

T.R.
GEBZE TECHNICAL UNIVERSITY
GRADUATE SCHOOL

**DESIGN AND ANALYSIS OF ANTI-ICING SYSTEM FOR A
TURBOSHAFT ENGINE**



EMİR MUSA KOYUNCU

A THESIS OF MASTER OF SCIENCE
DEPARTMENT OF ENERGY TECHNOLOGIES
APPLIED PROPULSION SYSTEM DESIGN AND ENGINEERING
FOR AEROSPACE TECHNOLOGIES PROGRAM

ADVISOR: ASSIST. PROF. DR. SALİH ÖZEN ÜNVERDİ

OCTOBER 2025

T.R.
GEBZE TECHNICAL UNIVERSITY
GRADUATE SCHOOL

DESIGN AND ANALYSIS OF ANTI-ICING SYSTEM
FOR A TURBOSHAFT ENGINE

EMİR MUSA KOYUNCU

A THESIS OF MASTER OF SCIENCE
DEPARTMENT OF ENERGY TECHNOLOGIES
APPLIED PROPULSION SYSTEM DESIGN AND ENGINEERING
FOR AEROSPACE TECHNOLOGIES PROGRAM

ADVISOR: ASSIST. PROF. DR. SALİH ÖZEN ÜNVERDİ

OCTOBER 2025

T.C.
GEBZE TEKNİK ÜNİVERSİTESİ
LİSANSÜSTÜ EĞİTİM ENSTİTÜSÜ

TURBOŞAFT MOTORLARDA BUZLANMA ÖNLEME
SİSTEMİ TASARIMI VE ANALİZİ

EMİR MUSA KOYUNCU

YÜKSEK LİSANS TEZİ
ENERJİ TEKNOLOJİLERİ ANABİLİM DALI
HAVACILIK VE UZAY TEKNOLOJİLERİNDE UYGULAMALI
İTKİ SİSTEMİ TASARIM MÜHENDİSLİĞİ PROGRAMI

DANIŞMAN: DR. ÖĞR. ÜYESİ SALİH ÖZEN ÜNVERDİ

EKİM 2025

MASTER of SCIENCE JURY APPROVAL FORM

A thesis submitted by Emir Musa Koyuncu defended on 02/10/2025 before the jury formed with the 30/06/2025 date and 2025/34 numbered decision of the GTU Graduate Administration Board, has been accepted as a MASTER of SCIENCE thesis in the Department of Energy Technologies, Applied Propulsion System Design and Engineering for Aerospace Technologies Program.

JURY

MEMBER

(THESIS ADVISOR) : ASSIST. PROF. DR. SALİH ÖZEN ÜNVERDİ

MEMBER

: ASSIST. PROF. DR. OSMAN OKUR

MEMBER

: ASSOC. PROF. DR. BAHADIR DOĞAN

APPROVAL

Gebze Technical University Graduate Administration Board
...../...../..... date and/..... numbered decision.

SIGNATURE/SEAL



*To Mr. Ibrahim
Koyuncu, whose
lifelong efforts and
guidance have been
invaluable to me*

ABSTRACT

In aircraft and gas turbine engines, the icing phenomenon has posed a significant safety and performance challenge since the early days of aviation. Ice accretion on engine inlet surfaces throughout the flight envelope is a critical issue, particularly in icing-prone zones. These zones are typically defined by ambient air temperatures below 0 °C combined with visible moisture, such as when flying through clouds. In such conditions, accumulated ice can lead to Foreign Object Damage (FOD) by breaking off and entering the engine core. Even if the ice remains attached to the surface, it can still narrow the flow path, disrupt inlet aerodynamics, and distort the uniformity of the incoming airflow profile required for optimal engine performance.

In turboshaft engines, Inlet Particle Separator (IPS) structures are designed to prevent unwanted particles such as sand or dust from entering the engine. However, due to the same aerodynamic characteristics that enable them to separate heavy particles from the flow, these structures can also promote the collection of supercooled water droplets, making them highly susceptible to rapid ice accretion. To ensure uninterrupted engine operation, an anti-icing system must be integrated into the IPS design.

Developing such a system requires a multidisciplinary approach. This includes accurate aerodynamic modelling of the IPS, followed by droplet impingement and collection analyses—typically carried out using computational fluid dynamics (CFD) tools or experimental methods. These results are used as inputs in thermal models to estimate the heating requirements that would keep the IPS surfaces ice-free.

In this study, an anti-icing system was designed for the outer dome region of an existing IPS module in compliance with certification requirements. The design was then tested in an icing wind tunnel, and the numerical results were compared with the experimental data. The tunnel tests demonstrated that the thermal modelling was able to accurately predict the most critical test points in terms of icing risk.

Keywords: Anti-ice, Gas turbine icing, Thermal anti-ice system

ÖZET

Uçaklarda ve gaz türbini motorlarında buzlanma fenomeni, havacılığın ilk dönemlerinden bu yana önemli bir güvenlik ve performans sorunu olmuştur. Uçuş zarfı boyunca motor giriş yüzeylerinde buz birikimi uçuş güvenliği için önemli riskler oluşturur. Bu bölgeler genellikle ortam sıcaklığının 0 °C'nin altına düştüğü ve bulut içi gibi görünür nemin bulunduğu koşullarda oluşur. Böyle durumlarda biriken buz parçaları yüzeyden koparak motorun içine girip yabancı cisim hasarına yol açabilir. Buz yüzeyde kalsa dahi, akış yolunu daraltabilir, giriş aerodinamiğini bozabilir ve motorun optimum performansı için gerekli olan düzgün hava akış profilini bozar.

Turboşaft motorlarda, istenmeyen parçacıkları (toz, kum vb.) motordan ayırmak amacıyla kullanılan GPA (Giriş Partikül Ayırıcı) yapıları, akışı yönlendiren geometrik özellikleri sayesinde su taneciklerini de yüzeylerinde toplayarak hızlı buzlanmaya karşı savunmasız hale gelir. Bu nedenle, GPA tasarımına uygun bir buzlanma önleme sistemi gereklidir.

Bu tür bir sistemin geliştirilmesi multidisipliner bir yaklaşım gerektirir. Bu süreç, GPA'nın aerodinamik olarak modellenmesini, ardından süper soğumuş damlacıkların yüzeylere çarpma ve birikme analizlerini (genellikle HAD yazılımları veya deneysel yöntemlerle) içerir. Elde edilen sonuçlar, yüzeyin buz tutmaması için gerekli ısıtma ihtiyacını hesaplamak üzere termal modellere girdi olarak kullanılır.

Bu çalışmada, mevcut bir GPA (modülün dış kubbe bölgesinin) geometrisinin buzlanma önleme sistemi sertifikasyon isterlerine uygun şekilde tasarlanmıştır. Ardından tasarım buzlanma rüzgâr tüneline test edilerek nümerik sonuçlar test sonuçlarıyla karşılaştırılmıştır. Tünel testleri sonucunda buzlanma açısından en riskli tasarım koşullarının termal buz modelleme ile doğru tahmin edilebildiği görülmüştür.

Anahtar Kelimeler: Buzlanma önleme, Gaz türbinli motor buzlanma, Termal buz önleme

ACKNOWLEDGEMENTS

I would like to express my sincere gratitude to Abdullah Talha Aslan for his continuous mentorship and unwavering support throughout this journey. I am also deeply thankful to Dr. Erinç Erdem for his readiness to help and for generously sharing his extensive technical expertise. Finally, I wish to extend my heartfelt appreciation to my thesis advisor, Dear Assist. Prof. Dr. Salih Özen Ünverdi, for his constant availability, solution-oriented guidance, and steady support. This study would not have been possible without your invaluable contributions.



TABLE OF CONTENTS

	<u>Page</u>
ABSTRACT	vi
ÖZET	vii
ACKNOWLEDGEMENTS	viii
TABLE OF CONTENTS	ix
LIST OF SYMBOLS AND ABBREVIATIONS	x
LIST OF FIGURES	xi
LIST OF TABLES	xiii
1. INTRODUCTION	1
1.1 Icing Meteorology and Cloud Types	4
1.2 Effect of Liquid Water Content	5
1.3 Effect of MVD	6
2. LITERATURE SURVEY	8
2.1 Numerical and Thermal Modelling Approaches	8
2.2 Experimental Investigations on Guide Vanes and Airfoils	10
2.3 Engine Inlet and Inlet Particle Separator Oriented Studies	12
2.4 Thesis Objective	17
3. TESTING	19
3.1 Icing Test Similitude Methodology	20
3.2 Test Setup of the Current Study	21
4. ANALYTICAL MODELLING	26
4.1 Energy Balance and Messinger Thermal Ice Model	26
4.2 Droplet Distribution	31
4.3 Thermal Analysis Boundary Conditions	36
5. RESULTS	41
5.1 Validation	44
6. CONCLUSIONS AND FURTHER STUDIES	54
REFERENCES	55
BIOGRAPHY	58
PUBLICATIONS AND PRESENTATIONS FROM THE THESIS	59

LIST OF SYMBOLS AND ABBREVIATIONS

P_{amb}	: Ambient pressure (kPa)
$\dot{Q}_c$	: Convection heat flux ($W.m^{-2}$)
H	: Convective heat transfer coefficient ($W.m^{-2}.K^{-1}$)
λ_r	: Darcy friction coefficient for rough surface
λ_s	: Darcy friction coefficient for smooth surface
ρ	: Density ($kg.m^{-3}$)
C_d	: Drag coefficient
V_n	: Droplet velocity ($m.s^{-1}$)
μ	: Dynamic viscosity ($Pa.s$)
$\dot{Q}_e$	: Evaporation heat flux ($W.m^{-2}$)
T_∞	: Free stream temperature (K)
V_{inf}	: Free stream velocity ($m.s^{-1}$)
K	: Inertia parameter
$\dot{Q}_k$	: Kinetic energy of droplets heat flux ($W.m^{-2}$)
L_e	: Latent heat of vaporization ($J.kg^{-1}$)
β	: Local collection efficiency
S_L	: Longitudinal pitch (m)
$\dot{m}_e$	: Mass flux of evaporating water ($kg.s^{-1}$)
K_0	: Modified inertia parameter
N_u	: Nusselt number
$P_{v,l}$	: Partial pressure of water vapor in the bulk air (kPa)
$P_{v,s}$	: Partial pressure of water vapor on the surface (kPa)
Pr	: Prandtl number
$\dot{R}_w$	: Rate of water catch ($g.m^{-2}.s^{-1}$)
Re	: Reynolds number
R_{air}	: Specific gas constant for dry air ($J.kg^{-1}K^{-1}$)
R_{wa}	: Specific gas constant for water vapour ($J.kg^{-1}K^{-1}$)
C_p	: Specific heat ($J.kg^{-1}.K^{-1}$)
λ	: Stokes number
T_s	: Surface temperature (K)
S_T	: Transverse pitch (m)
$\dot{Q}_a$	: Viscous heating heat flux ($W.m^{-2}$)
$\dot{Q}_w$	: Warming of droplets heat flux ($W.m^{-2}$)
CFD	: Computational Fluid Dynamics
DAQ	: Data acquisition system
FEM	: Finite element method
IGV	: Inlet guide vane
IPS	: Inlet particle separator
IWT	: Icing wind tunnel
LWC	: Liquid water content ($g.m^{-3}$)
MVD	: Mean volume diameter (μm)
RANS	: Reynolds-Averaged Navier-Stokes
RMS	: Root mean square
RPM	: Rotations per minute
SLIWT	: Sea level icing wind tunnel

LIST OF FIGURES

	<u>Page</u>
Figure 1.1: Ice formation on an IGV structure [1].	1
Figure 1.2: Schematic of an engine intake anti-icing system [2].	2
Figure 1.3: T700 turboshaft engine anti-ice system circuit [3].	2
Figure 1.4: EH101 engine bay cooling scoop icing [4].	3
Figure 1.5: Stratiform clouds.	4
Figure 1.6: Cumuliform clouds.	5
Figure 2.1: Schematic of the analysis setup and physical model [10].	9
Figure 2.2: Predicted collection efficiencies [13].	11
Figure 2.3: 100% LWC test results from the icing wind tunnel [21].	12
Figure 2.4: Contour plot of the local water catch coefficient (β) on IPS internal surfaces under reference conditions (MVD = 20 μ m, LWC = 2 g/m ³ , U_0 = 65 m/s) [15].	13
Figure 2.5: Effect of liquid water content (LWC) on ice accretion [7].	14
Figure 2.6: Velocity distribution before and after ice accretion [7].	14
Figure 2.7: NH90 intake icing tests (1200 m altitude, 150 km/h, $T = -5^\circ\text{C}$, LWC=0.7) [16].	15
Figure 2.8: Anti-icing air flow of T800 engine [19].	16
Figure 2.9: Inlet particle separator and outer dome region section.	17
Figure 3.1: Example IWT setup.	19
Figure 3.2: Test rig setup for the IPS component.	22
Figure 3.3: Test rig infrastructure and test piece 3d model.	23
Figure 3.4: Spray bar array and inlet bellmouth.	23
Figure 3.5: Spray bar configuration.	24
Figure 3.6: Schematic of tunnel calibration sensors.	24
Figure 3.7: IPS outer dome metal thermocouple instrumentation locations.	25
Figure 4.1: Messinger icing model representation [18].	27
Figure 4.2: Net surface heat load plot for example condition.	30
Figure 4.3: Messinger heat flux terms visualized (red indicates heat gain, blue indicates heat loss).	31
Figure 4.4: Design cases droplet distributions.	35
Figure 4.5: Real and conservative droplet distributions.	36
Figure 4.6: Representation of physical problem.	36
Figure 4.7: Heat transfer enhancement dimples at the hot-air supply route.	37
Figure 4.8: Defining ST and SL [27].	38
Figure 4.9: Case-3 dimpled vs non-dimpled geometry temperature difference contour.	39
Figure 4.10: Solid domain mesh structure from 2d thermal solver.	40
Figure 5.1: Thermal modelling and calibration flowchart.	41
Figure 5.2: Case-1 metal temperature contour for dry and wet solutions.	42
Figure 5.3: Case-4 metal temperature contour for dry and wet solutions.	42
Figure 5.4: Heat flux terms on all the design cases.	43

Figure 5.5: Case 1-a test metal thermocouple measurements and model temperatures.	46
Figure 5.6: Case 1-b test metal thermocouple measurements and model temperatures.	47
Figure 5.7: Case 2-a test metal thermocouple measurements and model temperatures.	48
Figure 5.8: Case 2-b test metal thermocouple measurements and model temperatures.	49
Figure 5.9: Case-3 test metal thermocouple measurements and model temperatures.	50
Figure 5.10: Case-4 test metal thermocouple measurements and model temperatures.	51
Figure 5.11: Case-5 test metal thermocouple measurements and model temperatures.	52
Figure 5.12: Borescope image from Case-3.	53

LIST OF TABLES

	<u>Page</u>
Table 2.1: Design cases.	18
Table 4.1: Polynomial coefficients of saturation pressure.	28
Table 4.2: Parameters for example heat flux calculations.	29
Table 4.3: Langmuir-D distribution for 20 μ MVD [24].	33
Table 4.4: Heat transfer correction regarding the dimple row number [27].	39
Table 5.1: Design cases to test with additional test points.	44
Table 5.2: RMS errors (test-model) for test cases 1-2-3.	45
Table 5.3: Case 1-a cooling values on the metal thermocouple locations for test and model.	46
Table 5.4: Case 1-b cooling values on the metal thermocouple locations for test and model.	47
Table 5.5: Case 2-a cooling values on the metal thermocouple locations for test and model.	48
Table 5.6: Case 2-b cooling values on the metal thermocouple locations for test and model.	49
Table 5.7: Case-3 cooling values on the metal thermocouple locations for test and model.	50
Table 5.8: Case-4 cooling values on the metal thermocouple locations for test and model.	51
Table 5.9: Case-5 cooling values on the metal thermocouple locations for test and model.	52

1. INTRODUCTION

Atmospheric icing is a serious operational hazard that remains a persistent challenge in aviation safety. It occurs when aircraft surfaces encounter supercooled water droplets—typically while flying through visible moisture in clouds—under ambient temperatures below 0 °C. Upon contact, these droplets freeze and form ice on various parts of the airframe or propulsion system. The consequences of ice accretion range from aerodynamic performance degradation to severe engine malfunctions such as compressor stall, flameout, and even mechanical damage caused by ice shedding.

While airfoils and pitot tubes are commonly discussed in the context of icing, engine inlets and internal components such as guide vanes, spinner sections, and separators are equally vulnerable. Ice accumulation in these regions may disturb airflow uniformity, reduce compressor efficiency, or in worst-case scenarios, lead to Foreign Object Damage (FOD).

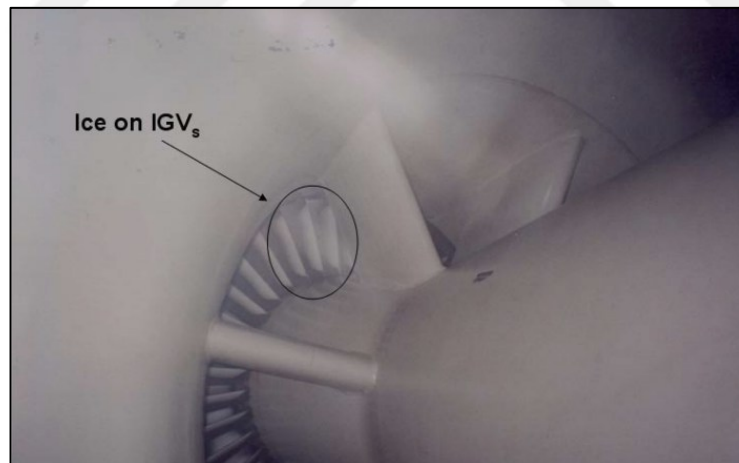


Figure 1.1: Ice formation on an IGV structure [1].

To mitigate these hazards, aircraft engines employ dedicated anti-icing and de-icing systems. Anti-icing techniques function by maintaining the surface temperature above the freezing point, thereby preventing ice accretion. A widely applied method is the extraction of hot-air from compressor bleed ports and its distribution to vulnerable structures. While effective, this approach introduces a trade-off: excessive heating

reduces engine performance, whereas insufficient heating risks incomplete protection. An example hot-air supplied anti-icing system is shown in Figure 1.2.

