3D Conjugate Heat Transfer Simulation of Aircraft Hot-Air Anti-Icing Systems

Hong Zhi Wang

Department of Mechanical Engineering
McGill University
Montreal, Québec
September 2005

A Thesis Submitted to McGill University in Partial Fulfillment of the Requirements of the Degree of Master of Engineering

© Hong Zhi Wang, 2005
NOTICE:
The author has granted a non-exclusive license allowing Library and Archives Canada to reproduce, publish, archive, preserve, conserve, communicate to the public by telecommunication or on the Internet, loan, distribute and sell theses worldwide, for commercial or non-commercial purposes, in microform, paper, electronic and/or any other formats.

The author retains copyright ownership and moral rights in this thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without the author's permission.

In compliance with the Canadian Privacy Act some supporting forms may have been removed from this thesis.

While these forms may be included in the document page count, their removal does not represent any loss of content from the thesis.
Acknowledgments

First of all, I would like to express my great appreciation of my Professor, Dr. Wagdi Habashi who opened the door and led me into the marvelous world of CFD. I benefited from his superb academic attainments and outstanding leadership, not only in scientific research but also in philosophy of life during the course of my study at the CFD Lab.

I would like to thank the NSERC-J. Armand Bombardier Industrial Research Chair of Multidisciplinary CFD, whose financial support made my research possible.

I would also like to express my deep gratitude to my thesis co-supervisor, Dr. François Morency, currently a professor at ETS. He helped me during the entire course of my research and taught me a lot about icing and anti-icing simulation, with excellent and patient guidance. Moreover, many thanks are due to Professor Héloïse Beaugendre of Bordeaux University, my colleague at the beginning of my research, who shared a detailed knowledge of aero-icing simulation with me and instructed me in the hands-on experience to use different CFD tools.

I am also grateful to Dr. Frédéric Tremblay, Dr. Claude Lepage, and Martin Aubé at Newmerical Technologies International (NTI) for their valuable advices and sincere help in my research. My thanks also go to Dr. Croce of the University of Udine, a long-term associate of the CFD lab, for his advice and help in Conjugate Heat Transfer.

My colleges at the CFD Lab have given me continuous help and support in both my studies and my life. They have made my intensive Master study full of pleasure and beautiful memories. Because of them, I more and more love this peaceful country, my second motherland. I would like to give my special thanks to my buddies Raimund Honsek, Nabil Ben Abdallah, France Suerich-Gulick, Peter Findlay, Farid Kachra, LiangKan Zheng ...
I also wish to thank Laboratoire de Mécanique des Fluides (LMF) of Laval University for providing the anti-icing experiment conducted in 2003 by Jean Lemay, Yvan Maciel et al.

I would like to dedicate this thesis to my parents who have always loved and supported me unconditionally throughout my whole life. Also to be thanked are my best friends. They are the most important people in my life and love me as a member of their family.
Abstract

When an aircraft flies through clouds under icy conditions, supercooled water droplets at temperatures below the freezing point may impact on its surfaces and result in ice accretion. The design of efficient devices to protect aircraft against in-flight icing continues to be a challenging task in the aerospace industry. Advanced numerical tools to simulate complex conjugate heat transfer phenomena associated with hot-air anti-icing are needed. In this work, a 3D conjugate heat transfer procedure based on a loose-coupling method has been developed to solve the following four domains: the external airflow, the water film, conduction in the solid, and the internal airflow. The domains are solved sequentially and iteratively, with an exchange of thermal conditions at common interfaces until equilibrium of the entire system is achieved. A verification test case shows the capability of the approach in simulating a variety of anti-icing and de-icing cases: fully evaporative, running wet, or iced. The approach is validated against a 2D dry air experimental test case, because of the dearth of appropriate open literature 3D test data.
Résumé

Quand un avion vole à travers les nuages à des conditions glaciales, de la glace risque de se former sur ses surfaces en raison de l'impact des gouttelettes d'eau surfondues. La conception d'efficaces dispositifs qui protégerait l'avion contre cette formation de glace s'avère une tâche ardue et défiaante pour l'industrie aérospatiale. Des outils numériques simulant le transfert de chaleur conjugué associé au antigivrage par air chaud constituent un premier choix. Ce papier présente une méthode tridimensionnelle de transfert de chaleur conjugué, qui résout les quatre domaines suivants : l’écoulement d'air extérieur, le film d'eau, la conduction dans le solide ainsi que l’écoulement d'air interne. Les domaines sont résolus séquentiellement et itérativement, avec un échange de conditions frontières thermales au niveau des interfaces communes, jusqu'à l'établissement d'un équilibre global. Un cas test de vérification montre les capacités de l'approche dans la simulation d'une variété de situations d'antigivrage et dégivrage : évaporation totale, écoulement liquide ou glace. L'approche est ensuite validée à travers une comparaison bidimensionnelle avec un cas test expérimental. Le choix bidimensionnel est dicté par la rareté de cas tests tridimensionnels disponibles dans la littérature.
# Table of Contents

Acknowledgments ................................. i
Abstract ........................................... iii
Résumé ............................................... iv
Table of Contents ................................... v
List of Symbols ..................................... viii
List of Figures .................................... xiv
List of Tables ..................................... xvii

CHAPTER 1: Introduction .......................... 1
1.1 In-flight Icing Phenomena .................... 1
1.2 In-flight Icing Protection and Aircraft Anti/De-Icing Systems 2
1.3 CFD Tools for In-Flight Icing and Anti/De-Icing Analysis 5
1.4 Objective of Current Work .................... 7

CHAPTER 2: Literature Review .................... 9
2.1 Methodology of Conjugate Heat Transfer Calculation 9
  2.1.1 Empirical Method for Heat Transfer Problems 9
  2.1.2 Conjugate Method for Heat Transfer Problems 11
    2.1.2.1 Tight-Coupling Approach 11
    2.1.2.2 Loose-Coupling Approach 13
2.2 Overview of Numerical Simulations for Aircraft Hot-Air Anti-Icing Systems 16
  2.2.1 Hot-Air Anti-Icing Simulation in LEWICE 17
  2.2.2 Hot-Air Anti-Icing Simulation in CHT2D 20
2.3 Introduction of A Simplified Anti-Icing Experiment 22
2.4 Review of Heat Transfer for A Single Slot Jet Impingement 24
2.5 Conclusions .................................... 26

