermal = 454K

- Rotating Picolo tube model:
  - a. Solver:pressure based b. Time: steady
  - c. Turbulence model: Realizable,K-ε (2equation)is used
  - d. Near wall treatment: Enhanced wall treatment
  - e. Pressure velocity coupling: Coupled scheme
  - f. Boundary conditions are,

1. Inlet: velocity inlet=96.5m/s, thermal=253K
2. Outlet: pressure outlet
3. Pipe wall: moving wall
4. Motion: Rotational relative to adjacent cell zone
5. Secondary inlet: pressure inlet
6. RPM:85
7. Gauge Total Pressure = 90000 Pascal.
8. Thermal = 454K

5. RESULT

1. These two models are analyzed and observed the temperature distribution along either side of the chord line and it is absorbed that fixed model gives more uniform temperature distribution.
2. This result appear to be optimized when compared to the rotational piccolo tube model and fixed model gives more effective results compared to rotating piccolo tube.

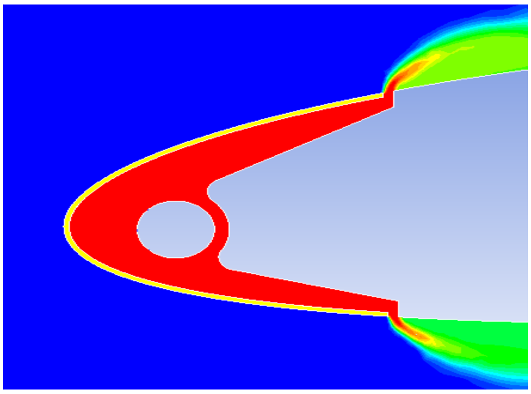

Fig.15 Fixed piccolo tube model

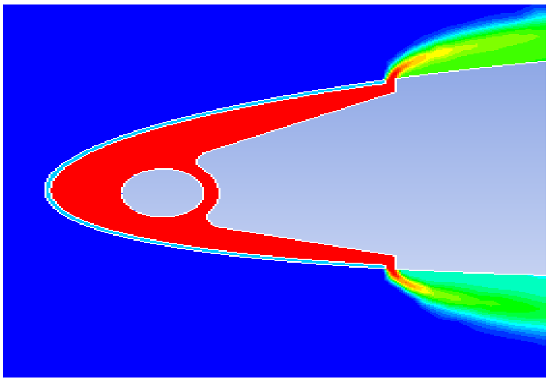

Fig.16 Rotating Piccolo tube model

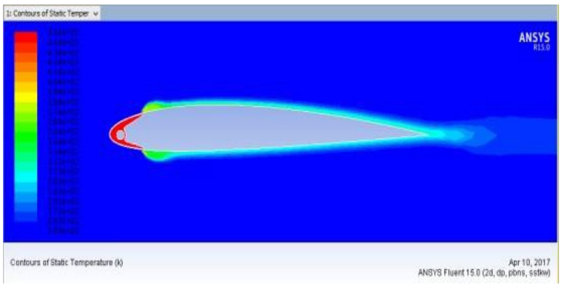

Fig.17 Fixed Piccolo tube model with wing

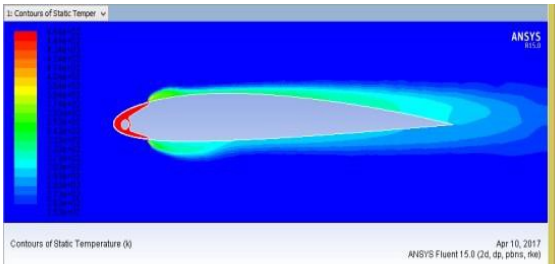

Fig.18 rotating piccolo tube model with wing

From the above result we concluded that fixed piccolo tube model gives better temperature distribution than the Rotating piccolo tube model.

6. CONCLUSIONS

The jet 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 shows in the figure total temperature distribution for all the impingement distance with +45^0 and -45^{0}jets. The hot air from piccolo tube holes is injected in temperature of 453K. The temperature distribution around the rotating piccolo tube and fixed piccolo tube almost symmetrical about both sides along the chord line Higher temperature is identified in the fixed piccolo tube model is compared to the rotational piccolo tube model.

However, a relatively temperature distribution is observed compared to the Rotational piccolo tube model. Also, a higher, but concentrated, temperature zone is obtained near the wing surface for fixed piccolo tube model. While lower temperature observed in the rotational piccolo tube model. In fixed piccolo tube model temperature distribution along the inner surface of the wing after the jets diverge at the upper and lower regions of the wing surface.

REFERENCES

[1] C. Sreedharan, Q. H. Nagpurwala, S. Subbaramu, “Effect of Hot Air Jets from a Piccolo Tube in Aircraft Wing Anti-Icing Unit”.
[2] JunHua, Fanmei Kong, Hugh, “Unsteady thermodynamic CFD Simulations of Aircraft Wing Anti-Icing Operation”.
[3] B. Balakrishna and V. Gangadhar Praveen Ketha. Ȧ, “Validation of Unsteady Thermodynamic CFD Simulations of Aircraft Wing Anti- Icing Operation”.
[4] Mathieu Pellissier, Wagdi G.Habashi and Alberto Pueyo, “Design optimization of hot-air anti-icing systems by FENSAP-ICE”, AIAA 2010-1238.H. S. Yan and W.R. Chen, "On the output motion characteristics of variable input speed servo-controlled slider-crank mechanisms," Mechanism and Machine Theory, vol. 35, pp. 541-561, 2000.
[5] Jun Hua, Hugh H.T. Liu, “Fluid Flow and Thermodynamic Analysis of a Wing Anti-Icing System”.
[6] Hugh H. T. Liu, and Jun Hua University of Toronto, Toronto, Ontario M3H 5T6, Canada “Three-Dimensional Integrated Thermodynamic Simulation for Wing Anti- Icing System”.