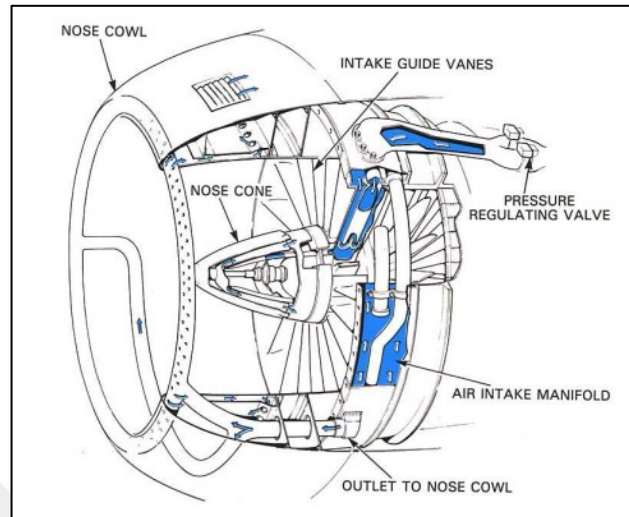


Figure 1.2: Schematic of an engine intake anti-icing system [2].

In addition to large turbofans, smaller turboshaft engines adopt integrated anti-icing concepts that diverge from traditional bleed-air-only designs. A representative example is the General Electric T700 [3], which incorporates an Inlet Particle Separator (IPS) within its forward section. While the IPS is primarily intended to protect the core from sand and dust erosion, it also employs a dual protection scheme. Thermal energy is provided by routing hot oil through structural members and supplementing it with regulated bleed air directed toward sensitive areas such as the inlet guide vanes, splitter nose, and swirl frame.

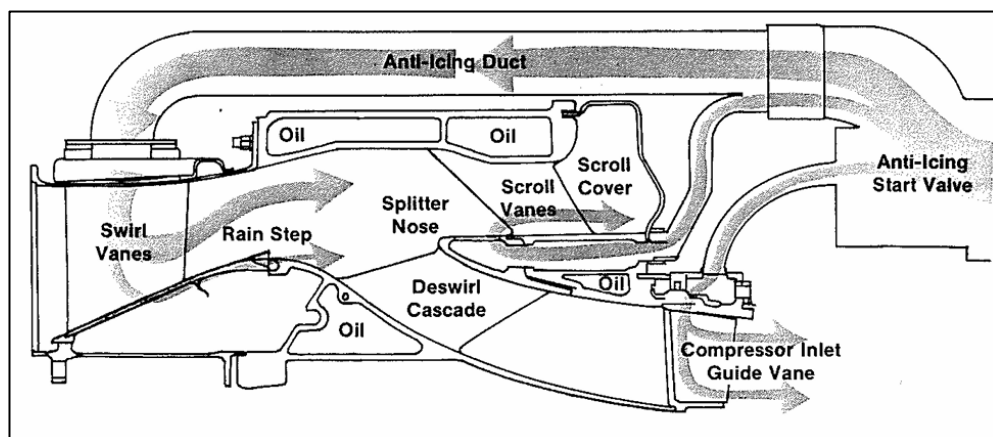


Figure 1.3: T700 turboshaft engine anti-ice system circuit [3].

It should also be emphasized that engine icing is not limited to the inlet itself. Several subsystems are susceptible, such as temperature and pressure probes, oil lines, hydraulic actuators, and bleed valves. Ice formation in these areas may obstruct flow passages, distort sensor readings, disrupt thermal management, or ultimately compromise engine reliability. For this reason, comprehensive ice protection strategies often extend beyond the primary flow path and cover a wide range of supporting components.

A related case is the EH101 [4] helicopter, where icing was reported not only at the main engine inlet but also at auxiliary cooling bay scoops. To address this, a ‘fully evaporative’ thermal protection strategy was applied, which proved effective under both glaze and rime icing conditions.



Figure 1.4: EH101 engine bay cooling scoop icing [4].

From these cases, a common conclusion can be drawn; effective ice protection relies on maintaining exposed surfaces above the freezing point, regardless of variations in geometry, operating conditions, or heating method. Whether achieved by hot-air bleed, oil circulation, or electrothermal heaters, the fundamental challenge remains delivering enough thermal energy to counteract external icing loads. Accordingly, it becomes critical to quantify the heat demand required to sustain an ice-free state. This demand is governed not only by surface geometry and the surrounding convective environment, but also by the additional sensible and latent heat loads introduced when supercooled droplets impinge on the surface.

1.1. Icing Meteorology and Cloud Types

Atmospheric conditions play a crucial role in icing risk. Meteorologically, both cloud altitude and type strongly influence the amount of supercooled water available to form ice on aircraft surfaces.

Supercooled droplets typically exist in visible moisture environments at temperatures below 0 °C. These droplets can accrete on aircraft components, leading to aerodynamic degradation or even engine malfunction when they freeze.

Two major cloud categories are associated with icing risk:

Stratiform clouds: Horizontally extensive and layered, they cover broad areas but usually have moderate liquid water content (LWC). These clouds can cause milder, longer-duration icing. An example of stratiform clouds is given in Figure 1.5.



Figure 1.5: Stratiform clouds.

Cumuliform clouds: Characterized by vertical development and a more localized nature, these clouds often contain higher LWC. The icing they produce can be more intense, even over shorter exposures. An example of cumuliform clouds is given in Figure 1.6.



Figure 1.6: Cumuliform clouds.

To ensure safe engine operation under such conditions, certification standards specify design in icing envelopes. The EASA standard CS-E 780 [5] (Amendment 7) provides a test-point table detailing temperature ranges, droplet sizes, LWC levels, and altitude limits for various icing scenarios—including supercooled large drops, freezing fog, and ice crystals. This table serves as the benchmark for evaluating engine inlet anti-icing system performance. Also in the table “Continuous Max” values represent stratiform, “Intermittent Max” values represents cumuliform clouds.

Table 1.1: CS-E 780 standard test points table [5].

Ambient Air Temperature (°C)	Altitude (ft)	(m)	Liquid Water Content (LWC) (g/m ³)-(a) Continuous Max	Liquid Water Content (LWC) (g/m ³)-(b) Intermittent Max	Median Volumetric Diameter (μm)
-10	17000	5182	0.6	2.2	20
-20	20000	6096	0.3	1.7	20
-30	25000	7620	0.2	1.0	20

1.2. Effect of Liquid Water Content

Liquid Water Content (LWC) represents the mass of liquid water suspended in a unit volume of air, typically expressed in grams per cubic meter (g/m³). It is one of the most critical parameters influencing aircraft icing risk, as higher LWC leads to a greater volume of water impinging on aircraft surfaces, thereby increasing both the rate and severity of ice accretion.

LWC not only governs how much ice forms but also influences the morphology of the ice. As LWC increases, ice shapes may evolve from smooth rime formations to more hazardous horn shapes [6] due to the increased mass and energy of incoming droplets. These geometries disturb the local airflow more significantly, resulting in a higher aerodynamic penalty.

In addition, LWC directly impacts the thermal load on anti-icing systems. For evaporative systems, where all collected water must be converted to vapor to keep the surface dry, a higher LWC significantly increases energy requirements.

At elevated altitudes and lower ambient air temperatures the LWC naturally declines. This reduction is due to the increasing fraction of frozen droplets and the atmospheric drying effect at high altitudes. According to CS-E 780 [5] icing certification data, LWC becomes negligible in extremely cold conditions—typically below $-40\text{ }^{\circ}\text{C}$ —where the risk of liquid droplet impact becomes minimal or vanishes altogether. Therefore, accurate estimation of LWC, especially in critical icing conditions, is essential for the proper sizing and evaluation of aircraft anti-icing systems.

1.3. Effect of MVD

The Mean Volumetric Diameter (MVD) is a representative measure of droplet size within a cloud, defined as the diameter at which the volume of all droplets smaller than this value equals that of the larger droplets. In icing studies, MVD is a key parameter because it strongly influences both the impingement characteristics and the type of ice that forms.

As the MVD increases, droplets carry greater inertia, making them less capable of following local flow deflections. Consequently, larger droplets tend to impact surfaces further downstream or in regions of high curvature, broadening the impingement footprint. This generally enhances the probability of glaze ice formation, since the larger droplets generate a thicker water film upon impact, which may not freeze instantaneously. In contrast, smaller droplets (low MVD) follow the flow more closely and freeze rapidly, favoring rime ice accretion.

Numerical studies have confirmed these trends. For example, simulations of an Inertial Particle Separator (IPS) under varying droplet sizes (MVD = 5, 10, and 20 μm) demonstrated that ice thickness on hub and shroud surfaces grows with increasing

MVD due to the higher impact inertia of large droplets [7]. However, on splitter surfaces, the maximum accretion occurred at small diameters ($\approx 5 \mu\text{m}$), because larger droplets tended to bypass these regions instead of adhering to them.

Overall, the effect of MVD is twofold: it not only modifies the collection efficiency of droplets on critical structures but also dictates the dominant ice type and the severity of the thermal load that the anti-icing system must counteract.



2. LITERATURE SURVEY

Existing literature on engine icing covers a wide range of components, from nacelle lips and inlet guide vanes to more complex structures such as turboprop bearing assemblies and turboshaft engine intakes. Investigations in this field rely on a combination of wind tunnel experiments and numerical simulations, aiming to characterize droplet impingement, ice accretion processes, and the effectiveness of various protection concepts. Accordingly, the following review is organized into three subsections. The first outlines general numerical and thermal modelling approaches commonly applied to icing problems. The second addresses experimental investigations carried out on airfoils. Finally, the third subsection concentrates on engine inlets and Inlet Particle Separator (IPS) configurations.

2.1. Numerical and Thermal Modelling Approaches

In a Messinger-based numerical framework, Ignatowicz et al. [8] simulated the surface thermal response of an airfoil subjected to icing conditions, with the primary aim of predicting convective heat losses under varying environmental and surface parameters. The study emphasized the sensitivity of thermal models to surface roughness formulations, demonstrating that even minor variations in the heat transfer coefficient (H) can significantly affect the estimated thermal load required to maintain an ice-free condition. Although the simulation framework incorporated ice accretion logic, the core methodology—particularly the coupling of aerodynamic flowfield, water droplet impingement, and wall energy balance—offers valuable insight for purely anti-icing applications. The analysis supports the use of refined convective modelling techniques when estimating required internal heat fluxes in systems that aim to fully prevent ice formation, such as hot-air-based anti-icing configurations.

Fukudome et al. [9] conducted a CFD-based anti-icing analysis on a NACA0012 airfoil incorporating an asymmetrically arranged heating surface. The simulation framework included a coupled RANS solver, droplet impingement modelling, and surface energy balance equations to evaluate temperature distributions and ice protection effectiveness. Unlike traditional uniform heating setups, their configuration supplied thermal energy only to the upper half of the leading edge. Results showed

that such asymmetric heating led to incomplete ice prevention, with localized surface temperatures dropping below freezing on the unheated lower surface. Even when the heated area was increased by 30%, full protection could not be achieved without further thermal input or a change in heating layout. These findings underline the importance of tailoring heat flux distributions to the actual impingement and convective load profiles—particularly in designs aiming for minimal energy use while maintaining an entirely ice-free surface.

Domingos et al. [10] performed a comprehensive numerical investigation on a thermal anti-icing system using engine bleed air as the heat source for protecting an aircraft wing leading edge.

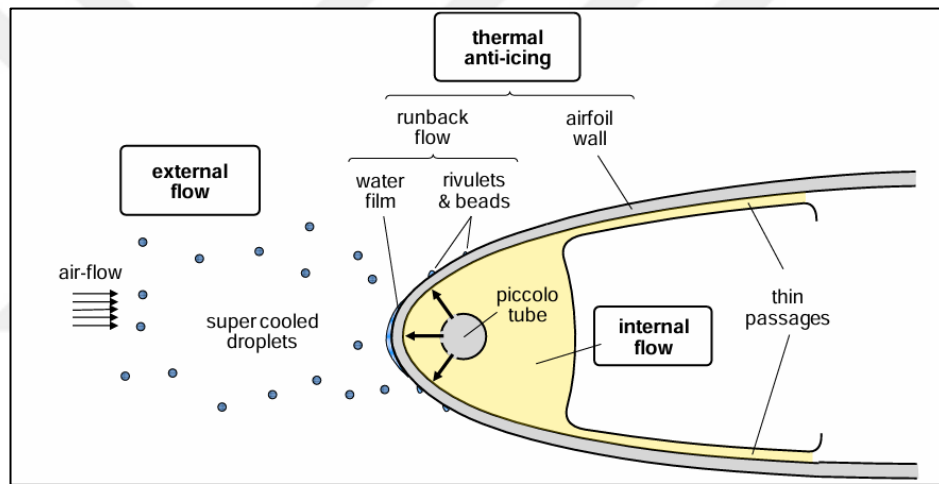


Figure 2.1: Schematic of the analysis setup and physical model [10].

Their approach combined conjugate heat transfer simulations with external convective and droplet-induced heat loads, aiming to determine the minimum internal energy supply required to maintain the surface above freezing. The model assumed a fully ice-free operating condition, aligning with anti-icing strategies rather than de-icing. Results indicated that while bleed air temperature plays a role, the mass flow rate of hot-air is the dominant factor in ensuring effective protection. For instance, even at supply temperatures exceeding 100 °C, insufficient flow rates resulted in localized freezing, especially near the stagnation zone. This finding reinforces the need for accurate internal heat flux prediction under prescribed flight conditions—a key consideration for optimizing anti-icing system design in both conventional wing sections and complex inlet geometries.

Bu et al. [11] conducted a numerical simulation of an electrothermal anti-icing system applied to a NACA 0012 airfoil, focusing on maintaining an entirely ice-free surface under various operating conditions. The model accounted for convective heat loss and droplet-induced thermal loads while simulating internal heating via a constant-temperature electrothermal layer. Their findings demonstrated that increasing flight speed and decreasing ambient temperature substantially elevated the required surface heat flux, especially near the stagnation zone. This region was shown to be the most thermally vulnerable due to high droplet impingement. The authors concluded that effective anti-icing design should include spatially resolved heat input to avoid localized freezing without excessive energy consumption elsewhere.

2.2. Experimental Investigations on Guide Vanes and Airfoils

Linkai Li et al. [12] conducted a comprehensive experimental study on a hot-air-based anti-icing system developed for the inlet guide vanes (IGVs) of aircraft engines. The test campaign involved an IGV prototype equipped with an internal U-shaped hot-air duct, evaluated under controlled icing conditions in a wind tunnel. Ice accretion and surface temperature distributions were measured using a combination of subsurface thermocouples and high-speed imaging techniques. Their results indicated that more than 80% of the thermal energy delivered to the surface was transferred via convection over the IGV.

Al-Khalil et al. [13] carried out a series of icing wind tunnel experiments using a NACA0012 airfoil to validate computational ice protection models and to generate benchmark data on water droplet impingement. Their investigation focused particularly on measuring the local droplet collection efficiency, β , across the leading edge under varying freestream velocities. The results demonstrated that collection efficiency is strongly influenced by inertial effects, especially near the stagnation region of the airfoil. For example, when the freestream velocity was doubled, the peak collection efficiency at the stagnation line increased (Figure 2.2) from approximately 0.5 to 0.6.

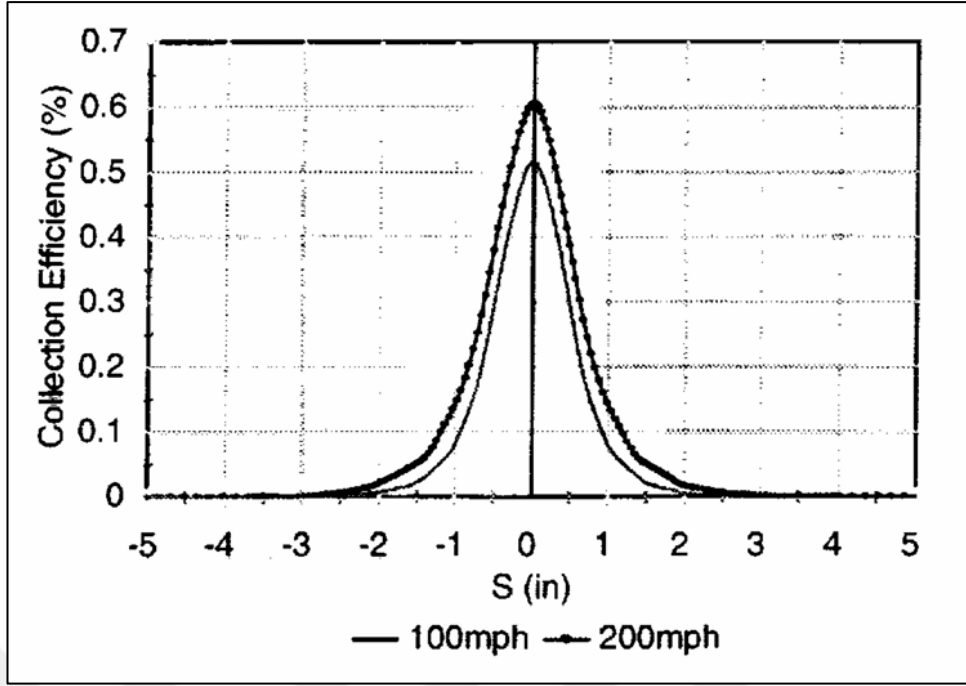


Figure 2.2: Predicted collection efficiencies [13].

This change indicates that higher momentum droplets are more likely to deviate less from the airstream path and directly impact the leading-edge surface. The study highlighted the importance of accurately capturing this velocity dependence in numerical icing models, particularly for estimating thermal loads and predicting the onset of ice accretion in anti-icing system design.

Castillo and Ortega [14] analyzed the development of the anti-icing system for the TP400-D6 turboprop engine, focusing on the Front Bearing Structure (FBS) that integrates the Inlet Guide Vanes (IGVs). Unlike earlier turboprop designs where structural and aerodynamic elements are separate, the TP400 combines both into a single radial structure, requiring the IGVs themselves to incorporate hot-air passages for thermal-ice protection.

A major challenge identified was the limitation of conventional thermal FEM tools, which could not account for the influence of supercooled droplet impingement and runback water. These omissions led to underprediction of heat demand and surface temperature gradients, especially near the trailing edge. To mitigate this, the authors proposed an iterative correction framework, combining FEM-calculated dry-surface temperatures with additional external heat flux terms derived from water interaction.



Figure 2.3: 100% LWC test results from the icing wind tunnel [14].

To validate the approach, icing wind tunnel experiments were performed in the Sea Level Icing Wind Tunnel (SLIWT) at Cranfield University, using a full-scale FBS sector under both certification and design icing envelopes. Instrumented vanes showed that metal temperature dropped sharply with increasing liquid water content (LWC). Ice formation consistently initiated at the trailing edge due to runback water (Figure 2.3), despite active hot-air supply. When calibrated with test data, the corrected FEM model provided accurate leading-edge predictions, though trailing-edge errors persisted, reflecting unresolved complexities in runback behavior.

The authors concluded that excluding water impingement and runback effects could introduce errors comparable to the magnitude of the predicted surface temperatures themselves. Their iterative FEM–test method was presented as a pragmatic tool to optimize bleed-air requirements and ensure robust IGV anti-icing design for next-generation turboprop engines.