CHAPTER 3: Mathematical Equations and Numerical Methods .......... 27
3.1 Airflow Model (FENSAP) ....................... 27
  3.1.1 Governing Equations 27
  3.1.2 Numerical Discretization 29
  3.1.3 Turbulence Models 31
3.1.3.1 Spalart-Allmaras Turbulence Model
3.1.3.2 k-ε Turbulence Model
3.1.4 Rough Walls Prediction in Spalart-Allmaras Model
3.1.5 Convective Heat Flux
3.2 Droplets Impingement Model (DROP3D)
3.3 Water Film/ICE Accretion Thermodynamic Model (ICE3D)
3.4 Heat Conduction Model (C3D)

CHAPTER 4: Algorithm for Hot-Air Anti-Icing Simulation
4.1 General Introduction of Hot-Air Anti-Icing Simulation
4.2 Overall Energy Balance in the Hot-Air Anti-Icing System
4.3 Methodology of Hot-Air Anti-Icing Simulation
4.4 Algorithm of CHT3D Hot-Air Anti-Icing Model
4.4.1 Thermal Boundary Conditions for Each Solver
4.4.2 Flow Chart of CHT3D Algorithm
4.4.3 Boundary Condition Exchange at Each Interface
4.4.4 Implementation of Hot-Air Anti-Icing Simulation in CHT3D
4.4.5 Dual Surface Meshes
4.4.6 Boundary Condition Exchange for Matching and Non-Matching Grids
4.4.7 The Stability Analysis of Conjugate Heat Transfer Anti-Icing Simulation
4.5 Conclusions

CHAPTER 5: Testing of CHT3D Code
5.1 Description of a 2-D Verification Hot-Air Anti-icing Test Case
5.2 Hot-Air Anti-Icing Simulations
5.2.1 Case 1: Hot-Air Anti-Icing Simulation with Ice Accretion
5.2.2 Case 2: Hot-Air Anti-Icing Simulation with Running Wet
5.2.3 Case 3: Hot-Air Anti-Icing Simulation with Full Evaporation
5.2.4 Case 4: Hot-Air Anti-Icing Simulation Without Droplets Impingement
5.3 Conclusion

CHAPTER 6: Validation of CHT3D Code
6.1 Validation of Turbulence Models in FENSAP for Impinging Slot Jet
6.1.1 Description of Impinging Slot Jet Experiment
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.1.2</td>
<td>Numerical Implementation with Spalart-Allmaras Turbulence Model</td>
<td>72</td>
</tr>
<tr>
<td>6.1.3</td>
<td>Numerical Implementation with k-ε Turbulence Model</td>
<td>74</td>
</tr>
<tr>
<td>6.1.4</td>
<td>Comparison and Conclusion of Impinging Slot Jet Simulations</td>
<td>75</td>
</tr>
<tr>
<td>6.2</td>
<td>Validation of CHT3D Code with a Simplified Hot-Air Anti-Icing Test Case</td>
<td>76</td>
</tr>
<tr>
<td>6.2.1</td>
<td>Description of the Test Case</td>
<td>76</td>
</tr>
<tr>
<td>6.2.2</td>
<td>Initial Wind Tunnel Airflow Simulation</td>
<td>78</td>
</tr>
<tr>
<td>6.2.3</td>
<td>Initial Impinging Jet Simulation</td>
<td>82</td>
</tr>
<tr>
<td>6.2.4</td>
<td>CHT3D Numerical Simulation</td>
<td>84</td>
</tr>
<tr>
<td>6.3</td>
<td>Conclusions</td>
<td>90</td>
</tr>
<tr>
<td><strong>Conclusions and Future Work</strong></td>
<td></td>
<td>91</td>
</tr>
<tr>
<td><strong>Bibliography</strong></td>
<td></td>
<td>93</td>
</tr>
</tbody>
</table>
List of Symbols

**Latin alphabet**

\( c_n \) Spanwise distance between holes [m]
\( c_p \) Pressure coefficient at the wall
\( c_p \) Specific heat capacity [J/(kg.K)]
\( c_{p,w} \) Specific heat capacity of water [J/(kg.K)]
\( c_{p,ice} \) Specific heat capacity of ice [J/(kg.K)]
\( c_s \) Specific heat of solid [J/(kg.K)]
\( c_w \) Specific heat of water [J/(kg.K)]
\( d \) Jet hole diameter [m]
\( d \) Droplets diameter [m]
\( h_c \) Convective heat transfer coefficient [W/(m².K)]
\( h_f \) Water film thickness [m]
\( k \) Thermal conductivity [W/(m.K)]
\( k_{eff} \) Effective thermal conductivity [W/(m.K)]
\( k_f \) Thermal conductivity of jet
\( k_s \) Thermal conductivity of solid [W/(m.K)]
\( k_\varphi \) Thermal conductivity of body in direction normal to the surface [W/(m.K)]
\( k_s \) Equivalent sand-grain roughness [m]
\( l \) or \( L \) Characteristic length [m]
\( \dot{m} \) Mass flow rate per unit span [kg/(m.s)]
\( \dot{m}_{evap} \) Mass flux of evaporation [kg/(m².s)]
\( \dot{m}_f \) Mass flux of water freezing in the control volume [kg/(m².s)]
\( \dot{m}_{ice} \) Mass flux of accumulation of ice [kg/(m².s)]
\( \dot{m}_{sh} \) Mass flux of water shedding in the control volume [kg/m².s]
\( \dot{m}_w \) Mass flux of standing water in control volume [kg/m\(^2\)s]

\( \dot{m}_w \) Water flux at the aerodynamic surface \( \dot{m}_w = LWCU_\omega \beta \) [kg/(m\(^2\)s)]

\( \vec{n} \) Normal to the aerodynamic surface

\( p \) Pressure [Pa]

\( q \) Heat flux [W/m\(^2\)]

\( q''_{sc} \) Net convective heat loss from body [W/m\(^2\)]

\( q''_{ke} \) Kinetic heat gain to body from impinging droplets [W/m\(^2\)]

\( q''_{lat} \) Latent heat gain to body from freezing [W/m\(^2\)]

\( q''_{sens} \) Sensible heat loss (or gain) from surface water [W/m\(^2\)]

\( q_w \) Heat flux at the wall [W/m\(^2\)]

\( q^f_w \) Heat flux at the CHT interface in fluid domain [W/m\(^2\)]

\( q^s_w \) Heat flux at the CHT interface in solid domain [W/m\(^2\)]

\( r \) Radial distance from holes [m]

\( u \) Fluid velocity of the main stream flow [m/s]

\( \vec{u}_a \) Non-dimensional velocity of air

\( \vec{u}_d \) Non-dimensional velocity of droplets

\( \vec{u}_f \) Liquid water film velocity [m/s]

\( u_i \) Component I of the velocity, \( i = 1, 2, 3 \) [m/s]

\( u_r \) Friction velocity [m/s]

\( u^* \) Non-dimensional velocity in wall coordinates \( u^* = \frac{U}{u_r} \)

\( y^* \) Non-dimensional distance from wall coordinates \( y^* = \frac{\rho u_r y}{\mu} \)

\( z_n \) Distance from hole to wall [m]

\( A^f \) The cell area at the CHT interface in fluid domain [m\(^2\)]

\( A^s \) The cell area at the CHT interface in solid domain [m\(^2\)]

\( C_D \) Drag coefficient
\( C_p \) Specific heat capacity of jet \([\text{W.s/(kg.K)}]\)

\( D \) Diameter [m]

\( D_{AB} \) Binary diffusion

\( Fr \) Froude number \( Fr = \frac{U_\infty}{\sqrt{g_0}} \)

\( H \) Total enthalpy \([\text{J/kg}]\)

\( \dot{H} \) Flux of enthalpy \([\text{J/s}]\)

\( K \) Inertia parameter \( K = \frac{\rho d^2 U_\infty}{18 l \mu} \)

\( L_f \) Latent heat of fusion \([\text{J}]\)

\( L_{\text{evap}} \) Latent heat of evaporation \([\text{J/kg}]\)

\( L_{\text{sub}} \) Latent heat of sublimation \([\text{J/kg}]\)

\( L_{\text{fus}} \) Latent heat of fusion \([\text{J/kg}]\)

\( LWC \) Liquid water content \([\text{kg/m}^3]\)

\( M_\infty \) Mach number at freestream

\( N_r \) Number of jet rows

\( N_u \) Local Nusselt number, convective heat transfer parameter, \( N_u = \frac{h_L}{k_f} \)

\( \overline{Nu} \) Based on the assumption of constant temperature difference

\( \overline{Nu_q} \) Based on the assumption of constant heat flux

\( P_t \) Prandtl number \( P_t = \frac{\nu}{\alpha} \)

\( \text{Pr}_\infty \) Prandtl number \( \text{Pr}_\infty = \frac{\mu \rho c_p \infty}{k_\infty} \)

\( \dot{Q}_{\text{cond}} \) Conductive heat flux \([\text{W/m}^2]\)

\( \dot{Q}_{\text{conv}} \) Convective heat flux \([\text{W/m}^2]\)

\( \dot{Q}_{\text{evap}} \) Evaporative heat flux \([\text{W/m}^2]\)
\( \dot{Q}_h \) Convective heat flux [W/m\(^2\)]

\( \dot{Q}_{\text{rad}} \) Radiative heat flux [W/m\(^2\)]

\( \dot{Q}_{\text{conv}} \) Convective heat flux [W/m\(^2\)]

\( \dot{Q}_p \) Kinematic heat flux from impinging droplets [W/m\(^2\)]

\( R \) Gas constant

\( \text{Re}_d \) Droplets Reynolds number  \( \text{Re}_d = \frac{\rho d U_\infty |\bar{u}_w - \bar{u}_d|}{\mu} \)

\( \text{Re}_\infty \) Reynolds number  \( \text{Re}_\infty = \frac{\rho \infty L U_\infty}{\mu_\infty} \)

\( \text{Re}_B \) Reynolds number based on the width of slot jet B,  \( \text{Re}_B = \bar{u}B / \nu \)

\( \text{Re}_D \) Reynolds number based on the hydraulic diameter D,  \( \text{Re}_D = \bar{u}D / \nu \)

\( S_c \) Schmidt number  \( S_c = \frac{\nu}{D_{AB}} \)

Error! Objects cannot be created from editing field codes. Sherwood numbers, convective mass transfer parameter  \( \text{Sh} = \frac{h_n L}{D_{AB}} = + \frac{\partial C_A^*}{\partial y^*}\bigg|_{y^*=0} \)

\( T \) Temperature [K]

\( \bar{T} \) Equilibrium temperature within the air/water film/ice/wall interface [°C]

\( \bar{T}_{d,\infty} \) Droplets temperature at freestream [°C]

\( T_b \) Bulk temperature [K]

\( T_{\text{jet}} \) Temperature of jet

\( T_{\text{ref}} \) Reference temperature [K]

\( T_w \) Temperature on wall [K]

\( T_{w}^f \) The local temperature at the CHT interface in fluid domain [K]

\( T_{w}^s \) The local temperature at the CHT interface in solid domain [K]

\( W \) or \( B \) Slot width [m]

\( W_i \) Weight function
Greek alphabet

\( \alpha \)  
Droplet volume fraction, ratio of the volume occupied by water over the total volume of the fluid element

\( \alpha \)  
Thermal diffusivity 
\[ \alpha = \frac{k_f}{\rho C_p} \]

\( \beta \)  
Local collection efficiency 
\[ \beta = -\alpha \vec{u} \cdot \vec{n} \]

\( \varphi \)  
Direction normal to body surface [m]

\( \gamma \)  
Ratio of specific heat

\( \sigma \)  
Boltzmann constant \([=5.670 \times 10^{-8} \text{ W/(m}^2\text{K}^4)]\)

\( \varepsilon \)  
Solid emissivity

\( \mu \)  
Dynamic viscosity coefficient \([\text{kg/(s.m)}]\)

\( \mu_T \)  
Turbulence dynamic viscosity coefficient \([\text{kg/(s.m)}]\)

\( \nu \)  
Kinematic viscosity \([\text{m}^2/\text{s}]\)

\( \rho \)  
Density \([\text{kg/m}^3]\)

\( \rho_a \)  
Density of air

\( \rho_s \)  
Density of solid

\( \tau_{ij} \)  
Shear stress tensor

\( \delta^{ij} \)  
Kronecker delta, i.e., \( \delta^{ij} = 1 \) if \( i=j \) and \( \delta^{ij} = 0 \) if \( i \neq j \)

\( \Delta r_{\text{current}} \)  
Surface distance spacing at current location [m]

\( \Delta r_{\text{next}} \)  
Surface distance spacing at next location [m]

Subscripts

\( \text{conv} \)  
Convection

\( \text{cond} \)  
Conduction

\( \text{evap} \)  
Evaporation

\( f \)  
Water film or fluid
Ice accretion
Impinging water
Entering a control volume
Leaving a control volume
Reference value
Solid
Turbulence
Boundary surface
Free stream value
List of Figures