2.3. Engine Inlet and Inlet Particle Separator Oriented Studies

A notable numerical study by Chen et al. [15] employed an Eulerian–Eulerian multiphase framework to analyze water droplet trajectories and impingement patterns within a generic IPS configuration. By varying medium volume diameter (MVD), liquid water content (LWC), and freestream velocity, the study quantified how these parameters affect droplet accumulation on IPS surfaces. The results revealed that droplet inertia plays a dominant role in determining accumulation regions, with larger droplets and higher velocities increasing impingement rates—particularly on the outer dome and inner dome walls. Importantly, the study also demonstrated that LWC alters the total thermal load but not the impingement locations or trajectories, indicating that heating requirements must be scaled with droplet flux rather than distribution. A representative contour of the local water catch coefficient (β), reproduced from this

study, is provided in Figure 2.4, highlighting the key impingement zones along the IPS walls.

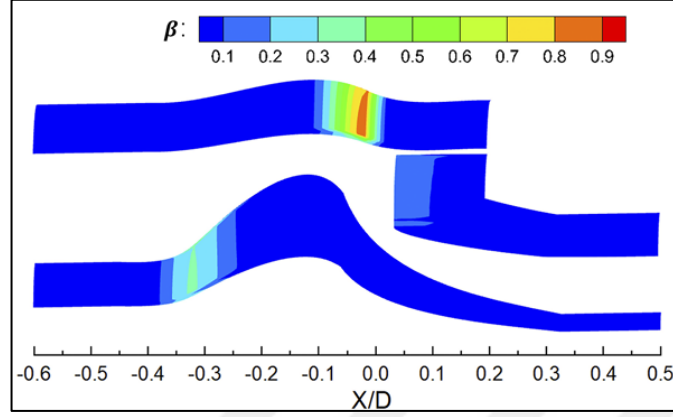


Figure 2.4: Contour plot of the local water catch coefficient (β) on IPS internal surfaces under reference conditions (MVD = 20 μm , LWC = 2 g/m^3 , $U_0 = 65 \text{ m/s}$) [15].

A study by Qiu et al. [7] developed a three-dimensional numerical model to simulate ice formation inside an IPS using a thin-film water layer approach. The model incorporates mass, momentum, and energy conservation equations to capture the freezing dynamics under various environmental conditions. The effects of droplet size (MVD), liquid water content (LWC), and freestream velocity were systematically analyzed to determine their influence on both ice thickness and distribution.

The simulations revealed that ice typically accumulates on three critical regions: the inner dome, throat section of the outer dome, and the leading edge of the splitter. While ice thickness on the outer dome and inner dome increases with MVD and velocity due to higher inertia and impingement rates, the splitter shows a non-linear response, with maximum ice occurring at the smallest droplet sizes. Higher LWC values further intensify the accumulation of ice, particularly in areas with strong impingement and lower local heat transfer. The spatial extent and severity of this LWC-induced accretion are visualized in Figure 2.5, which shows the ice growth under varying LWC conditions.

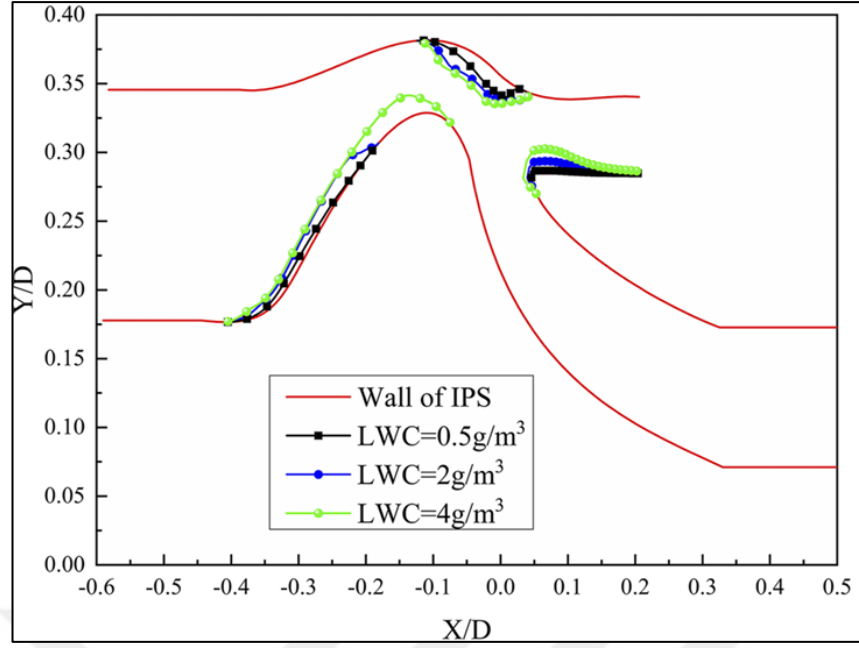


Figure 2.5: Effect of liquid water content (LWC) on ice accretion [7].

In addition to quantifying ice loads, the study also examined the aerodynamic effects of accretion. As shown in Figure 2.6, the presence of ice significantly narrows the flow passage near the throat, leading to a marked acceleration of the core flow and distortion of the internal velocity field. This flow alteration contributes to performance degradation, evidenced by reductions in both the scavenge ratio and total pressure recovery downstream of the icing zone.

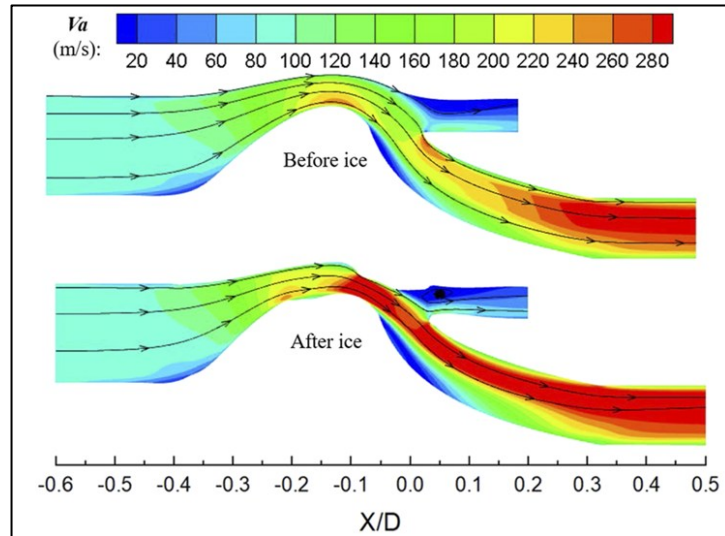


Figure 2.6: Velocity distribution before and after ice accretion [7].

Ferrigno et al. [16] described the icing qualification tests of the NH90 helicopter intake fitted with the T700-T6E1 engine, performed in the CIRA Icing Wind Tunnel. Over sixty runs were carried out across conditions ranging from $-30\text{ }^{\circ}\text{C}$ to $-1\text{ }^{\circ}\text{C}$, altitudes up to 4000 m, and droplet sizes between 20–35 μm .

The campaign employed a full-scale mock-up with protective screens, pressure and temperature probes, and video monitoring to assess intake performance under both continuous and intermittent icing.



Figure 2.7: NH90 intake icing tests (1200 m altitude, 150 km/h, $T=-5^{\circ}\text{C}$, $\text{LWC}=0.7$) [16].

Figure 2.7 clearly illustrate heavy ice accretion on the external screens, particularly at ground-idle and low-speed conditions. These results emphasize that intake geometry and protective meshes significantly influence ice buildup, and that qualification tests are essential to verify compliance with FAR Appendix C and military standards.

In their numerical study, Bu et al. [17] analyzed a thermal anti-icing system for an aircraft wing leading edge using a conjugate heat transfer (CHT) approach coupled with external droplet impingement loads. Their model relied on a similar energy balance methodology to Messinger's [18], incorporating both convective and phase-change heat flux components. The anti-icing strategy was based on supplying hot-air internally to maintain the surface temperature above freezing. Simulation results showed that the required internal heat flux varied significantly along the chordwise direction, with higher demands observed near the stagnation zone due to intense droplet impingement and convective losses. The study emphasized the need for

spatially resolved heating strategies to ensure complete surface protection while avoiding energy oversupply in low-risk regions.

Stiefel [19] reported on the environmental icing tests of the LHTEC T800 turboshaft engine, developed for the U.S. Army's LHX helicopter program. The engine, which also incorporates an IPS, required extensive anti-icing coverage due to its large wetted surface and limited bleed-air budget.

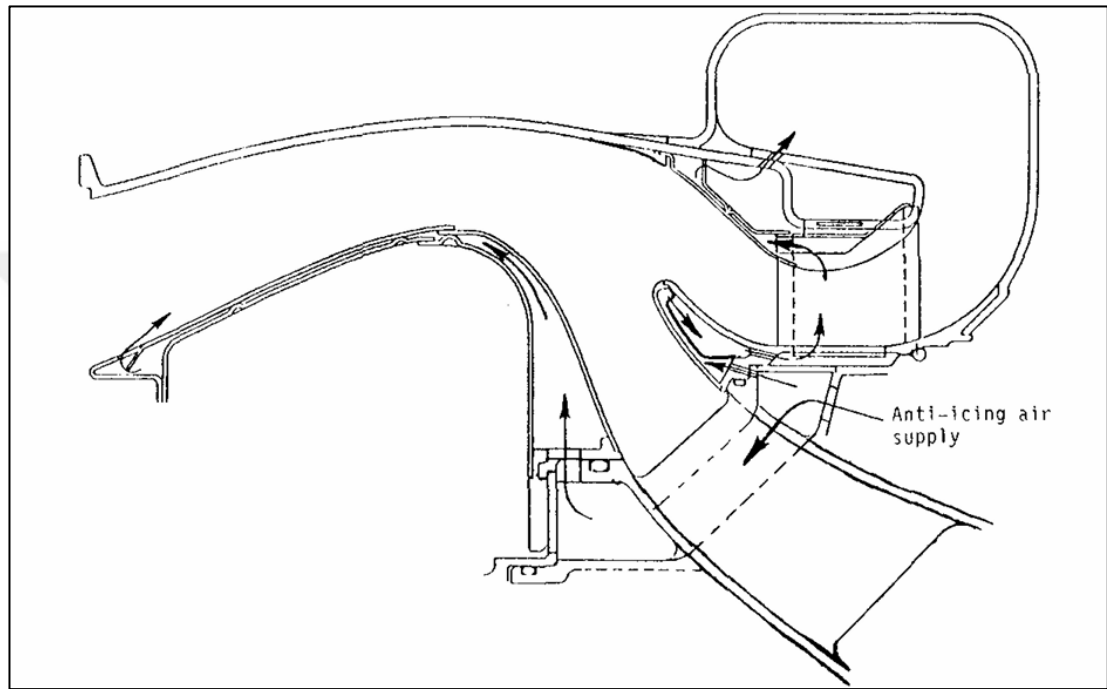


Figure 2.8: Anti-icing air flow of T800 engine [19].

The system relied on compressor bleed air routed through the splitter lip, center body, struts, and scavenge vanes, operating in a icing mode to maintain above freezing surface temperatures. Testing with a dedicated icing simulation facility and borescope visualization allowed direct observation of ice-prone areas and confirmed the need for localized design improvements

This study highlights the trade-off between effective thermal protection and performance penalties in IPS-equipped turboshafts, underlining the importance of early environmental testing to mitigate risks before qualification.

2.4. Thesis Objective

This thesis mainly focuses on the design and evaluation of a hot-air thermal anti-icing concept for the IPS outer dome region. The methodology combines droplet trajectory modelling to quantify external heat loads with iterative adjustments of internal hot-air supply, ensuring that the heated surface temperature remains above freezing at all conditions. The objective is to achieve effective ice prevention while minimizing bleed-air demand and associated efficiency penalties. Validation will be carried out in an icing wind tunnel, allowing direct comparison of predictions with experimental results and critical evaluation of the underlying assumptions.

A schematic of the whole IPS module and outer dome region is given in the Figure 2.9.

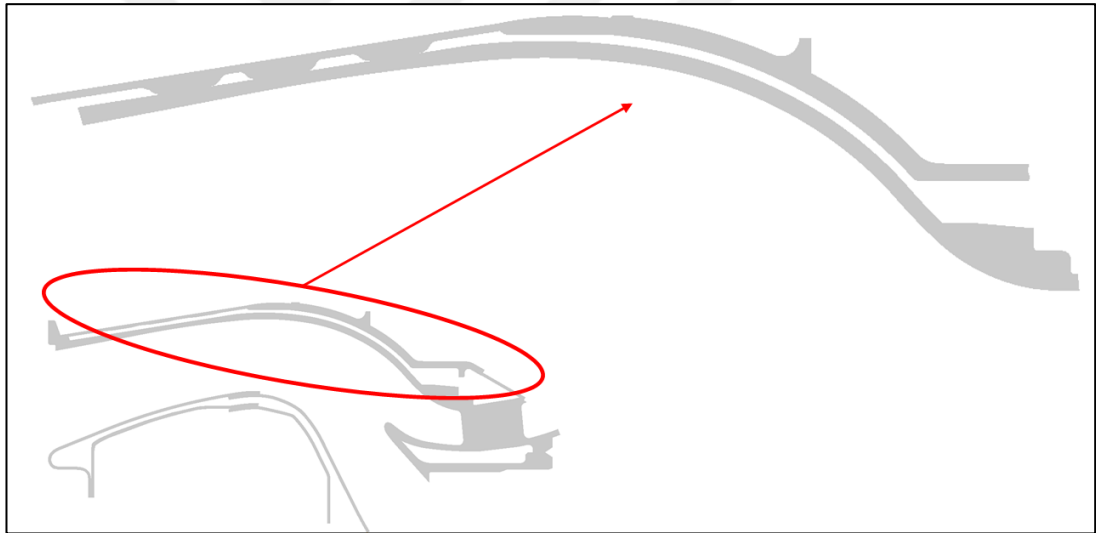


Figure 2.9: Inlet particle separator and outer dome region section.

The IPS outer dome is especially prone to icing due to its curved shape, which is designed to divert sand and dust particles away from the engine core. This same geometry, however, enhances the interception of supercooled water droplets, making the region highly vulnerable to ice accretion. To address this, the structure integrates embedded ducts that channel compressor bleed air toward the most icing-critical surfaces. The corresponding design and test cases for this configuration are summarized in Table 2.1. These cases are derived from the CS-E 780 icing

certification tables and subsequently adapted to wind tunnel similitude conditions, with the detailed scaling procedure presented in the upcoming sections.

Table 2.1: Design cases.

Design Cases	Case-1	Case-2	Case-3	Case-4	Case-5
Ambient Temperature	-10	-20	-20	-20	-20
LWC [g/m ³]	4.3	3.19	3.68	3.86	3.73
Flowpath Re	1.63E+05	1.64E+05	1.79E+05	2.79E+05	2.23E+05
Hot Air Re	2.84E+03	3.38E+03	2.65E+03	2.82E+03	3.06E+03
Non-Dimensional Hot Air Temperature	0.1	0	0.03	0.63	0.3

The test matrix primarily considers ambient temperatures between -10°C and -20°C . Alongside these values, the table lists Reynolds numbers for inner&outer air and a non-dimensional bleed-air temperature parameter, both of which serve to characterize the operating conditions. Cases 1–3 represent lower engine power settings, where the bleed-air temperature is comparatively low and the associated icing risk is therefore greater. In contrast, Cases 4–5 correspond to higher power settings, where bleed-air is substantially hotter and the conditions are less critical with respect to icing.

3. TESTING

Icing Wind Tunnels (IWTs) offer a controlled setting where atmospheric icing conditions can be reproduced with high fidelity [14]. Such facilities make it possible to regulate key parameters—including airspeed, ambient temperature, liquid water content (LWC), and mean volumetric diameter (MVD)—thereby enabling systematic assessment of engine ice protection concepts. Unlike natural flight tests, which are often unpredictable and difficult to replicate, tunnel-based experiments provide repeatable conditions and allow efficient evaluation of different thermal load scenarios and heating strategies [4].

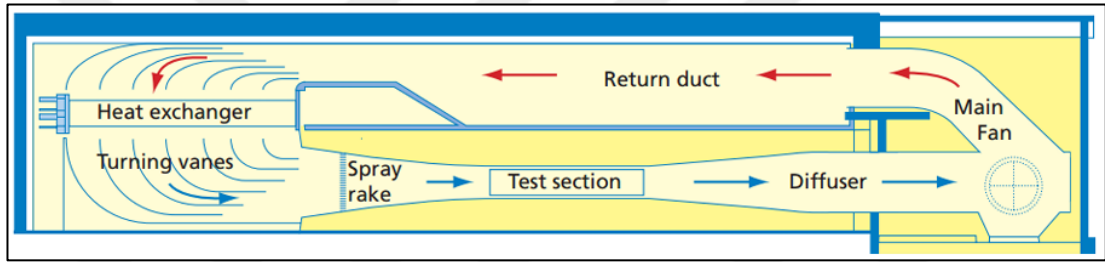


Figure 3.1: Example IWT setup.

An example schematic of an icing wind tunnel is presented in Figure 3.1. In general layout, these facilities are nearly identical to conventional aerodynamic wind tunnels, consisting of a closed-loop flow circuit with test section, diffuser, and drive fan. The main distinction lies in the additional subsystems: spray bars are installed upstream of the test section to atomize supercooled droplets, while heat exchangers are integrated to regulate air temperature to prescribed values.

Test campaigns typically employ either scaled or full-scale models of engine inlets, vanes, or other sensitive components. Instrumentation such as thermocouples, heat flux sensors, and optical access windows are integrated to record surface temperature, accretion shapes, and system performance. The acquired datasets not only support compliance with standards like CS-E 780 but also serve as validation material for numerical simulations.

While icing wind tunnels can reproduce a wide range of atmospheric conditions, limitations still exist. In particular, the facility used in this study operates at sea level

and does not have altitude simulation capability. As a result, direct replication of certification icing envelopes is not feasible. To address this, similitude methods are applied to scale key parameters, ensuring that the test environment remains representative of in-flight conditions.

3.1. Icing Test Similitude Methodology

The wind tunnel setup for which IPS anti-ice design solution will be tested and validated located at the sea level. Therefore LWC and MVD values are scaled using similitude approach.

In icing wind tunnel studies, two parameters are generally regarded as the minimum requirements for similitude: the liquid water content (LWC) and the droplet trajectory scaling. The first governs the mass flux of supercooled liquid reaching the surface, while the second ensures that the impingement pattern is correctly reproduced.