Figure 1.1 Areas That May Require Ice Protection, Source: FAA, Technical Report ADS-4, December 1963 ........................................................................................................ 2
Figure 1.2 Temperature Profile and Streamlines From the Piccolo Jet, Inside A 3D Wing Slat [15] ........................................................................................................................................ 5
Figure 2.1 The Empirical Method for Conjugate Heat Transfer Problem ........................................ 10
Figure 2.2 CHT2D Anti-Icing Algorithm (From Reference [26]) ....................................................... 20
Figure 2.3 Water Film Region (From Reference [32]) ......................................................................... 21
Figure 2.4 The Construction of the Laval University Anti-Icing Experimental Assembly (From Reference [33]) ............................................................................................... 23
Figure 2.5 The Flow Pattern of A Single Round or Slot Impinging Jet...................................... 25
Figure 4.1 The Hot-Air Anti-Icing Heat Transfer Processes ......................................................... 44
Figure 4.2 The Overall Energy Balance in the Hot Anti-Icing System ............................................ 47
Figure 4.3 The Methodology of CHT3D Anti-Icing Simulation ....................................................... 49
Figure 4.4 The Flow Chart of CHT3D Algorithm ............................................................................. 51
Figure 4.5 Dual Meshes on Structured Grid ................................................................................... 56
Figure 4.6 Construction of the Virtual Grid for A Square Surface: (A) Fluid and Solid Grids at Fluid/Solid Interface; (B) Final Virtual Grid at Fluid/Solid Interface [47]. 58
Figure 4.7 Evaluation of A Surface Integral on the Virtual Grid [47]: the Red Triangle Is a Solid Face and Blue Triangles Are Fluid Faces .................................................... 59
Figure 5.1 The Verification Test Case for CHT3D ........................................................................... 62
Figure 5.2 Water Film Mass Flow on Upper Surface of the Flat Plate ........................................... 63
Figure 5.3 Heat Flux Distribution on Upper Surface of the Flat Plate ............................................ 63
Figure 5.4 Temperature Distribution on Both Surfaces of the Flat Plate ..................................... 64
Figure 5.5 Water Film Mass Flow on Upper Surface of the Flat Plate ........................................... 65
Figure 5.6 Heat Flux Distribution on Upper Surface of the Flat Plate ............................................ 65
Figure 5.7 Temperature Distribution on Both Surfaces of the Flat Plate ..................................... 66
Figure 5.8 Water Film Mass Flow on Upper Surface of the Flat Plate ........................................... 67
Figure 5.9 Heat Flux Distribution on Upper Surface of the Flat Plate ............................................ 67
Figure 5.10 Temperature Distribution on Both Surfaces of the Flat Plate .................................... 67
Figure 5.11 Convective Heat Flux Distribution on the Upper Surface of the Flat Plate .. 68
Figure 5.12 Temperature Distribution on Both Surfaces of the Flat Plate ...................... 68
Figure 6.1 Fully Developed Laminar Slot Jet with Impingement on a Flat Plate .............. 70
Figure 6.2 Mesh with 11800 Nodes for Spalart-Allmaras Turbulence Model ................. 73
Figure 6.3 $Y^+$ of the First Layer Nodes Away From Impinging Wall ......................... 73
Figure 6.4 Mach Number Distribution Solved with Spalart-Allmaras Model ................. 74
Figure 6.5 Mesh with 5880 Nodes for k-ε Turbulence Model ....................................... 75
Figure 6.6 Mach Number Distribution Solved with k-ε High Reynolds Model ............... 75
Figure 6.7 Comparison of Local Nusselt Number between k-ε Model, Spalart-Allmaras Model, and Experimental Result ................................................................. 76
Figure 6.8 The Geometry of the Simplified Anti-Icing Test Case ................................. 77
Figure 6.9 The Wind Tunnel Section .............................................................................. 77
Figure 6.10 The Aluminum Flat Plate Section ............................................................... 78
Figure 6.11 The Impinging Jet Section ........................................................................... 78
Figure 6.12 The Partial Mesh of the Wind Tunnel Airflow ............................................ 79
Figure 6.13 Inlet Velocity Profile of Wind Tunnel Air Flow .......................................... 80
Figure 6.14 The Mach Number Distribution of the Wind Tunnel Airflow ....................... 80
Figure 6.15 Comparison of the Pressure Coefficient on the Wind Tunnel Floor with Experimental Result ................................................................. 81
Figure 6.16 The Mesh of the Impinging Jet flow ............................................................ 82
Figure 6.17 The Mach Number Contour Lines of Jet Flow .......................................... 83
Figure 6.18 Comparison of Heat Transfer Coefficient Distribution on Impinging Surface ................................................................................................................. 84
Figure 6.19 The Mesh of the Aluminum Flat Plate ......................................................... 85
Figure 6.20 CHT3D Convergence History of the Temperature on the Flat Plate Upper Surface ........................................................................................................... 86
Figure 6.21 Heat Transfer Coefficients on Both Surfaces of Aluminum Flat Plate .......... 87
Figure 6.22 Convective Heat Fluxes on Both Surfaces of Aluminum Flat Plate ............... 87
Figure 6.23 Temperature Distribution in the Aluminum Plate (The Thickness of the Plate Is Scaled for Better View) ................................................................. 88
Figure 6.24 Temperature Distributions on Both the Upper and Lower Surfaces of Aluminum Flat Plate .................................................................................................. 88
Figure 6.25 Comparison of Heat Fluxes on Upper Surface of the Aluminum Flat Plate between CHT3D Calculation and Experimental Result with Error Bar .......................... 89
List of Tables

Table 6.1 Boundary Conditions for Slot Impinging Jet Test Case........................................ 72
Table 6.2 Boundary Conditions for the Wind Tunnel Airflow........................................... 79
Table 6.3 Boundary Conditions for the Impinging Jet Flow............................................. 82
CHAPTER 1: Introduction

1.1 In-flight Icing Phenomena

In-flight icing is a major aviation hazard that seriously threatens flight safety and has been the cause of several aircraft accidents, such as the accidents of American Eagle ATR-72 in 1994 and Embraer EMB-120 in 1997 [1]. A recent survey of commercial pilots indicates that the frequency of icing encounters is quite high, with de-icing devices having to be activated on up to 80% of turboprop flights [2]. Therefore, the guaranteed protection of aircraft surfaces and critical engine components against ice or ice effects remains a major concern to aircraft and engine manufacturers and to certification regulators.

When an aircraft flies through clouds under icy conditions, supercooled water droplets at temperatures below the freezing point may impact on aircraft surfaces and result in ice accretion. The ice accretion on critical surfaces of an aircraft such as wings, propellers and stabilizers can have a significant impact on operation and controllability of an aircraft. It may increase the total weight of the aircraft, shift its center of gravity, freeze the movable components such as flaps and slats and deteriorate the aerodynamic properties of airflow causing a substantial decrease of lift, increase in drag, and reduction of stall margin. The accreted ice on the nacelle or the wing in front of an engine may be ingested into the engine inlet as foreign object damage (FOD), therefore causing power fluctuations, thrust loss, roll back, flame out and loss of transient capability [3].

There are three types of icing formations: rime ice, glaze ice and mixed ice [3]. Rime ice usually occurs at low temperatures from -40 to -10 °C, low airspeed and low Liquid Water Content (LWC). In this situation, all the super-cooled water droplets that impact on a surface will freeze immediately. Rime ice still has a reasonable aerodynamic shape but with higher surface roughness, that greatly increases the aerodynamic drag.
Glaze ice appears at higher temperatures from −3 °C up to the freezing point of water, high airspeed and high LWC. The ice is formed from runback water and has the shape of horns or lobster tails that have less surface roughness but create a big change in original airfoil shape and cause the separation of airflow. Mixed ice is the combination of rime ice and glaze ice.

1.2 In-flight Icing Protection and Aircraft Anti/De-Icing Systems

In order to ensure flight safety in icing conditions and meet FAA or other national aircraft certification regulations, which require an aircraft to be able to operate safely throughout the icing envelope of Part 25 Appendix C, icing protection mechanisms have to be employed on some critical locations of an aircraft. The majority of these locations are shown in the figure below, referenced from a FAA technical report.

![Figure 1.1 Areas That May Require Ice Protection, Source: FAA, Technical Report ADS-4, December 1963](image-url)
The classification of active methods used to prevent ice accretion on critical aircraft surfaces is generally divided into Anti-icing and De-icing. Anti-icing systems will completely prevent ice formation at all times by evaporating most impinging water droplets on the protected surfaces and keeping the temperature of these surfaces above the freezing point. Anti-icing systems are usually only used on the most critical locations because of their high-energy requirement. In contrast, De-icing systems will periodically melt or break up the ice that has already formed on impingement surfaces, thus allowing accretion of ice before and between de-icing cycles.

Many kinds of ice protection devices have been developed, which give airframers a wide variety of choices according to the requirements of a particular application. They are commonly classified into three categories as follows.

**Thermal anti-icing/de-icing** may use either hot-air or electro-thermal energy to heat the protected surfaces. The anti-icing/de-icing effect depends on the amount of the added heat flux. High heat flux can result in “fully evaporative” anti-icing, which will evaporate all impinging droplets and hence keep the surface dry. Usually, as much as 70% of the thermal energy is used to evaporate the water in such a fully-evaporative case [4]. Low heat flux can result in a “running wet” condition on the surface, where liquid droplets run back on the surface and may freeze in unprotected locations. The hot air, bled from the compressors of aircraft engines, is forced into piccolo tubes by way of an interconnecting ductwork and subsequently discharged from small holes at their end, impacting on the internal surface of protected locations. Hot-air systems present a simple, highly reliable and efficient solution, especially for fully-evaporative anti-icing purposes. The penalty associated with this system is the increase in both fuel consumption and turbine temperature [5]. Electro-thermal systems usually use electro-thermal heaters embedded into the solid to heat protected surfaces where anti-icing/de-icing hot air is not available, such as propellers, spinners, and center windshield panels. This system has a short response time and facilitates heat customization; however, it is limited by high power requirement and the need for an additional generator [3]. On the recently developing Boeing’s 787 Dreamliner, a pair of large generators on each engine can generate total 500
kilovolt-amps (kva) power per engine, which is more than four times the amount of power extracted from engines on other aircraft. This innovative design makes it possible to run all aircraft systems by electricity and to essentially eliminate the bleed air from engines. On this so-called all-electric airplane, electric heat will be used for the anti-icing of the wing, which needs a large electrical load of approximate 100kw. The only remaining bleed air is used for the engine inlet anti-icing [6].

**Boots, or mechanical de-icing** is designed to break and shed the ice from protected surface into the air stream by suddenly deforming the iced surface. A lot of surface deformation systems have been developed for mechanical de-icing, such as Pneumatic Boot Systems, Pneumatic-impulse De-icing Systems, Electro-mechanical Expulsive De-icing Systems (EMEDS), Electro-impulse De-icing Systems (EIDI), Eddy Current De-icing Systems (ECDS), Electro-expulsive De-icing System (EEDS) and Shape Memory Alloys.

The boot is not an anti-icing device but a de-icer. Hence, a minimum initial ice is needed for boot to be effective. Moreover, the mechanical de-icing systems cannot operate continuously and they are intermittent by nature with blackout periods as the cycle can only do wings, empennage, etc. in sequence. As a result, inter-cycle and intra-cycle ice may form. In addition, residue ice may remain after each de-icing cycling due to the less effective of these systems for thin ice.

**Fluid protection system** prevent the accretion of ice by using freezing point depressant fluids (FPD), usually deposited by spray bar of windshield, slinger ring and inertia of propeller, and porous wing surfaces to reduce the freezing point. It is limited by fluid supply, weight, and expense.

Of all anti/de-icing mechanisms discussed above, the hot bleed air systems are the most reliable and efficient ones, and are therefore widely used on critical ice protection regions of most commercial aircraft, such as leading edge wing panels and high lift devices, empennage surfaces, engine inlet and air scoops, radomes, and some types of instruments.
The following figure shows the numerical simulation of a typical 3D hot-air anti-icing device detailing the complex 3D nature of the jet flow around a piccolo tube inside a wing slat, which was performed by Croce, Habashi et al. [15].

![Figure 1.2 Temperature Profile and Streamlines From the Piccolo Jet, Inside A 3D Wing Slat [15]](image)

1.3 CFD Tools for In-Flight Icing and Anti/De-Icing Analysis

Traditionally, the icing, anti/de-icing analysis and icing certifications were performed by a combination of simulation methods that comprise CFD, icing tunnels and by tanker testing, and by natural flight testing. The combination of the above methods often cannot simulate and test the full range of the icing envelope required by FAR Part 25 Appendix C, because each exhibits some deficiencies [7].