Rate of water catch (used to calculate $\dot{Q}_w$) was given by:

$$\dot{R}_w = \beta LWC \quad (3.1)$$

Where β is the collection efficiency, LWC is the liquid water content, and V is the freestream velocity. To preserve same water catch between scaled (S) and reference conditions,

$$LWC_S V_S = LWC_R V_R \quad (3.2)$$

This equilibrium showcases that scaled LWC values are proportional to air density ratio between reference and scaled values. As a numerical example for a reference altitude of 20,000 ft. at -20°C , the air density yields around 0.63 kg/m^3 . At sea level under the same temperature $\rho_{SL} = 1.39 \text{ kg/m}^3$. Thus,

$$\frac{\rho_{SL}}{\rho_{ALT}} = 2.2 \quad (3.3)$$

Therefore, scaled LWC value becomes 3.74 g/m^3 (reference is 1.7 g/m^3), indicating that the tunnel test must employ a higher liquid water content to compensate for the lower velocity at sea level.

To utilize trajectory scaling, early analyses used a simple droplet inertia parameter K ,

$$K = \frac{\rho_a d^2 U_\infty}{18 d \mu_a} \quad (3.4)$$

Langmuir and Blodgett [20] introduced a modified inertia parameter K_0 , which corrects for non-Stokes drag effects. It is defined as

$$K_0 = \frac{1}{8} + \left(1 - \frac{1}{8}\right) \left(\frac{\lambda}{\lambda_{Stokes}}\right) \quad (3.5)$$

Where the range parameter is,

$$\frac{\lambda}{\lambda_{Stokes}} = \frac{0.1747 R_{e\delta}}{1 + 0.8388 R_{e\delta}^{-0.001483}} \quad (3.6)$$

With droplet Reynolds number,

$$R_{e\delta} = \frac{\rho_a V d}{\mu} \quad (3.7)$$

Trajectory similitude requires that by tuning MVD ,

$$K_{0,S} = K_{0,R} \quad (3.8)$$

Using mentioned MVD similitude approach, nominal 20 μm values scaled up to 30 μm in the test setup and analyses.

3.2. Test Setup of the Current Study

This study concentrates on the thermal validation of anti-icing performance for the IPS outer dome region. Before presenting the thermal analysis inputs and methodology, the testing phases and general considerations are outlined.

The experiments were conducted at the Cranfield University Icing Wind Tunnel (Bedford, UK), a facility that does not provide altitude simulation capability. As a result, similitude-based scaling calculations were performed prior to the tests, and the derived parameters were used in the setup. Three principal challenges must be

addressed in tunnel-based validation: (i) mechanical integration of the test article with the tunnel interface, (ii) calibration of both internal and external flow conditions, and (iii) adjustment of LWC and MVD values through the tuning of spray-bar systems.

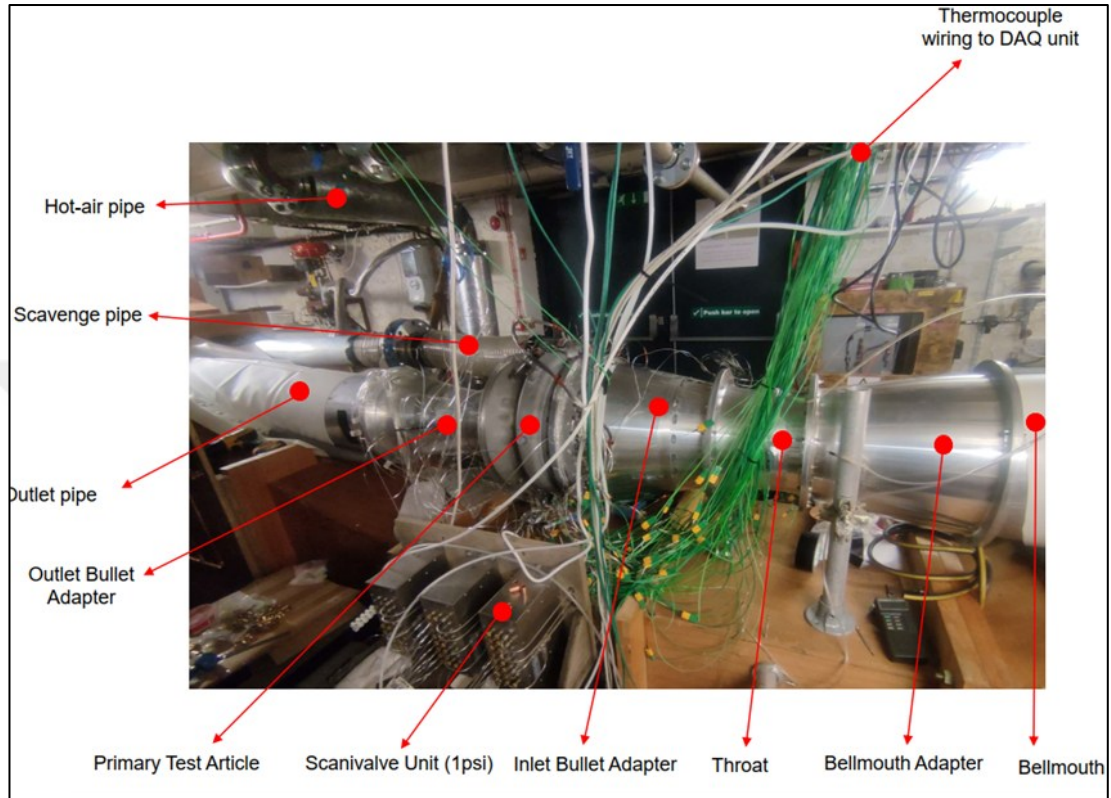


Figure 3.2: Test rig setup for the IPS component.

Figure 3.2 illustrates the interface and assembly of the test article with the tunnel. All piping connections were inspected for leakage prior to testing. Subsequently, thermocouple wiring and static pressure tubes were routed to the data acquisition (DAQ) and scanivalve systems. The hot-air bleed ports were connected to the test rig pressure vessel, where the flow was heated and its mass flow rate and temperature were regulated through a valve–orifice assembly. A schematic of the orifice plate and hot-air bleed measurement system is presented in Figure 3.3.

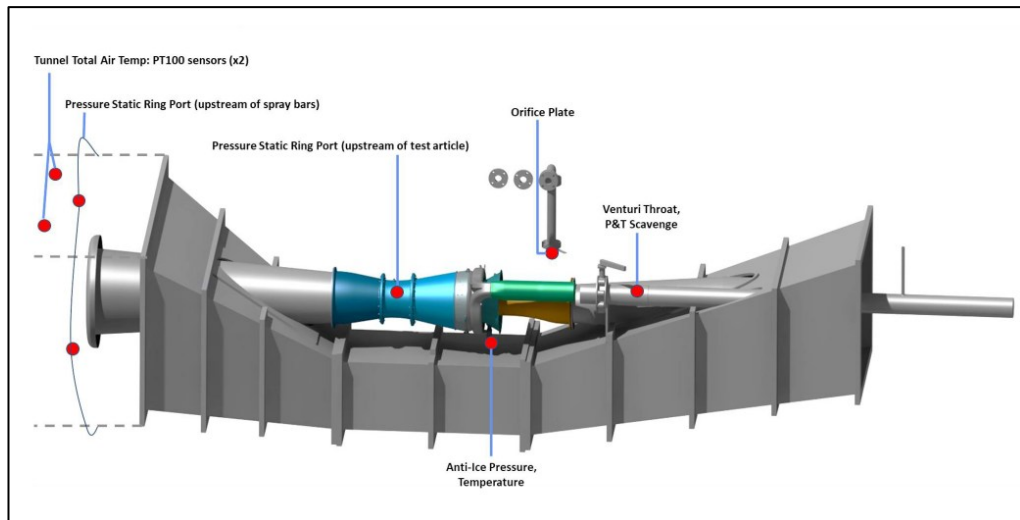


Figure 3.3: Test rig infrastructure and test piece 3d model.

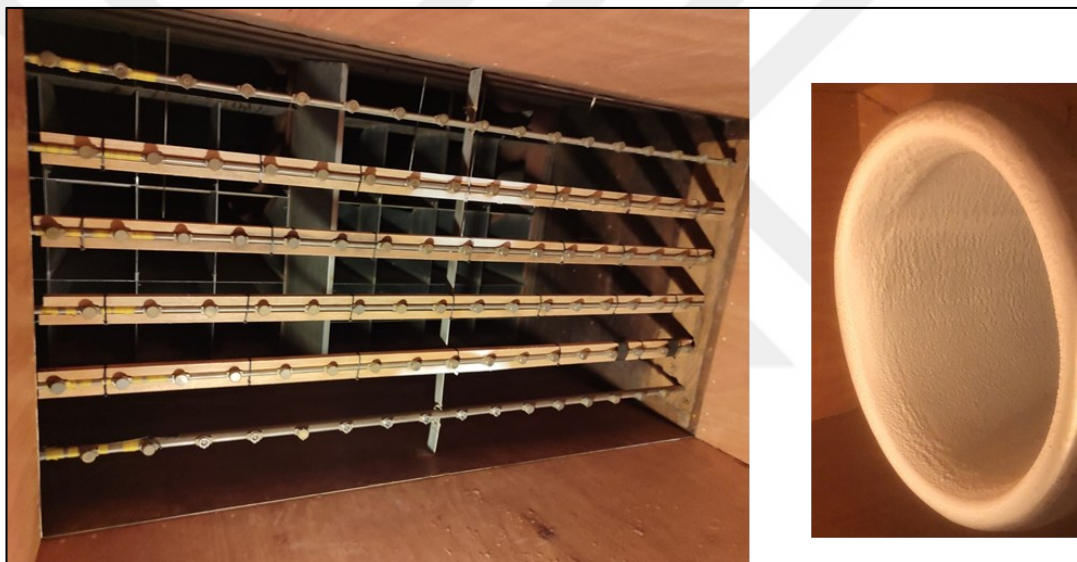


Figure 3.4: Spray bar array and inlet bellmouth.

Spray bar calibration was achieved by adjusting both air and water pressures, as well as the spray bar configuration. Figure 3.4 illustrates the complete spray bar array together with the bellmouth section of the test equipment. An example spray bar configuration for a specific test case is provided in Figure 3.5, where filled circles indicate the open spray bars. The cloud calibration process typically follows an iterative trial-and-error procedure, based on measurements acquired with isokinetic probes.

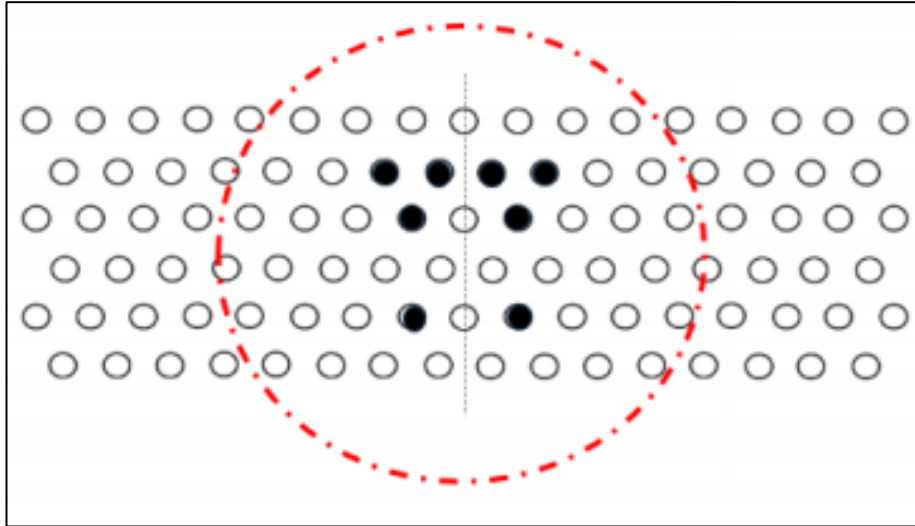


Figure 3.5: Spray bar configuration.

Tunnel speeds were calibrated using a hot-wire anemometer mounted on a traverse mechanism positioned at the throat of the inlet pipes. The measured velocity values were then correlated with the tunnel's static pressure sensors. Once the calibration was completed, the hot-wire anemometer was no longer required, and the tunnel speed was subsequently controlled through the exit fan RPM, as illustrated in Figure 3.6.

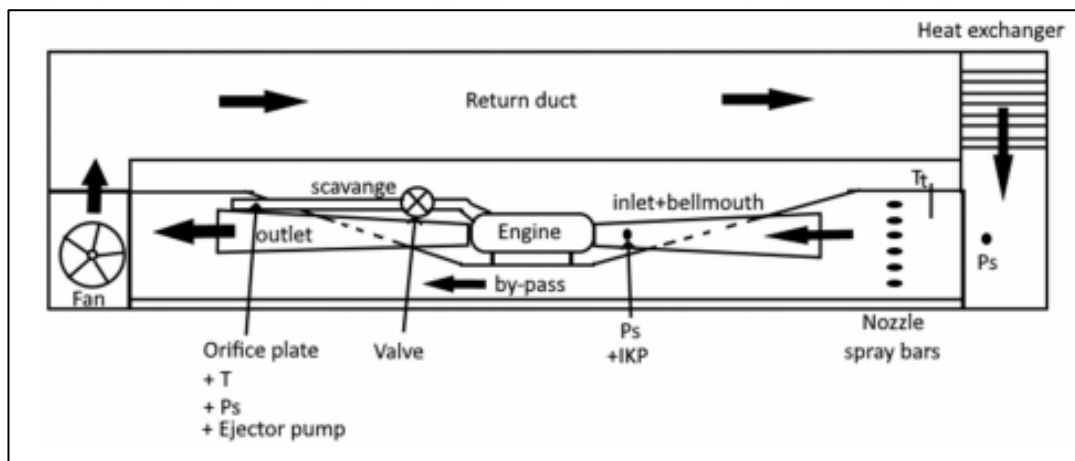


Figure 3.6: Schematic of tunnel calibration sensors.

Prior to delivery to the facility, the test article was fully instrumented. The measurement suite included static pressure taps, air thermocouples located at selected passages and manifolds, and surface (metal) thermocouples placed at icing-prone locations. For the outer dome, thermocouples were arranged as shown in Figure 3.7 with three circumferential positions at each axial station to ensure redundancy.

After global tunnel parameters (e.g., airspeed and LWC/MVD) had been calibrated, hot-air supply calibration tests were performed. The bleed-air circuit was pressurized and pressures in internal passages and manifolds were recorded to verify that flow was routed through the intended ducts with the prescribed split ratios. Mass flow rate and temperature were regulated by the valve–orifice assembly after the pressure vessel, and the resulting settings were used as reference conditions for subsequent icing runs.

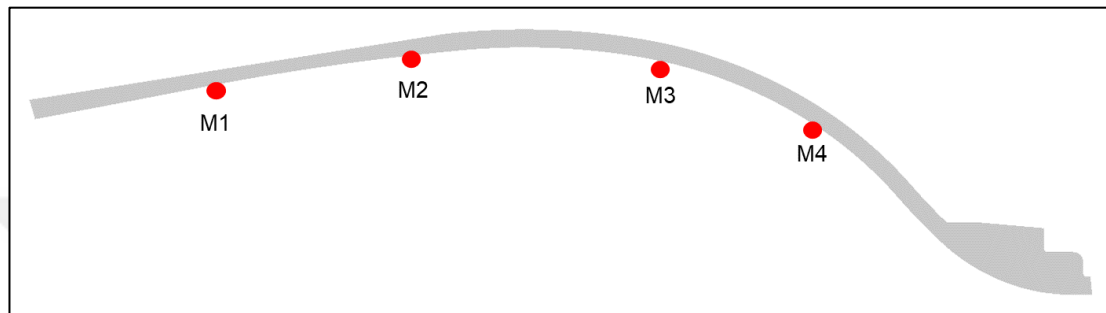


Figure 3.7: IPS outer dome metal thermocouple instrumentation locations.

The standard testing procedure is summarized as follows:

1. The tunnel is started by conditioning both temperature and speed.
2. Once the airflow stabilizes at the desired conditions, the hot-air bleed supply is adjusted to the target values.
3. Metal thermocouples are monitored in real time to confirm that both the test piece and the tunnel have reached steady-state conditions.
4. The spray bars are activated, and thermocouple readings continue to be observed closely.
5. After 3–5 minutes, once all thermocouples confirm steady-state conditions, the spray bars are switched off to avoid excessive icing on tunnel surfaces.
6. Borescope imaging is then initiated to inspect critical surfaces.

If the measured data and borescope observations are consistent, the test campaign proceeds to the next operating condition.

4. ANALYTICAL MODELLING

This section addresses the analytical modelling of thermal–ice heat fluxes and the associated energy balance. Developing such a framework requires a solid understanding of icing physics, fluid dynamics, and thermal system design principles. Within this context, the methodology combines droplet distribution calculations with a thermal analysis setup, both of which are detailed in the following subsections.

4.1. Energy Balance and Messinger Thermal Ice Model

To evaluate the heating requirements for anti-icing systems, it is essential to determine the thermal energy losses caused by impinging supercooled water droplets. One of the most widely accepted analytical tools in the literature for this purpose is the Messinger model [18], which formulates a one-dimensional steady-state energy balance over the control volume of an icing surface.

This approach categorizes surface thermal behavior into three distinct cases depending on the surface temperature: below 0 °C, at 0 °C, and above 0 °C. In the context of anti-icing systems—where the goal is to prevent any ice formation—the relevant case is the above-freezing scenario.

In this regime, the incoming energy components include aerodynamic heating and the kinetic energy of the droplets, while losses occur via convective cooling, droplet warming (sensible heat), and evaporation. These competing terms define the net external heat flux that must be counteracted by internal heating to ensure a fully ice-free surface.

Although Messinger’s original formulation did not account for internal conduction through the substrate or radiative losses, later enhancements have been proposed [21] to improve accuracy under various icing conditions. However, the classical model still provides a practical foundation for estimating heat flux in engineering-level anti-icing design.

This study adopts the above-freezing surface assumption aligned with anti-icing objectives and employs Messinger’s energy balance to calculate external heat flux values under critical icing conditions.

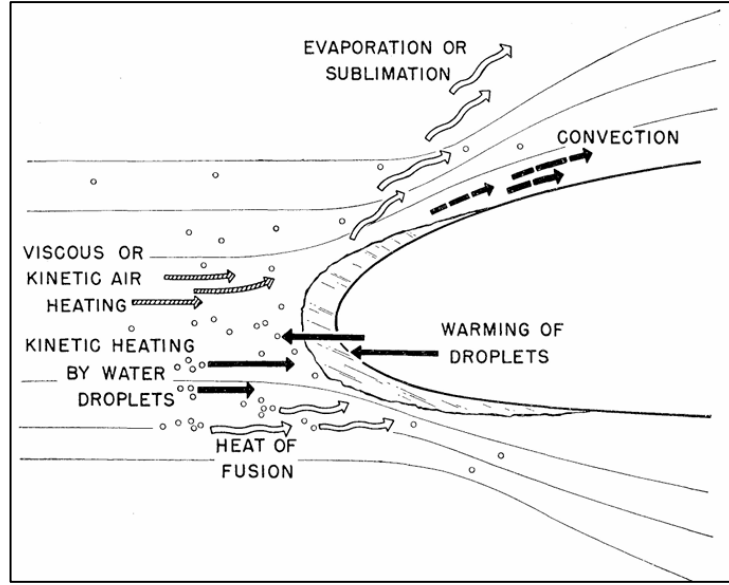


Figure 4.1: Messinger icing model representation [18].

Following the formulation used in Messinger's approach, the external heat flux terms that contribute to surface cooling&heating in an icing environment can be itemized individually. The first term is convective heat loss, representing the energy transferred from the heated surface to the surrounding airstream. This is governed by Newton's law of cooling:

$$\dot{Q}_c = H(T_s - T_\infty) \quad (4.1)$$

here, H is the convective heat transfer coefficient, T_s is the surface temperature, and T_∞ is the free-stream air temperature.

The second term is the energy required to warm incoming supercooled water droplets from the ambient temperature to the surface temperature (enthalpy exchange). This warming process occurs before any phase change and is given by:

$$\dot{Q}_w = \dot{R}_w C_{p_w} (T_s - T_d) \quad (4.2)$$

where $\dot{R}_w$ is the rate of water catch (mass flux of impinging water droplets), C_{p_w} is the specific heat capacity of liquid water, and T_d is the droplet temperature (typically equal to T_∞).

$$\dot{R}_w = \beta LWC V_n \quad (4.3)$$

Here β represents the local droplet collection efficiency, which is the dimensionless ratio of the water mass impinging and adhering to a surface to the theoretical maximum mass of water that could be collected. LWC is the abbreviation for liquid water content, which represents the mass of liquid water droplets per unit volume of air (g/m^3). V_n is the velocity of the droplets.

The third term is latent heat required for evaporation, a surface cooling mechanism that occurs as a portion of the collected water evaporates rather than freezing:

$$\dot{Q}_e = \dot{m}_e Le \quad (4.4)$$

where $\dot{m}_e$ the mass flux of evaporating water, Le is the latent heat of vaporization.

$$\dot{m}_e = \frac{R_{air}}{R_{wa}} \frac{H}{C_{p_{air}}} \left(\frac{P_{v,s} - P_{v,l}}{P_{amb} - P_{v,s}} \right) \quad (4.5)$$

Here vaporization mass flux calculated by using Chilton-Colburn analogy [17] $P_{v,s}$ is partial pressure of water vapor on the surface, $P_{v,l}$ is the partial pressure of water vapour in the bulk air, P_{amb} is the ambient pressure.

$$P_{v,s} = a_0 + a_1T + a_2T^2 + a_3T^3 + a_4T^4 + a_5T^5 + a_6T^6 \quad (4.6)$$

Here saturation pressures can be calculated by Lowe et al.'s approximation [22] using surface metal temperature (assuming metal temperature equals water film temperature) with coefficients given in Table 4.1.

Table 4.1: Polynomial coefficients of saturation pressure.

Coefficient	Value
a0	6984.5053
a1	-188.90393
a2	2.133357675
a3	$-1.288580973 \times 10^{-2}$
a4	$4.393587233 \times 10^{-5}$
a5	$-8.023923082 \times 10^{-8}$
a6	$6.136820929 \times 10^{-11}$

The fourth component is viscous (aerodynamic) heating, which arises due to frictional effects between the high-speed airflow and the surface. This heating is typically small compared to other terms but becomes relevant at especially high velocities. The heat flux generated from this effect can be expressed as:

$$\dot{Q}_a = Hr \left(\frac{V_{inf}^2}{2C_{p_{air}}} \right) \quad (4.7)$$

where r is recovery factor, V_{inf} is the free stream velocity.

The fifth term is the kinetic energy of the droplets, which becomes thermal energy upon impact with the surface. This contribution, although often minor, is included for completeness and is defined by:

$$\dot{Q}_k = \dot{R}_w \left(\frac{V_{inf}^2}{2} \right) \quad (4.8)$$

The heat gain terms defined in Equations (4.7) and (4.8) — namely viscous heating and kinetic energy of droplets — can generally be neglected at airspeeds below Mach 0.3, as their net contribution to the surface heat flux is negligible. An illustrative calculation is presented below using representative parameters in Table 4.2 including collection efficiency (β), liquid water content (LWC), surface and ambient temperatures, and freestream velocity.

Table 4.2: Parameters for example heat flux calculations.

H	525	W/m^2K
β	0.25	-
LWC	2.2	g/m^3
$R_{w,max}$	0.132	$\frac{g}{m^2 \cdot s}$
V_{inf}	60-240	m/s
$P_{v,s}$	748.92	Pa
B	100000	Pa
T_{sur}	276	K
T_{inf}	263	K

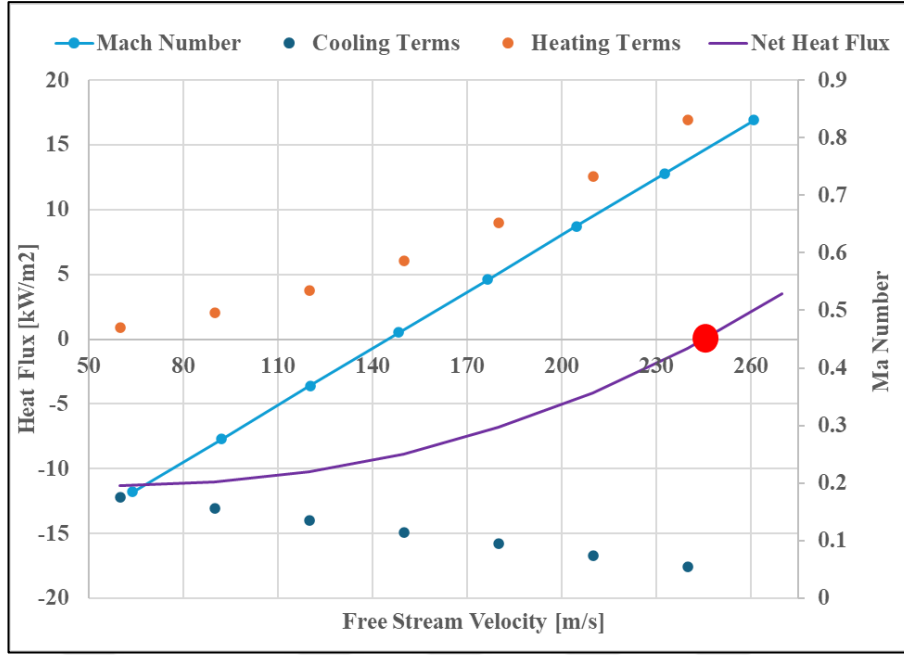


Figure 4.2: Net surface heat load plot for example condition.

By varying the freestream velocity between 60 and 240 m/s, the resulting effect of the heat gain components on the total surface heat flux is visualized in Figure 4.2. As illustrated, at lower velocities, these heat gain terms contribute negligibly to the overall flux. However, since both viscous heating and kinetic energy transfer scale with the square of the freestream velocity, their influence becomes significant at higher speeds. Beyond the 0.74 Ma (approximately 245 m/s in this case), the total heat gain exceeds the combined heat loss, resulting in net surface heating rather than cooling. Consequently, under such conditions, an active anti-icing system may no longer be required.

While this example highlights the behaviour of specific heat flux terms under varying flow conditions, it is important to recognize that—under typical icing scenarios—multiple external thermal loads act simultaneously on the surface. Together, these loads constitute the dominant external thermal inputs in the “above-freezing” anti-icing regime. For accurate prediction of the internal heating requirement, all major heat transfer terms must be carefully calculated under prescribed meteorological boundary conditions. A representative overview of these thermal interactions is illustrated in Figure 4.3. Additional minor losses, such as radiation or conduction through the substrate, are often neglected but may be considered in refined models.

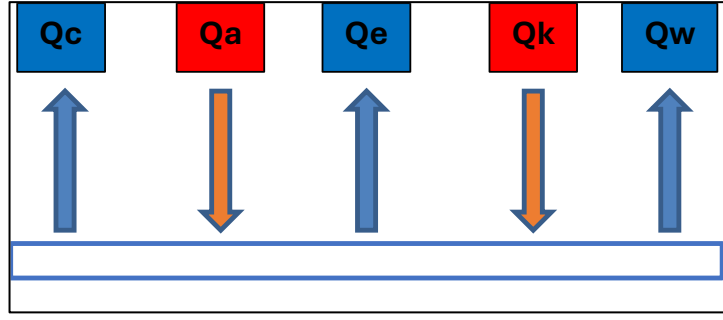


Figure 4.3: Messinger heat flux terms visualized (red indicates heat gain, blue indicates heat loss).

4.2. Droplet Distribution

In order to achieve droplet distribution (important for calculating β), the airflow field is first obtained by solving the governing conservation equations of mass and momentum for the continuous phase using CFD analysis. As these CFD simulations were conducted in a separate external study and subsequently supplied to the present research, their methodology is not elaborated here. The resulting flow solution is then coupled to the droplet transport solver (DROP3D), where the dispersed water phase is treated in an Eulerian sense. This coupling enables the prediction of local impingement fluxes on surfaces without invoking additional energy or film models. [23]

The airflow is described by the continuity and momentum equations:

$$\frac{\partial \rho_a}{\partial t} + \nabla \cdot (\rho_a \vec{V}_a) = 0 \quad (4.9)$$

$$\partial \left(\frac{\rho_a \vec{V}_a}{\partial t} \right) + \nabla \cdot (\rho_a \vec{V}_a \vec{V}_a) = \nabla \cdot \sigma_{ij} \quad (4.10)$$

Where ρ_a is the air density, $\vec{V}_a$ is the velocity vector of the airflow, and $\sigma_{ij} = -\delta^{ij} p_a + \tau_{ij}$ combines the isotropic pressure and the viscous stress tensor. These equations provide the velocity and pressure distribution of the continuous phase, which is subsequently required by the droplet solver.

The dispersed water phase is modelled as a continuous scalar field, governed by the local liquid volume fraction α and the droplet velocity $\vec{V}_d$.

- Conservation of liquid volume fraction:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \vec{V}_d) = 0 \quad (4.11)$$

Here, α is initialized from the freestream liquid water content (LWC) as $\alpha_{inlet} = LWC/\rho_l$, with ρ_l denoting the density of liquid water. This variable describes how much of the control volume is occupied by the dispersed water phase.

- Droplet momentum balance:

$$\frac{\partial (a \vec{V}_d)}{\partial t} + \nabla \cdot [a \vec{V}_d \vec{V}_d] = \frac{C_D Re_d}{24K} a (\vec{V}_a - \vec{V}_d) \quad (4.12)$$

The left-hand side represents the unsteady transport of the droplet velocity field, while the right-hand side models the drag force driving the droplets toward the local air velocity.

- Droplet Reynolds number:

$$Re_d = \frac{\rho_a d |\vec{V}_a - \vec{V}_d|}{\mu_a} \quad (4.13)$$

This dimensionless quantity expresses the ratio of inertial to viscous forces for a droplet moving relative to the surrounding air.

- Drag coefficient:

$$C_d = \frac{24}{Re_d} (1 + 0.15 Re_d^{0.687}), Re_d < 1000 \quad (4.14)$$

This semi-empirical relation accounts for the deviation from Stokes drag at higher Reynolds numbers.

- Inertia parameter:

$$K = \frac{\rho_a d^2 U_\infty}{18 d \mu_a} \quad (4.15)$$

This parameter reflects the ratio of inertial forces (associated with droplet mass and freestream velocity) to viscous forces (associated with air viscosity and reference diameter d). Higher K indicates more inertia-dominated droplets, whereas lower K implies that droplets closely follow the airflow streamlines.

➤ Definition of droplet diameter (MVD)

In icing aerodynamics, the droplet size d is usually defined through the Median Volume Diameter (MVD). This represents the diameter at which half of the total liquid volume is contained in droplets smaller than MVD and the other half in droplets larger. For a simplified approach $d = MVD$ can be used directly. But for more realistic conditions, a size spectrum is considered, typically the Langmuir–D [24] distribution. In this case, the total LWC is divided into several size bins, each with its own fraction of droplets. DROP3D [23] solves Eqs. (4-11) and (4-12) for each size bin and the resulting impingement is summed:

$$\beta = \sum_k \beta_k \quad (4.16)$$

Table 4.3: Langmuir-D distribution for 20 μ MVD [24].

Droplet Diameter (micron)	Weight %
44.4	5
34.8	10
27.4	20
20	30
14.2	20
10.4	10
6.2	5

This distribution allows the model to capture both small droplets that track the airflow closely and larger droplets with significant inertia that impact surfaces more effectively.

➤ Surface impingement and collection efficiency:

The impingement flux at a given surface location is obtained by evaluating the normal projection of the droplet velocity:

$$\beta_i = -a\vec{V}_a \cdot \vec{n} \quad (4.17)$$

Where $\vec{n}$ is the outward-pointing surface normal. This quantity β_i represents the local impingement flux.

Finally, a nondimensional collection efficiency is defined by normalizing with respect to the freestream flux:

$$\beta = \frac{\rho_l \beta_i}{LWC U_\infty} \quad (4.18)$$

where β expresses how effectively the surface captures droplets relative to the available freestream water flux. This term is used to calculate warming of droplets ($\dot{Q}_w$) in the messinger model.

All five design cases were analysed using the same procedure to obtain outer dome surfaces droplet distribution. As illustrated in Figure 4.4, a large portion of the surface area exhibits negligible (≈ 0) collection efficiency, whereas values as high as 0.7 were observed in the curved regions of the outer dome. This outcome is consistent with expectations, given the deflecting geometry of the outer dome flowpath. Thermocouple placements with respect to the collection efficiencies also pointed out (M1,M2,M3,M4) on the Figure 4.4.

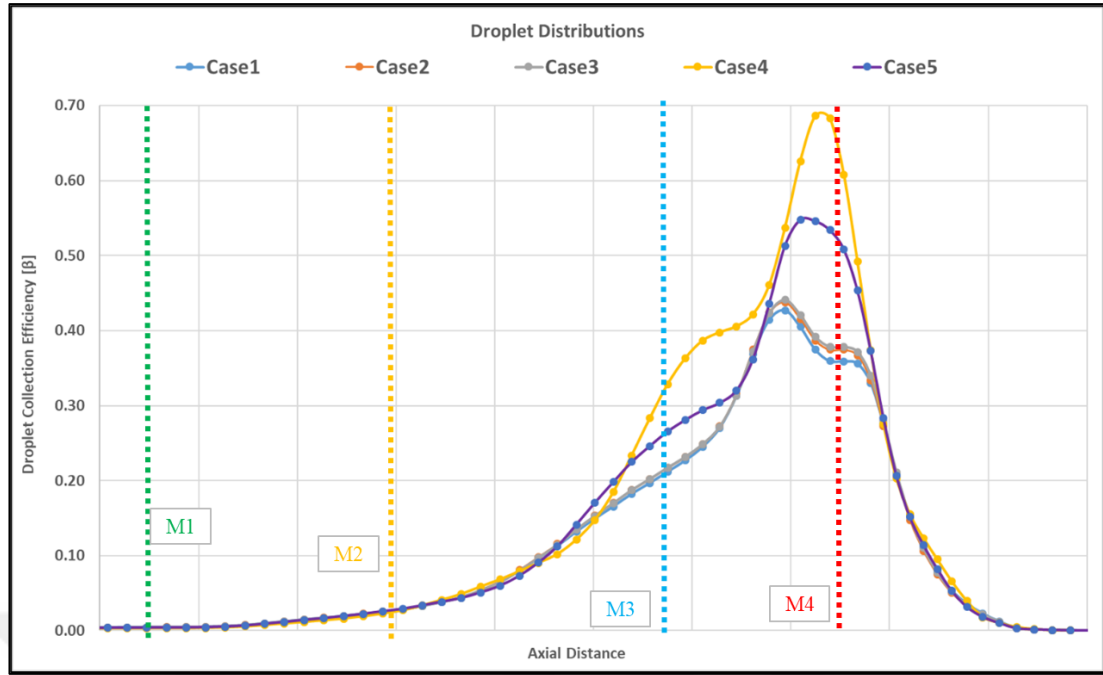


Figure 4.4: Design cases droplet distributions.

Although most of the surface area appears droplet-free, runback from unheated interface pipes may transport water into regions that would otherwise remain unaffected. Additionally, numerical errors or modelling uncertainties could cause surfaces with zero predicted collection efficiency to exhibit non-negligible values in practice. To address these possibilities, all surfaces were treated as ‘wet’ by incorporating evaporation heat loss into the boundary conditions. Furthermore, a conservative approach was adopted by enforcing a minimum collection efficiency of 0.1 for all wet cases, thereby generating parallel solutions that account for potential underprediction. An example of the conservative approach droplet collection distribution for Case-2 is presented in Figure 4.5.

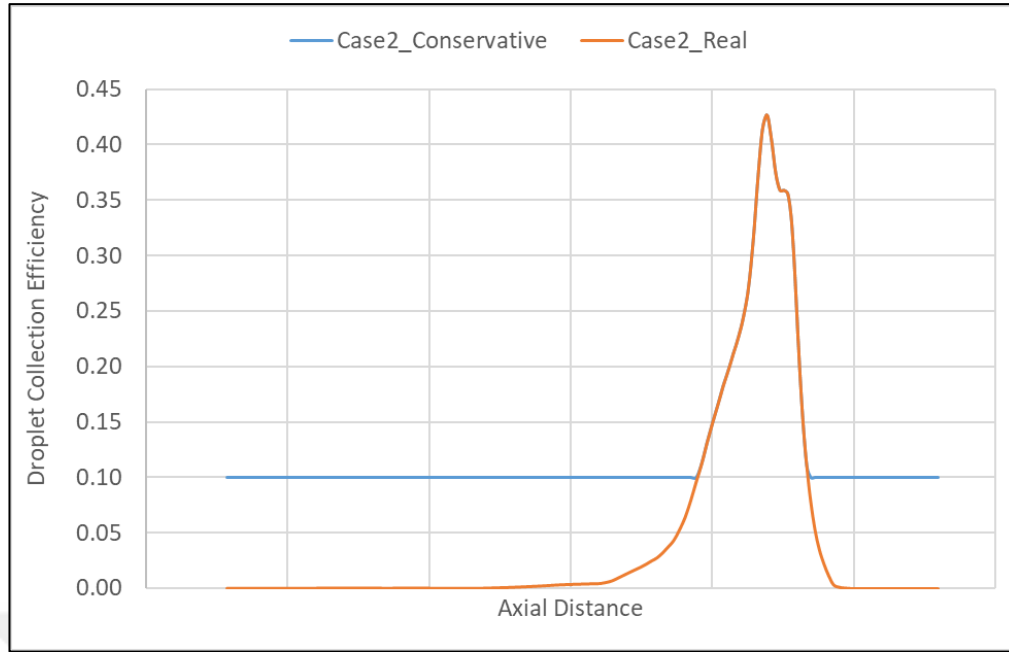


Figure 4.5: Real and conservative droplet distributions.