The icing tunnel experimental testing is usually done at low Reynolds number (Re) and cannot simply extrapolate the result to high Re for larger aircraft, as the scaling process is highly nonlinear. The results from different icing tunnels often present some disagreement and the experimental result cannot be repeated in some tunnels. The errors may also be due to the partial geometries tested, relatively primitive ice shape measurement techniques, tunnel wall effects and scaling techniques for controlling icing parameters, still a maturing area of research.
Flight-testing behind a tanker features the following deviations from natural in-flight icing conditions:

- the generation of supercooled droplets is not possible due to the impurity of tanker water
- the dyed water may cause more runback effect than supercooled water
- the tanker moves when the droplets are released
- the droplets AoA is not exactly the same as the aircraft AoA because of the width and breadth of the spray
- more uncertainties may also be caused by the sub-saturated condition, droplets evaporation, coagulation, break-up and spectra limitations
- No real measurements can be made during flight, only qualitative observations

Natural flight-testing has seasonal limitations and is risky in some extreme situations. It is not completely general and not suitable to qualify ice shapes or performance effects. Moreover, ice accretions on some locations are difficult to be observed, such as on the wing tips, bottom of wings, or on top of high-wings.

In recent years, with the dramatically increased computational power and decreased cost to solve the Navier-Stokes equations of complex 3-D fluid flows, accurately capturing the physics of the flow has become realistic. Computational Fluid Dynamics (CFD) has been playing an important role in in-flight icing analysis, anti-/de-icing system design and airworthiness certification, such as the design of icing tunnel models, determining worst case scenarios for icing encounters, estimating the ice shapes and aerodynamics penalties for delayed turn-on and failure cases and for unprotected surfaces, estimating temperature and heat flux of anti-icing system, completing study of design parameters for preliminary anti-/de-icing system design and optimization of anti-/de-icing system. Comparing with traditional methods, CFD analysis has shown advantages such as reducing and focusing experiments and flight tests, shortening the development period, flexibility for optimization design and parameter study, simulating a wider range of icing condition, avoiding experimental assumptions and measurement errors, capability to analysis the whole anti-/de-icing system with much more detailed data than measurement at
instrumented areas [7]. On the other hand, CFD analysis also has its disadvantages, which may arise from neglecting some physical phenomena in the mathematical equations, inaccuracy of numerical discretization, incomplete convergence and instability of numerical calculations, etc. Therefore, CFD analysis still needs to combine with other methods for engineering applications.

More and more countries and aerospace companies have been engaging in their own icing and anti-/de-icing research and CFD tools development such as LEWICE from Glenn Icing Research Center of NASA in U.S. [8], ONERA-ICE from ONERA in France, CANICE from Bombardier Aerospace in Canada [9], FENSAP-ICE from Numerical Technologies International Canada [10]. LEWICE is a 2D ice accretion numerical model originally designed for solving the flow field, droplets trajectory and impingement, and ice accretion. It has also expanded for hot-air or electro-thermal anti-icing simulations. CANICE is also a 2D ice accretion model based on the similar algorithm as LEWICE. FENSAP-ICE is a fully 3D CFD package including 3D Navier-Stokes airflow model, 3D Eulerian droplet impingement model, 3D ice accretion thermodynamic model, 3D mesh optimization tool and 3D conjugate heat transfer model.

1.4 Objective of Current Work

The FENSAP-ICE package developed by CFD Lab of McGill University and Numerical Technologies International (NTI) is a state-of-the-art 3D modular simulation tool to analyze the aerodynamics, droplet impingement, ice accretion, and thermal properties of any anti-icing system. The objective of this thesis is to develop a three-dimensional conjugate heat transfer (CHT) module labeled CHT3D, based on FENSAP-ICE’s capabilities, to simulate the complicated thermal processes of aircraft hot-air anti-icing systems. This procedure is based on a continuous exchange of boundary conditions at fluid/solid and fluid/water interfaces. It couples together a Navier-Stokes flow solver (FENSAP), an Eulerian droplet impingement module (DROP3D), a 3D ice accretion module (ICE3D) and a solid conduction module (C3D). Such a general CHT3D procedure can also be applied to other CHT problems such as mist-cooled heat exchangers, turbine blade cooling, automotive and aircraft brake cooling, etc.
A literature review of CHT methodologies and anti-icing simulations is presented in chapter 2. The mathematical governing equations and numerical methods used for the entire anti-icing simulation are discussed in chapter 3. In chapter 4, a loose-coupling CHT algorithm has been developed and implemented for anti-icing simulations and will be elaborated. In chapter 5, a test case shows the capability of CHT3D in simulating a span of anti-icing and de-icing cases: fully evaporative, running wet, or iced; A jet impingement test case is implemented for validating different turbulence models in the airflow solver for predicting the heat transfer properties of the impinging slot jet; The CHT3D procedure is finally validated against a 2D dry air experimental test case from Laval University, because of the dearth of appropriate open literature 3D test data.
CHAPTER 2: Literature Review

2.1 Methodology of Conjugate Heat Transfer Calculation

In a typical anti-icing system of an aircraft wing, the hot air is bled from engine compressors, ducted into a piccolo tube from the pneumatic manifold and impinged on the internal front surface of the wing to prevent ice accretion on the external surface. Modern turbofan engines are designed with higher and higher bypass ratios in order to improve fuel economy and noise control, causing a decrease of the core engine size, thus limiting the available bleeding air. Therefore, maximizing the anti-icing efficiency in order to minimize bleed air becomes increasingly crucial in anti-icing system design to reduce the performance penalties of turbofan engines. For this purpose, CFD numerical tools are continuously being developed to assist in designing efficient anti-icing systems and in certification [7]. Similarly, in a jet engine, cooling air is deviated to high thermal load regions to reduce turbine blade temperature and prevent material failure. Both applications rely on the same heat transfer process, which includes heat conduction through a solid and heat convection caused by both internal and external airflows.

Many numerical models have been developed for solving these types of heat transfer problems, and are generally divided into empirical and conjugate methods. Empirical methods do not take into account the interaction between the flow medium and the solid, using empirical correlations to predict the convective heat loads due to airflows on solid surfaces and are widely used in industry. Conjugate methods would be based on the iterative exchange of boundary conditions at each fluid-solid interface until thermal equilibrium is eventually achieved at each interface. This is the approach retained in the present thesis.

2.1.1 Empirical Method for Heat Transfer Problems

In this approach, the local heat transfer coefficients and fluid reference temperatures (or bulk temperatures) on both internal and external solid walls are evaluated from empirical formulae, prior to their use as boundary conditions of the heat conduction calculation to
determine the temperature distributions in a solid body. The empirical method is shown schematically in Figure 2.1 below.