Collection efficiency values are defined as a function of axial (X) coordinate of the model and used as input for thermal solver.

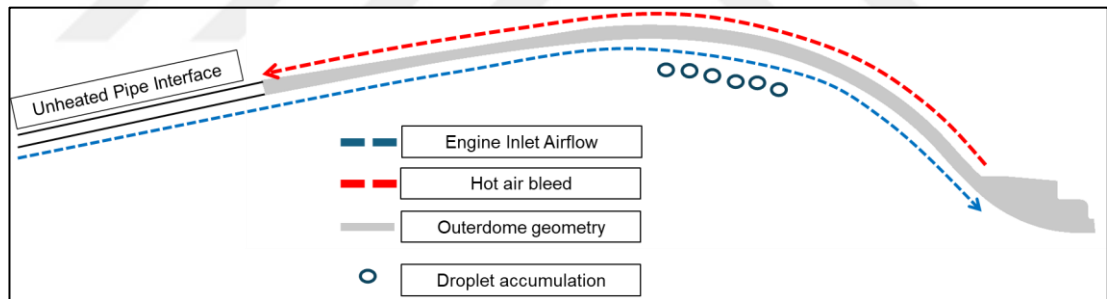


Figure 4.6: Representation of physical problem.

A representative figure is provided in Figure 4.6 to illustrate the interaction of hot bleed air, cold air flow, unheated pipe interfaces, and regions of high droplet accumulation.

4.3. Thermal Analysis Boundary Conditions

Another important parameter is the flow path and hot-air bleed heat transfer coefficients. It is crucial to determine the H values correctly, as they affect three different heat transfer terms (namely, warming of droplets, evaporation, and convective heat transfer).

The H values are calculated using the Nunner [25] Nusselt correlation, as given in equation (4.19).

$$Nu_d = \frac{\lambda_r/8 \ Re_d \ Pr}{(1 + 1.5 \ Pr^{-\frac{1}{6}} \ Re_d^{-\frac{1}{8}} \ (\frac{\lambda_r \ Pr}{\lambda_s} - 1))} \quad (4.19)$$

$$\lambda_s = \left(0.25 \left(\log_{10} \left(\frac{150.39}{Re_d^{0.98865}} - \frac{152.66}{Re_d} \right) \right) \right)^{-2} \quad (4.20)$$

$$\lambda_r = \left(-1.8 \log_{10} \left(\frac{6.9}{Re_d} + \left(\frac{e}{3.7 D} \right)^{1.11} \right) \right)^{-2} \quad (4.21)$$

Here, λ_r is the darcy friction coefficient for rough surfaces, λ_s is the darcy friction coefficient for smooth surfaces. All surfaces are assumed to be smooth, and entry effects for H enhancement are neglected (total pipe length is $> 10D$).

Hot air bleed is routed through narrow ducts to enhance the heat transfer coefficient and thereby reduce the required mass flow rate. At the downstream of the duct, where the bleed-air temperature is lowest, staggered dimple structures are incorporated to locally increase the heat transfer coefficient. Also these dimple formations provide better clearance control during the manufacturing process.

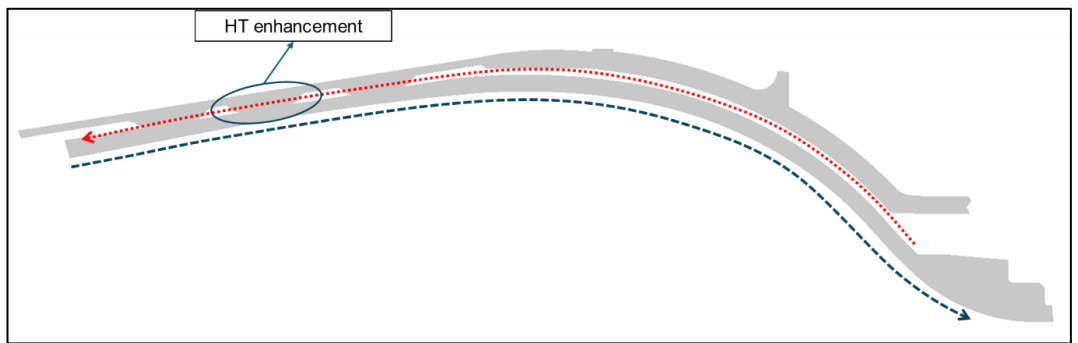


Figure 4.7: Heat transfer enhancement dimples at the hot-air supply route.

The heat transfer enhancement factor for three-row staggered dimple arrangements can be estimated using the correlation proposed by Zukauskas [26]. The expression for this correlation is given in [27]:

$$Nu_D = C_1 Re_{D,max}^m Pr^{0.36} \left(\frac{Pr}{Pr_s} \right)^{\frac{1}{4}} \quad (4.22)$$

$$Nu_D \text{ valid for } = \begin{cases} N_L \geq 20 \\ 0.7 \leq Pr \leq 500 \\ 10 \leq Re_{D,max} \leq 2 \times 10^6 \end{cases} \quad (4.23)$$

Where N_L is the number of tube rows, C_1 is correction factor for longitudinal and transverse pitch ratio.

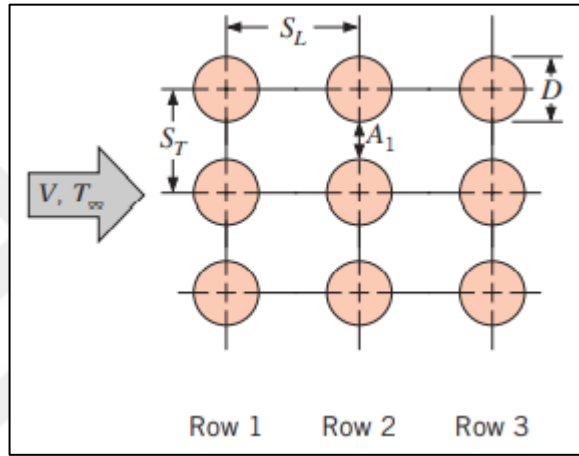


Figure 4.8: Defining S_T and S_L [27].

$$Re_{d,max} = \frac{\rho V_{max} D}{\mu} \quad (4.24)$$

Where V_{max} is given as,

$$V_{max} = \frac{S_T}{S_T - D} \quad (4.25)$$

Finally multiplying heat transfer coefficient with the C_2 correction for 3 row of dimples as in:

Table 4.4: Heat transfer correction regarding the dimple row number [27].

Number of Rows	1	2	3	4
Correction for Aligned	0.7	0.8	0.86	0.9
Correction for Staggered	0.64	0.76	0.84	0.89

The resulting heat transfer enhancement corresponds to a factor of 1.8 for the current design, and this multiplier was applied to the hot-air bleed boundary condition in the thermal solver. For comparison, one of the cases (Case-3) was also solved using a non-dimpled geometry. The results are presented in Figure 4.9 as a contour plot illustrating the difference between the dimpled and non-dimpled configurations.

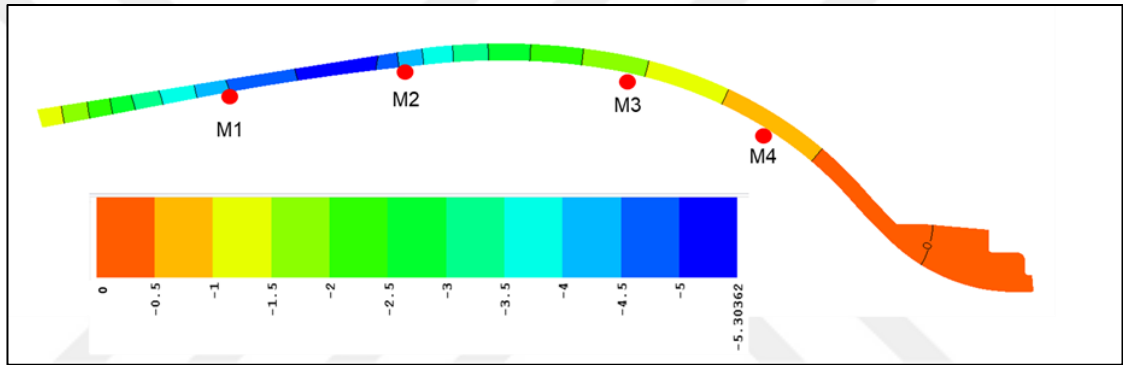


Figure 4.9: Case-3 dimpled vs non-dimpled geometry temperature difference contour.

From this comparison, it can be concluded that the three-row dimple arrangement enhanced surface heating by up to 5.4 °C relative to the non-dimpled design.

At this stage, all necessary convective boundary conditions—including the Messinger heat flux equations as well as external and internal flow effects—were implemented in the thermal model. To represent conduction, a solid domain was introduced within the two-dimensional (2D) thermal solver and discretized using triangular finite elements.

The triangular solid-domain mesh enabled the application of surface boundary conditions at each node and mid-point, while also accounting for conduction heat transfer. The outer dome material was specified as A357 aluminium alloy, with thermal conductivity values averaged at approximately 160 W/m·K and defined as a function of temperature.

Although the thermal solver was configured for steady-state calculations, a temperature refinement criterion of 1 K was applied, meaning that iterations continued until the maximum temperature error at any location dropped below this threshold. To verify mesh independence, the mesh density was doubled for one representative case. At a location of high droplet collection efficiency (Figure 4.10), the resulting metal temperature varied by less than 0.1 K between the coarse and refined meshes, confirming the robustness of the solution.

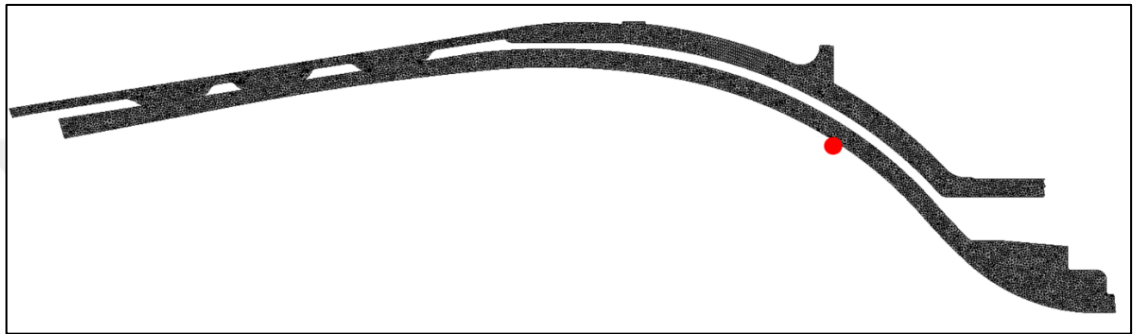


Figure 4.10: Solid domain mesh structure from 2d thermal solver.

5. RESULTS

A flowchart of the model inputs is presented in Figure 5.1. Each case was solved under both dry and wet conditions to evaluate the influence of water spray.

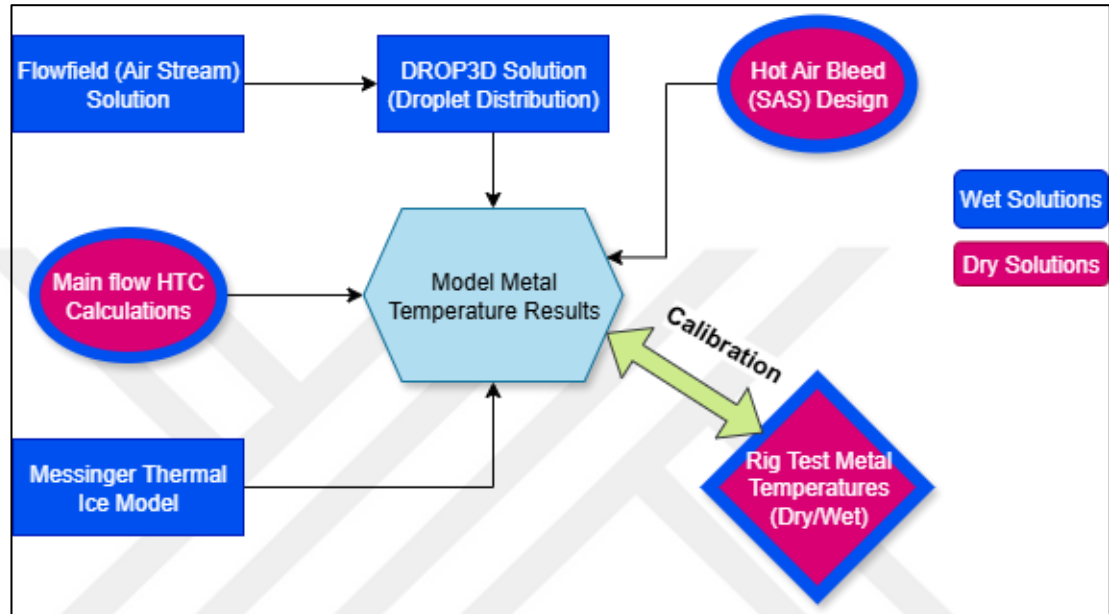


Figure 5.1: Thermal modelling and calibration flowchart.

In a typical dry solution, defining the inner and outer air temperatures together with the heat transfer coefficients are sufficient, effectively reducing the problem to that of a heated or cooled plate. Performing all cases under dry conditions first provides a clearer estimation of the convective heat transfer before accounting for the cooling effects of the spray. This approach allows the models to be calibrated in the dry state; then, if discrepancies arise in metal temperature measurements under wet conditions, they can be more readily attributed to uncertainties in water-effect predictions.

All five design cases were analysed in the thermal solver using the aforementioned boundary conditions under steady-state assumptions. An example metal temperature distribution for Case-1, comparing dry and wet solutions, is shown in Figure 5.2. Note that the minimum Beta distribution value was set to 0.1

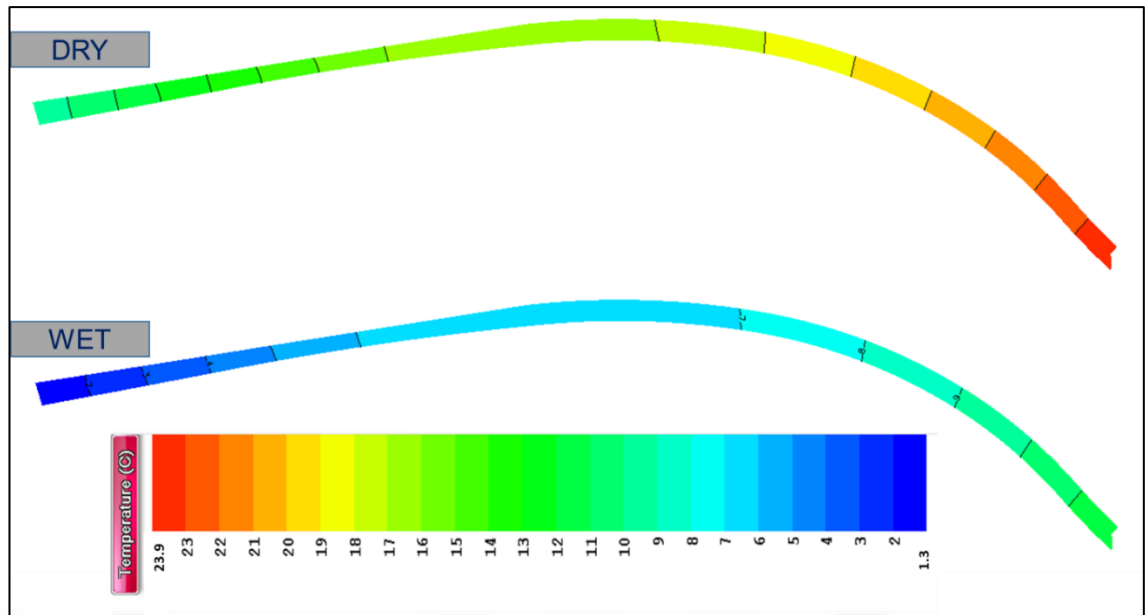


Figure 5.2: Case-1 metal temperature contour for dry and wet solutions.

In Case-1, the contours indicate that the combined effect of incoming droplets and evaporation reduced the metal surface temperature by approximately 10 °C.

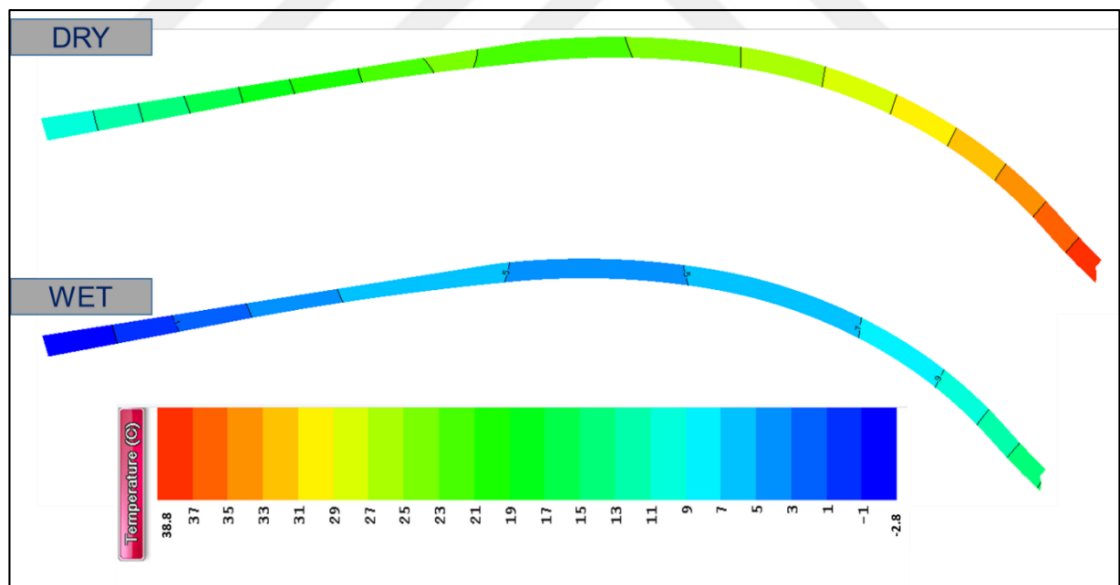


Figure 5.3: Case-4 metal temperature contour for dry and wet solutions.

For Case-4, the dry/wet comparison shows that the cooling effect was nearly doubled relative to Case-1. This difference is attributed to variations in the surrounding

physical environment. To clarify these effects, the calculated heat flux distributions are presented in Figure 5.4.