![Figure 2.1 The Empirical Method for Conjugate Heat Transfer Problem](image)

Chmielniak et al. [11] solved a turbine blade cooling problem with both empirical and conjugate methods. In their calculations based on the empirical method, the fluid velocity of the mainstream flow and the fluid temperature at the blade wall are obtained by solving the fluid flow problem assuming adiabatic boundary conditions at the blade wall. Then the local heat transfer coefficients at the blade wall can be calculated from empirical formulae, which are based on the previously calculated fluid velocity and temperature at the blade wall. It can be simply expressed as the following relationship:

\[ h_c = f(Nu) = g(u, T_w) \]  

(2.1)

Solving the problem with the empirical method is simpler and faster than using the conjugate method because the local heat transfer coefficients are based on empirical formulae and the airflow solution does not need to be updated. But the accuracy of the
results from empirical methods is very limited. One reason is the inaccuracy of the empirical formulae themselves to predict the local heat transfer coefficients. Another reason is that the fluid velocities and temperatures obtained from airflow computation are not updated in uncoupled methods and are only based on the adiabatic wall boundary condition, which is not a physically representative boundary condition. Thus, it will cause another source of error in the calculations of empirical formulae, which are based on above velocity and temperature. As well, the empirical formulae are application- and geometry-dependent and accurate correlations are difficult to obtain for complicated and three-dimensional geometries, thus the applicability of empirical methods to solve heat transfer problems is greatly restricted.

2.1.2 Conjugate Method for Heat Transfer Problems

The Conjugate Heat Transfer (CHT) method iteratively solves the thermal interactions between fluid and solid domains. In this method, multi-domains are used to separate individual fluid and solid fields, and each domain has its own mesh. The coupled method can directly calculate the heat transfer loads at fluid-solid interfaces instead of using empirical correlations and its result is more accurate than empirical methods, but it needs more computational power and longer times. The conjugate method can be represented by two different approaches: the tight-coupling method and the loose-coupling method. The loose-coupling method is the one used in the present anti-icing simulation. Both coupling methods are based on the same conception, which ensures temperature and heat flux equality conditions at each CHT interface as follow:

\[ T_w^f = T_w^s \]  \hspace{1cm} (2.2)

and

\[ q_w^f A^f = q_w^s A^s \] \hspace{1cm} (2.3)

2.1.2.1 Tight-Coupling Approach

In this approach, the heat conduction code is embedded into a CFD code, facilitating the solution of fluid flow and heat conduction by a single code. It is a very robust and stable coupling method because no further iteration process is required.
Marini [12] embedded the CHT capability into FENSAP, a finite element based Navier-Stokes CFD solver. In his work, the CHT problem is treated as a multi-domain problem and each domain has individual mesh. The meshes of all domains are later combined together inside FENSAP so that the solid energy equation and the fluid energy equations can be solved simultaneously in a fully implicit manner. A special treatment, instead of the nodal connectivity, is used to guarantee the equality conditions of temperatures and heat fluxes at the CHT interface. The nodes in the fluid grid at the CHT interface are termed 'dead' nodes and any other nodes in both fluid and solid grids are termed 'live' nodes. Similarly, the elements in the fluid grid at the CHT interface are termed 'dead' elements and any other elements in both fluid and solid grids are termed 'live' elements. The energy equations of both liquid and solid domains are discretized and simultaneously solved by the finite element method. The local element matrixes of all the live elements are assembled into global matrix using the standard finite element method according to the node connectivity of the meshes, so that, all live nodes have a direct representation in the global matrix and are directly solved by the matrix solver. In dead elements, which are the elements on the CHT interface in the fluid domain, the dead fluid nodes have an indirect representation in the global matrix and their contributions are communicated through the surrounding live solid nodes at the CHT interface. Therefore, at CHT interfaces, temperatures of the nodes belonging to solid domain are directly solved for and updated through the iterative solver. Temperatures in the fluid domain at the same interface are not directly solved by the matrix solver and are interpolated from the values of live solid nodes. Hence, the temperature equality in equation (2.2) is automatically guaranteed. The heat fluxes are forced to be identical at the CHT interface due to the implicit treatment in the weak-Galerkin finite element formulation. When the energy equations of fluid and solid domain are solved, it is assumed that surface integral terms in the weak-Galerkin residues of the fluid energy equation and the solid energy equation cancel out each other at the CHT interface. That is:

$$\int_{\alpha_f} W_i \left[ \frac{1}{Re_\infty Pr_\infty (\gamma_\infty - 1)M_\infty^2} q_f \right] dS = \int_{\alpha_s} W_i \left[ \frac{1}{Re_\infty Pr_\infty (\gamma_\infty - 1)M_\infty^2} q_s \right] dS$$

(2.4)
After canceling out the same terms on both sides of the equation, one can get the simplified form:

\[ \oint_{A_r} W_r q_f dS = \oint_{A_r} W_r q_d dS \]  \hspace{1cm} (2.5)

Hence, the heat flux equality at the CHT interface is guaranteed by this assumption.

2.1.2.2 Loose-Coupling Approach

This approach couples CFD code, structure conduction code, and other numerical thermal codes, such as ice accretion and fogging, externally only via a continuous and iterative exchange of boundary conditions. Field equations in each computational domain are solved individually to provide boundary conditions for the start of calculation of adjacent computational domains. The loose-coupling approach works as a pure interfacial algorithm and is completely independent from the details of existing solvers of each computational domain. Theoretically, it can couple any number of domains to simulate very complicated conjugate heat transfer problems such as hot-air anti-icing. It has great flexibility to use any available CFD code using heterogenous numerical discretization schemes. Thus, the loose-coupling method is thought to be the most efficient and versatile for complex conjugate heat transfer simulations. Numerical instability of loose-coupling approaches can be eliminated by properly choosing boundary conditions in each domain and using under-relaxation schemes during boundary condition exchanges. Giles [13] has used a simple one-dimensional finite difference mode to study the numerical stability of loose-coupling approach in fluid/structure thermal analysis and has shown that imposing temperature boundary conditions (Dirichlet boundary conditions) for the fluid calculation and heat flux boundary conditions (Neumann boundary conditions) for the structural calculation will achieve numerical stability, but that in contrast imposing heat flux boundary conditions (Neumann boundary conditions) for the fluid calculation and temperature boundary conditions (Dirichlet boundary conditions) for the structural calculation will lead to numerical instability, unless extremely small time steps are used.