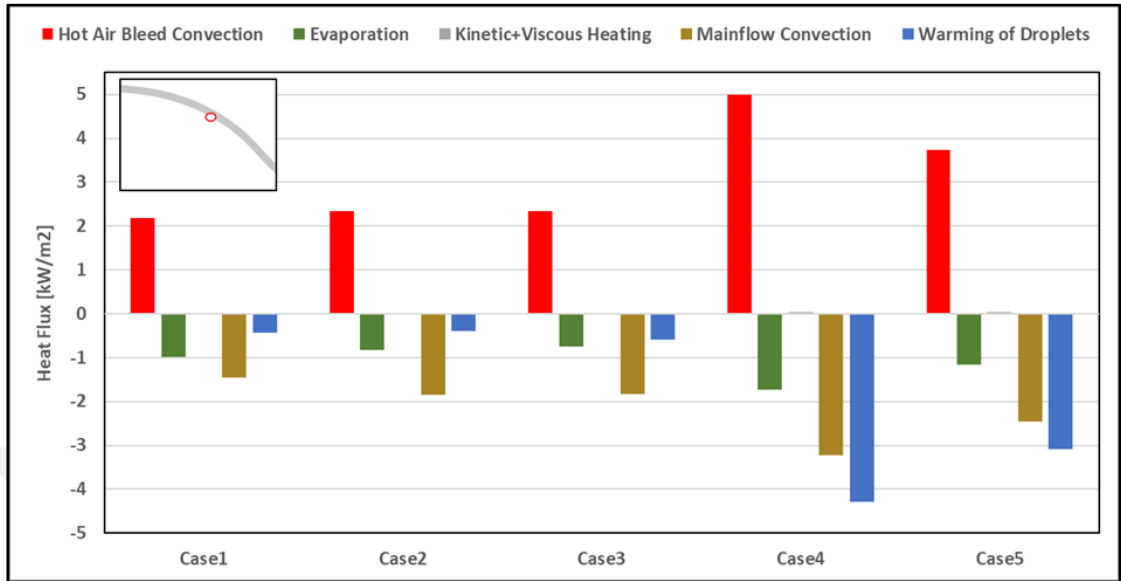


Figure 5.4: Heat flux terms on all the design cases.

The heat flux values listed in the table were calculated at the location of maximum collection efficiency, indicated in the image at the upper-left corner of the table. At lower engine power levels, the surface cooling is primarily driven by convection from the main flow and by evaporation, while the contribution of droplet warming is negligible. This is attributed to the low velocity profile in these cases, which results in small values of the $\dot{R}_w$ term.

In contrast, under high-power conditions (Cases-4&5), droplet warming becomes the dominant contributor to surface cooling. This is due to both the substantially higher velocity and elevated β values, which together make the $\dot{R}_w$ term significant. Another notable observation is the increase in evaporation rates. Since evaporation is governed by the pressure difference between the water film and the freestream saturation pressure, a larger temperature difference between the water film (equal to the metal surface temperature) and the external flow enhances evaporation. As the cooling contributions increase, the internal heat fluxes must correspondingly rise to counteract their effect.

5.1. Validation

To validate the thermal analysis results, icing rig tests were conducted under both dry and wet design conditions. In addition, two supplementary cases were introduced. The first modification was based on Case-1, where the nominal conditions corresponded to an LWC of 4.3 g/m^3 and a hot-air bleed Reynolds number of 2.84×10^3 . For this supplementary case, the LWC was reduced to 3 g/m^3 while the hot-air bleed Reynolds number was increased to 5.18×10^3 . This adjustment created an alternative scenario applicable to both Case-1 and Case-2, providing further insight into the sensitivity of the system. The complete set of executed test cases is summarized in Table 5.1.

Table 5.1: Design cases to test with additional test points.

Design Cases with Additional Test Points	Case-1a	Case-1b	Case-2a	Case-2b	Case-3	Case-4	Case-5
Ambient Temperature [°C]	-10	-10	-20	-20	-20	-20	-20
LWC [g/m ³]	3	4.3	3.19	3.19	3.68	3.86	3.73
Flowpath Re	1.7E+05	1.6E+05	1.6E+05	1.6E+05	1.8E+05	2.8E+05	2.2E+05
Hot Air Re	5.2E+03	2.8E+03	5.1E+03	3.4E+03	2.7E+03	2.8E+03	3.1E+03
Non-Dimensional Hot Air Temperature	0.32	0.1	0.18	0	0.03	0.63	0.3

The experimental results were post-processed and compared with the thermal model predictions. The differences between test and model metal temperatures were used to calculate the root-mean-square (RMS) error. Two different modelling approaches were considered in this comparison: the first corresponds to a conservative method where the minimum collection efficiency was fixed at 0.1, while the second uses the actual collection efficiency distribution. The results are summarized in Table 5.2.

Table 5.2: RMS errors (test-model) for test cases 1-2-3.

Test-Model RMS Error	Ambient Temperature - 10°C LWC = 4.3 g/m ³				Ambient Temperature -20°C LWC = 3.19 g/m ³			Ambient Temperature -20°C LWC = 3.68 g/m ³	
%LWC	0	70	0	100	0	100	100	0	100
MVD [micron]	30				30			30	
Case Detail	1-a (dry)	1-a (wet)	1-b (dry)	1-b (wet)	2-a (dry)	2-a (wet)	2-b (wet)	3-a (dry)	3-a (wet)
Test- Conservative Model (min. β is 0.1)	3.21	<u>3.98</u>	2.15	<u>2.6</u>	3.9	<u>3.04</u>	<u>2.06</u>	2.03	2.81
Test-Real Model		4.03		2.79		3.66	2.68		<u>2.53</u>

For the wet cases (specifically Cases 1–3), the conservative approach (min. collection efficiency is set to 0.1) RMS error values (underlined) are lower, typically ranging between 2–4 °C. In most instances, these values are smaller than those obtained from the real collection efficiency distributions. Consequently, subsequent comparisons focus on the conservative model predictions (with the minimum collection efficiency set to 0.1) against the experimental results.

The table also indicates that the RMS error values of the dry and wet cases are nearly identical. This consistency demonstrates that the application of the Messinger model, as well as the overall treatment of droplet thermal effects, is in good agreement between the model and the tests.

The figures below illustrate the differences between measured thermocouple data and model predictions for both dry and wet cases. On the X-axis, the positions 1–4 correspond to the axial thermocouple locations (M1–M2-M3-M4) on the outer dome surface, as previously described. Filled circles denote the test measurements, while hollow circles represent the model predictions for dry and wet conditions.

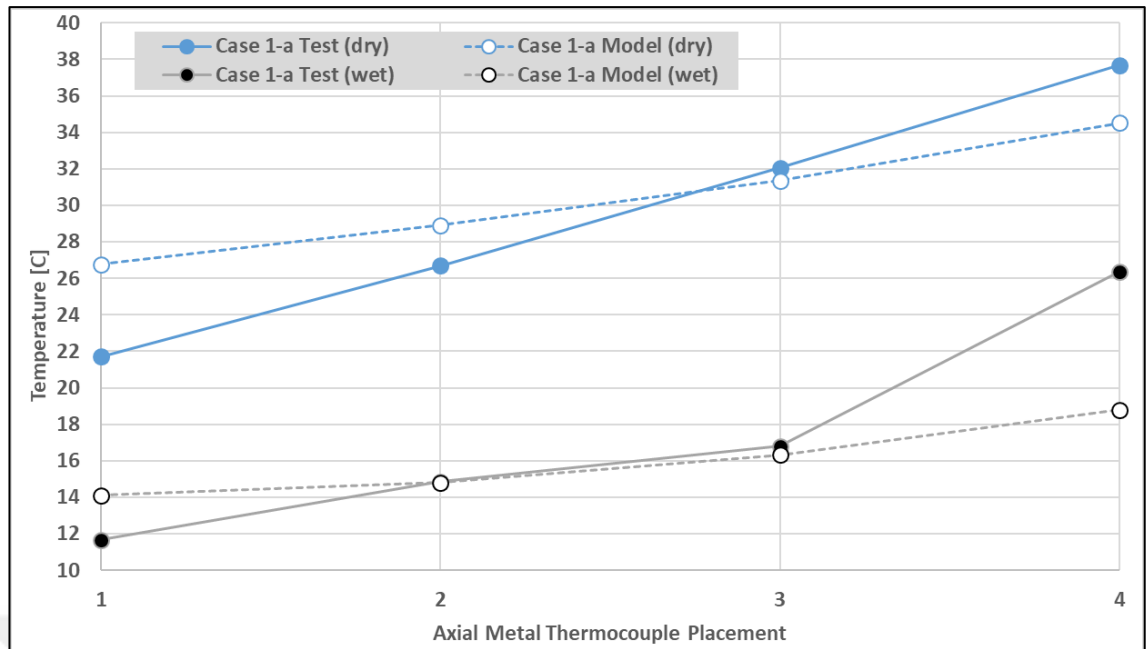


Figure 5.5: Case 1-a test metal thermocouple measurements and model temperatures.

Table 5.3: Case 1-a cooling values on the metal thermocouple locations for test and model.

Dry-Wet Cooling [°C]	Test	Model	Difference
M1	10.1	12.7	-2.6
M2	11.8	14.1	-2.3
M3	15.2	15	0.2
M4	11.3	15.7	-4.4

Case 1-a represents an additional test condition in which the LWC was reduced and the hot-air bleed mass flow increased. A dry-condition comparison between the test and the model revealed a maximum difference of 5 °C at the M1 thermocouple location. The test indicated a lower dry temperature, suggesting either a higher mainflow heat transfer coefficient or a lower hot-air-side coefficient than assumed in the model.

For the wet condition, the maximum discrepancy in surface cooling between the test and the model was 4.4 °C. The first three thermocouple readings were nearly identical, showing a consistent offset across both test and model results. However, at the final thermocouple position, the test recorded an 11.3 °C cooling, whereas the model predicted 15.7 °C. This discrepancy may be attributed to an overestimation of droplet effects at this specific location.

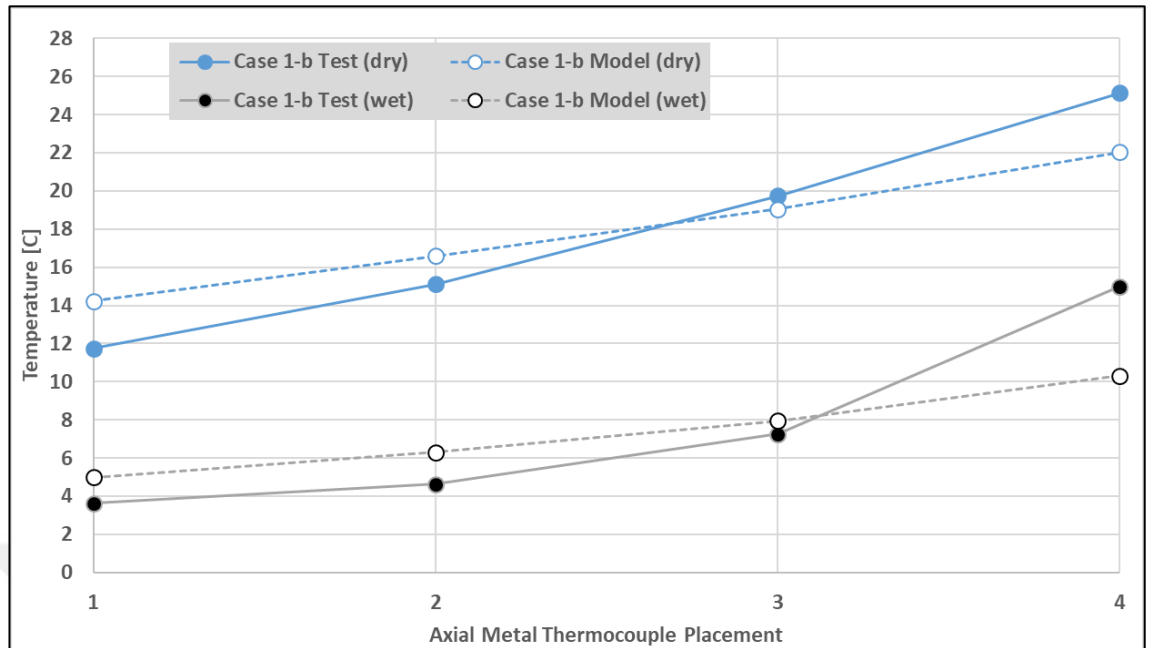


Figure 5.6: Case 1-b test metal thermocouple measurements and model temperatures.

Table 5.4: Case 1-b cooling values on the metal thermocouple locations for test and model.

Dry-Wet Cooling [°C]	Test	Model	Difference
M1	8.1	9.2	-1.1
M2	10.5	10.3	0.2
M3	12.5	11.1	1.4
M4	10.1	11.7	-1.6

Case 1-b represents the actual design condition (corresponding to Case-1 in the Thesis Objective section), characterized by a higher LWC and a lower hot-air mass flow compared to Case 1-a. The dry-condition temperatures were nearly identical between test and model, with a maximum deviation of 3 °C at the M4 location. The wet-condition results and cooling values also showed close agreement, indicating that, for this case, the water effects were realistically captured in the model.

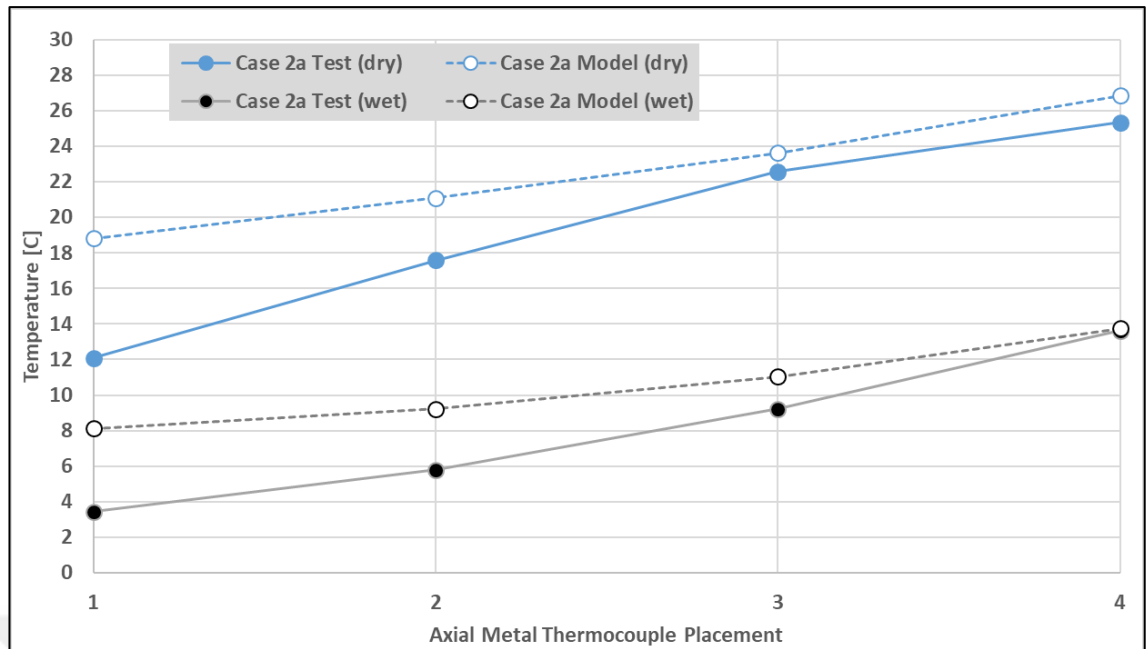


Figure 5.7: Case 2-a test metal thermocouple measurements and model temperatures.

Table 5.5: Case 2-a cooling values on the metal thermocouple locations for test and model.

Dry-Wet Cooling [°C]	Test	Model	Difference
M1	8.6	10.7	-2.1
M2	11.8	11.9	-0.1
M3	13.4	12.6	0.8
M4	11.7	13.1	-1.4

Similar to Case 1-a, Case 2-a represents an additional point where the hot-air mass flow was increased relative to the original case. A maximum dry-condition deviation of 6 °C was observed at the M1 location. This difference possibly due to models underpredicted convection heat loss on the surface. All other thermocouple positions showed closer agreement between test and model results. The cooling differences between dry and wet conditions were limited to a maximum of 2 °C, again demonstrating that the water modelling was realistically represented in the thermal analysis. The largest discrepancy in the wet-condition results can be traced back to the mismatch at M1 in the dry comparison (dry model shows higher surface temperature and therefore it could lead to a bigger temperature drop in the wet condition).

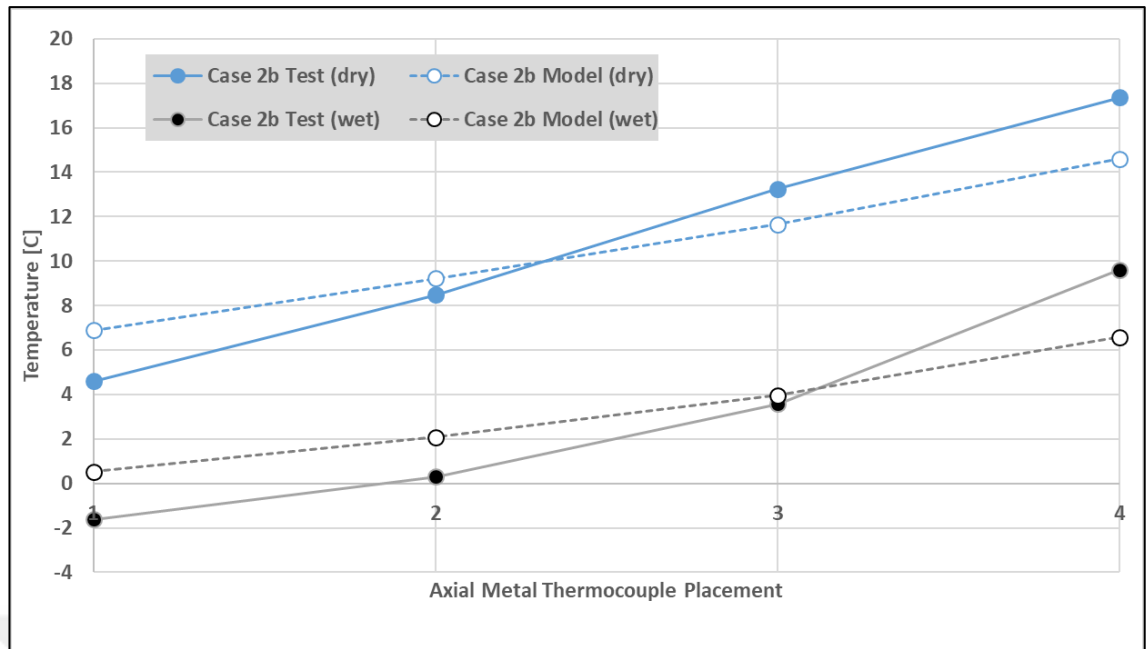


Figure 5.8: Case 2-b test metal thermocouple measurements and model temperatures.

Table 5.6: Case 2-b cooling values on the metal thermocouple locations for test and model.

Dry-Wet Cooling [°C]	Test	Model	Difference
M1	6.2	6.4	-0.2
M2	8.2	7.1	1
M3	9.7	7.7	2
M4	7.8	8	-0.3

Case 2-b represents the original design case (corresponding to Case-2 in the Thesis Objective section) and constitutes another critical condition for icing risk, primarily due to the low ambient temperature and dominant external convection. The dry-condition results from test and model were in close agreement, with a maximum deviation of 2 °C. The wet-condition results followed a similar trend, with dry–wet cooling differences between test and model also limited to 2 °C. These findings confirm that both the inner and outer heat transfer mechanisms under dry conditions, as well as the water effects under wet conditions, were realistically represented in the thermal model.

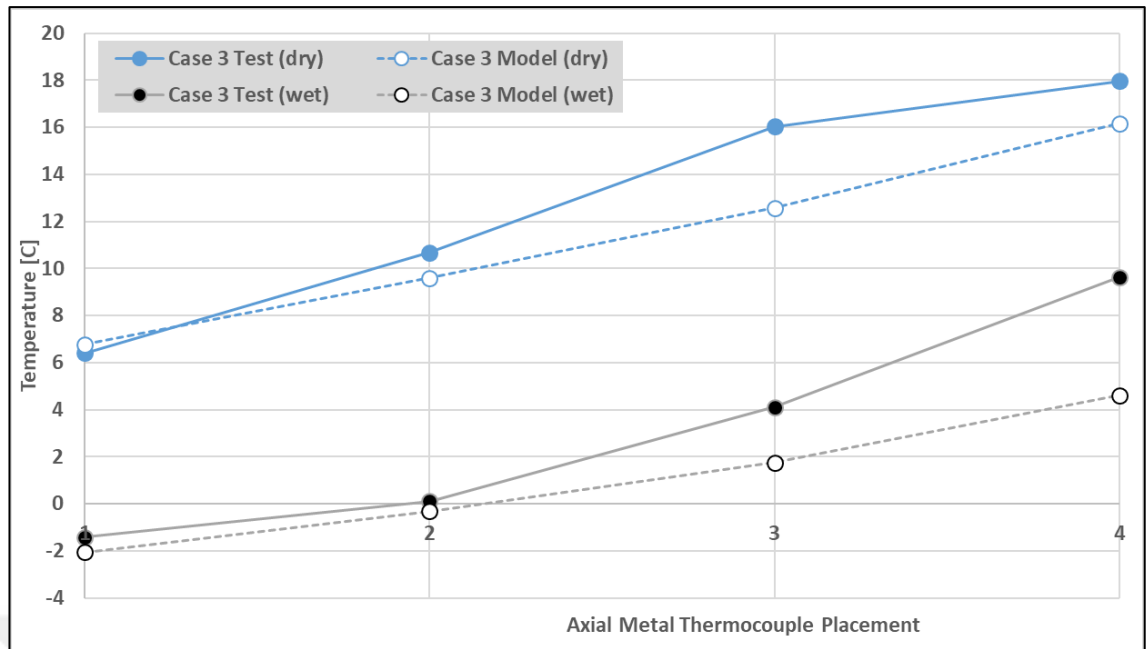


Figure 5.9: Case-3 test metal thermocouple measurements and model temperatures.

Table 5.7: Case-3 cooling values on the metal thermocouple locations for test and model.

Dry-Wet Cooling [°C]	Test	Model	Difference
M1	7.8	8.8	-1
M2	10.6	9.9	0.7
M3	11.9	10.8	1.1
M4	8.3	11.6	-3.3

Case-3 represents another critical design case in which the dry-condition results from test and model were in close agreement. The maximum dry-condition deviation was 3.5 °C at the M3 location. This difference is especially abrupt as normally dry conditions differ at M1 location. Under wet conditions, the largest cooling difference was 3.3 °C at M4, while all other locations showed excellent agreement between test and model predictions. At M4, the model predicted a stronger influence of water effects on the surface compared to the test. This discrepancy is likely caused by an overprediction of collection efficiency in this region, which is highly dominated by droplet impingement.

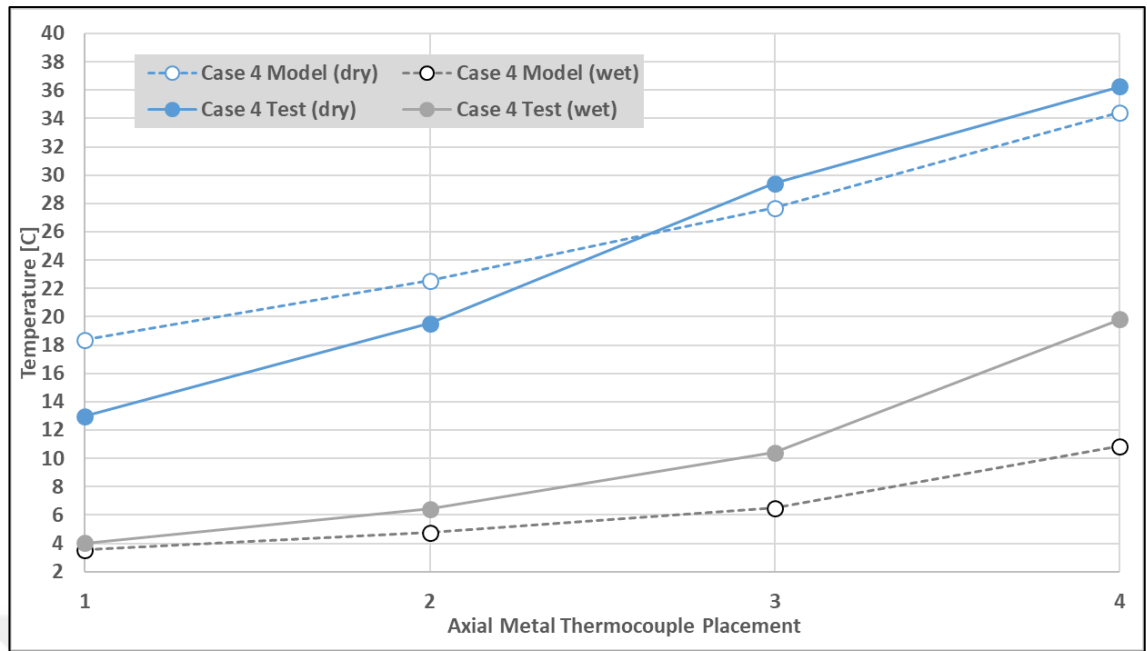


Figure 5.10: Case-4 test metal thermocouple measurements and model temperatures.

Table 5.8: Case-4 cooling values on the metal thermocouple locations for test and model.

Dry-Wet Cooling [°C]	Test	Model	Difference
M1	9	14.8	-5.9
M2	13.1	17.8	-4.7
M3	19	21.2	-2.2
M4	16.4	23.6	-7.1

Case-4 is one of the two high-power conditions, where the icing risk is generally lower and considered less critical. The dry-condition results showed close agreement between test and model, except at the M1 location, where the difference was 5 °C. However, the wet-condition results exhibited larger discrepancies, ranging between 2 °C and 7 °C. At all thermocouple locations, the model consistently predicted greater temperature drops due to water effects, with the largest difference observed at M4, the region of highest collection efficiency. These findings indicate that the droplet collection efficiency was likely overpredicted in the numerical analysis for this case.

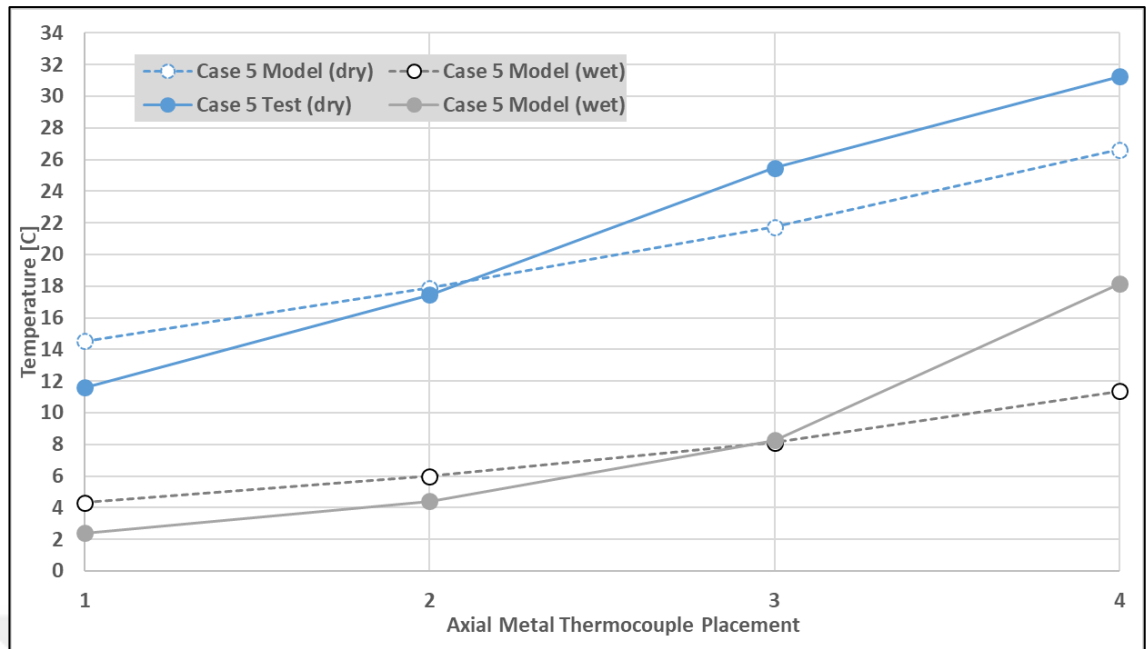


Figure 5.11: Case-5 test metal thermocouple measurements and model temperatures.

Table 5.9: Case-5 cooling values on the metal thermocouple locations for test and model.

Dry-Wet Cooling [°C]	Test	Model	Difference
M1	9.2	10.2	-1
M2	13.1	11.9	1.2
M3	17.2	13.6	3.6
M4	13.1	15.3	-2.2

Case-5 represents the final design case, with heating and cooling values and mass flows similar to those of Case-4, as both correspond to high-power conditions. Under dry conditions, the maximum test-model deviation was 4 °C at the M4 location. The wet-condition results, however, exhibited closer agreement than in Case 4, with the largest discrepancy of 3.6 °C occurring at M3. At M4, the model predicted greater cooling than the test, whereas at M3 the opposite trend was observed. This suggests that M3 may have experienced higher droplet collection, while M4 collected fewer droplets, despite the modest temperature differences.

Throughout all test executions, borescope images were also taken to visually assess icing risk. No ice accretion was observed in any of the design cases, and all surfaces remained clean.



Figure 5.12: Borescope image from Case-3.

Figure 5.12 presents a borescope image taken at the interface between the unheated pipe and the heated test section for Case-3. The image shows that rime ice formed on the unheated pipes, but this accretion rapidly dissipated upon reaching the heated IPS regions.

6. CONCLUSIONS AND FURTHER STUDIES

In this study, an anti-icing system was designed for the outer dome region of an integrated inlet particle separator. The design methodology and modelling strategies were introduced with the aim of achieving an icing-safe configuration. A thermal analysis framework was established, and simulation results were compared against validation cases.

Extensive rig testing demonstrated the critical role of comparing dry and wet test conditions to capture discrepancies between experimental observations and numerical predictions. In most scenarios, dry-condition tests produced lower metal temperatures than those estimated by the model. When the thermal influence of impinging water droplets was isolated and evaluated separately, the numerical predictions aligned well at low-power operating conditions. However, at higher collection efficiencies and elevated flow velocities, the simulations tended to overpredict the water-induced effects.

These findings suggest that a two-step optimization strategy could enhance predictive accuracy: first by refining heat-transfer coefficients under dry conditions, and then by applying the same procedure to collection efficiency coefficients. Furthermore, the discrepancies observed across some cases indicate that adopting fractional adjustments of liquid water content (LWC) and hot-air supply mass flow rates on a test case specific basis may help resolve remaining mismatches.

Overall, the results highlight that while the present methodology successfully reproduces the dominant thermal behaviour under critical icing conditions, further refinement of heat-transfer modelling and droplet collection efficiency is required for higher power (also higher-temperature cases). These conclusions provide valuable guidance for future efforts aiming at more efficient anti-icing designs and improved consistency between numerical and experimental outcomes.

REFERENCES

- [1] Sammak M., (2006), “Anti-Icing in Gas Turbines”, M.Sc. Thesis, Lund Institute of Technology.
- [2] Rolls-Royce plc, (1996), “The Jet Engine”, 5th Edition, John Wiley & Sons.
- [3] Ray M. G., Browne J. L., (1980), “T700 Engine Integral Inlet Separator: All Weather Operational Protection”, 6th European Rotorcraft and Powered Lift Aircraft Forum, 37-53, Bristol, U.K., 16-18 September.
- [4] Al-Khalil K., Irani E., Rowley D., (2001), “Tunnel and natural icing tests of a helicopter engine cooling bay inlet”, 39th AIAA Aerospace Sciences Meeting and Exhibit, 1-13, Reno, NV, USA, 8-12 January.
- [5] European Union Aviation Safety Agency (EASA), (2022), Certification Specifications for Engines, Section 2, Subpart E, CS-E 780 Icing.
- [6] M. Muhammed, M. S. Virk, (2022), “Ice accretion on fixed-wing unmanned aerial vehicle—A review study”, Drones, 6 (4), 1-23.
- [7] Qiu C., Chen N., Hu Y., Wang B., Hu S., (2022), “Numerical study of ice accretion inside an inertial particle separator”, AIP Advances, 12 (5), 1-9.
- [8] Ignatowicz K., Morency F., Beaugendre H., (2019), “Numerical simulation of ice accretion using Messinger-based approach: Effects of surface roughness”, CASI AERO 2019, 1-15, Laval, QC, Canada, 14-16 May.
- [9] Fukudome K., Tomita Y., Uranai S., Mamori H., Yamamoto M., (2021), “Evaluation of anti-icing performance for a NACA0012 airfoil with an asymmetric heating surface”, Aerospace, 8 (10), 1-18.
- [10] Domingos R. H., Papadakis M., Zamora A. O., (2020), “Computational methodology for bleed air ice protection system parametric analysis”, AIAA Atmospheric and Space Environments Conference, 1-10, Toronto, Ontario, Canada, 2-5 August.
- [11] Bu X., Lin G., Yu J., Yang S., Song X., (2012), “Numerical simulation of an airfoil electrothermal anti-icing system”, Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, 227 (10), 1608–1622.
- [12] Li L., Liu Y., Tian L., Hu H., Liu X., Hogate I., Kohli A., (2020), “An experimental study on a hot-air-based anti-/de-icing system for aero engine inlet guide vanes”, Applied Thermal Engineering, 167 (1), 1-14.
- [13] Al-Khalil K. M., Horvath C., Miller D. R., Wright W. B., (1997), “Validation of NASA Thermal Ice Protection Computer Codes – Part 3: The Validation of Antice”, 35th AIAA Aerospace Sciences Meeting and Exhibit, 1-10, Reno, Nevada, USA, 6-7 January.

- [14] Castillo J., Ortega G., (2008), “Error evaluation in metal surface temperature prediction due to non-simulation of the effect of water impingement and runback water on an anti-iced turboprop engine inlet guide vane structure”, ASME Turbo Expo 2008, 49-58, Berlin, Germany, 9-13 June.
- [15] Chen N., Du J., Hu Y., Ji H., Yuan Y., (2020), “Study of the flow and impingement of water droplets inside an inertial particle separator”, AIP Advances, 10 (4), 1-9.
- [16] Ferrigno F., Vernillo P., Inverno M., (2005), “NH90 Air Intake Version T700/T6E1 Icing Qualification Tests”, 43rd AIAA Aerospace Sciences Meeting and Exhibit, 1-9, Reno, Nevada, USA, 10-13 January.
- [17] Bu X., Lin G., Shen X., Hu Z., Wen D., (2019), “Numerical simulation of aircraft thermal anti-icing system based on a tight-coupling method”, Applied Thermal Engineering, 160 (1), 1-12.
- [18] Messinger B. L., (1953), “Equilibrium temperature of an unheated icing surface as a function of air speed”, Journal of the Aeronautical Sciences, 20 (1), 29–42.
- [19] Stiefel W., (1989), “Environment icing test of T800 helicopter engine with integral inlet particle separator”, 25th AIAA/ASME/SAE/ASEE Joint Propulsion Conference, 1-12, Monterey, CA, USA, 10-12 July.
- [20] Anderson D. N., Tsao J.-C., (2005), “Overview of Icing Physics Relevant to Scaling”, FAA In-Flight Icing/Ground De-Icing International Conference, 1-24, Chicago, Illinois, USA, 16-20 June.
- [21] Myers T. G., (2001), “Extension to the Messinger model for aircraft icing”, AIAA Journal, 39 (2), 211–212.
- [22] Lowe P. R., (1976), “An approximating polynomial for the computation of saturation vapor pressure”, Journal of Applied Meteorology, 16 (1), 100–103.
- [23] Croce G., Beaugendre H., Habashi W. G., (2002), “CHT3D: FENSAP-ICE Conjugate Heat Transfer Computations with Droplet Impingement and Runback Effects”, 40th AIAA Aerospace Sciences Meeting and Exhibit, 1-12, Reno, NV, USA, 14-17 January.
- [24] Papadakis M., Wong S.-C., Rachman A., Hung K. E., Vu G. T., Bidwell C. S., (2007), “Large and Small Droplet Impingement Data on Airfoils and Two Simulated Ice Shapes”, Technical Report No: NASA/TM-2007-213959, National Aeronautics and Space Administration, Washington D.C., USA.
- [25] Nikitaev D., Smith C. D., Palomares K., (2022), “Comparison of convective heat transfer correlations and their application to nuclear thermal propulsion reactors”, Nuclear and Emerging Technologies for Space NETS 2022, 696-705, Cleveland, OH, USA, 8-12 May.

[26] Zukauskas A., (1972), “Heat transfer from tubes in cross flow”, *Advances in Heat Transfer*, 8, 93–160, Academic Press.

[27] Incropera F. P., DeWitt D. P., Bergman T. L., Lavine A. S., (2011), “*Fundamentals of Heat and Mass Transfer*”, 7th Edition, John Wiley & Sons, Hoboken.



BIOGRAPHY

Emir Musa Koyuncu graduated from the Department of Mechanical Engineering at Erciyes University in 2021. He began his professional career at TUSAŞ Engine Industries, Inc. (TEI) as a Thermal Systems Design Engineer where he has been working for over four years on thermal and fluid systems design, engine anti-icing and engine aerothermal testing.



PUBLICATIONS AND PRESENTATIONS FROM THE THESIS

[1] Koyuncu E. M., Ünverdi S. Ö., (2025), “Anti-icing system for a NACA0012 airfoil: A case study,” 11th International Azerbaijan Congress on Life, Engineering, Mathematical, and Applied Sciences, 43, Baku, Azerbaijan, 21–22 June.