Croce et al. have developed a loose-coupling approach [14] for CHT calculations and have successfully implemented thermal analysis for an anti-icing device [15], a mist-flow
heat exchanger [16] and turbomachinery compressible flow applications [17]. This approach corresponds to the Shur Complement algorithm for domain decomposition of partial differential equations described by Funaro et al. [18] and can be summarized as follows:

1. Solve Navier-Stokes equations in fluid domain with Dirichlet boundary conditions \( T^k \) at the CHT interface.
2. Evaluate the heat flux distribution at the CHT interface from the fluid solution.
3. Solve the heat conduction equation in solid domain with the Neumann boundary condition from step 2 at the CHT interface.
4. Evaluate the temperature at the CHT interface \( T_r \) from the conduction solution.
5. Use a relaxation parameter to update the temperature distribution at the CHT interface as follows:
   \[
   T^{k+1} = (1 - \omega)T^k + \omega T_r
   \]
6. Use the new temperature distribution as Dirichlet boundary condition to solve the Navier-Stokes equations in fluid domain.
7. Repeat steps 2 to 6 until convergence is achieved.

Imlay et al. [19] have tried two different loose-coupling procedures to couple the Navier-Stokes and heat conduction modules at each time step. The first procedure is exactly the same as Croce’s approach, but without under-relaxation in Imlay’s application. This approach led to wild oscillations and instabilities of wall temperatures. In his second procedure, the instability of the coupling was damped by imposing a Robin boundary condition to the structure conduction module instead of the Neumann boundary condition. The reference temperatures \( T_{ref} \) are the stagnation temperatures of the fluid at first nodes away from the wall. Then, local heat transfer coefficients can be evaluated from equation (2.7).

\[
q_w = h(T_w - T_{ref})
\]

This coupling procedure is described as follow:

1. Solve Navier-Stoke equations to update the heat flux distribution at the fluid-solid interface with a given temperature distribution at the interface.
2. Calculate the reference temperature distribution \( T_{\text{ref}} \).

3. Calculate the heat transfer coefficient distribution \( (h) \) using equation (2.7).

4. Pass the heat transfer coefficient distribution and the reference temperature distribution at the interface from the fluid grid to the structure grid.

5. Solve the heat conduction code to update the temperature distribution \( (T_w) \) at the fluid-solid interface.

6. Pass the temperature distribution \( (T_w) \) at the interface from the structure grid to the fluid grid.

7. Repeat step 1 to 6 until a convergence is achieved.

In the coupling procedure above, the reference temperature distribution is determined first and then is used to calculate the heat transfer coefficient distribution. However, Montenay et al. [20] use an opposite way to obtain the Robin boundary condition. In their procedure, a constant heat transfer coefficient \( \alpha \) is chosen first at the CHT interface and is fixed in all coupling steps. The reference temperature distribution at the CHT interface will be updated in each CHT iteration based on the fixed heat transfer coefficient distribution and the solution of fluid domain as shown in equation (2.8).

\[
T_{\text{ref}}^n = T_w^{f,n} - \frac{q_w^{f,n}}{\alpha} \tag{2.8}
\]

In this procedure, the Robin boundary condition is considered as a kind of relaxation to the solid conduction calculation. The heat flux distribution at the CHT interface can be post-evaluated from the temperature distribution of the conduction solution as follows:

\[
q_w^{s,n+1} = \alpha(T_w^{s,n+1} - T_{\text{ref}}^n) \tag{2.9}
\]

Combining the above two equations, one can get:

\[
q_w^{s,n+1} = q_w^{f,n} + \alpha(T_w^{f,n+1} - T_w^{f,n}) \tag{2.10}
\]

which can also be expressed as:

\[
T_w^{f,n+1} = T_w^{f,n} + \frac{(q_w^{s,n+1} - q_w^{f,n})}{\alpha} \tag{2.11}
\]
This equation indicates that the big heat transfer coefficient \( \alpha \) will relax the temperature variations from last iteration. Therefore, the higher heat transfer coefficient \( \alpha \) will increase the stability of the CHT coupling and reduce convergence speed.

Kassab et al. used a loose-coupling approach to couple the NASA-Glenn turbomachinery FVM Navier-Stokes solver and a BEM steady state heat conduction code for solving the CHT problem of film-cooled turbine blades. In this coupling approach, a Neumann boundary condition is imposed for solid conduction calculation and a Dirichlet boundary condition is imposed for Navier-Stokes calculation, which is opposite to the suggestion of Giles' study [13]. Hence, very small time marching interval was used for the coupling between the fluid and solid solvers in their simulations in order to stabilize the CHT calculation. Their procedure can be described as follows:

1. Solve the fluid domain by the FVM Navier-Stokes solver with an initial adiabatic boundary condition at the CHT interface to get a temperature distribution at the CHT interface.
2. Impose the temperature distribution at the CHT interface to the BEM conduction solver to evaluate the heat flux distribution at the CHT interface.
3. Relax the heat flux distribution before transferring them to the FVM solver.
   \[
   q = \beta q_{\text{old}}^{\text{BEM}} + (1 - \beta)q_{\text{new}}^{\text{BEM}}
   \]  
   where, \( \beta \) is a relaxation parameter and a greater value of \( \beta \) corresponds to a higher relaxation. 0.2 is used in their application.
4. Impose the heat flux distribution to the FVM Navier-Stokes solver to calculate the temperature distribution at the CHT interface.
5. Repeat steps 2 to 4 until the convergence of temperature and heat flux distributions are obtained.

### 2.2 Overview of Numerical Simulations for Aircraft Hot-Air Anti-Icing Systems

From the above discussions, the implicit coupling approach is a robust and stable method for the application of unsteady problems or when time accuracy is critical. However, it is
difficult to implement for complicated applications with more than two domains, such as anti-icing simulation, where droplet impingement and water film/ice accretion calculations need to be included into the coupling as well. To simultaneously solve so many domains including a large set of equations is daunting. Furthermore, the mismatch in the structure of the coefficient matrices is a great difficulty if different types of solvers, such as FEM, FVM, or BEM, are used for different computational domains. The loose-coupling approach is the most practical and simplest method for complicated CHT problems because it only increases the complexity of the coupling itself, instead of changing any detail of the numerical solvers. It is also very flexible to replace solvers for each domain with minimum modifications of the coupling code. Therefore, the loose-coupling approach is strongly suggested for aircraft hot-air anti-icing simulations. However, only limited literature is available for fully CFD-based simulations of entire hot-air anti-icing processes.

2.2.1 Hot-Air Anti-Icing Simulation in LEWICE

LEWICE is an ice accretion prediction code developed by NASA Glenn Research Center. It also has the capability to analyze the performance of electro-thermal deicers or hot-air anti-icing systems based on the numerical model of Al-Khalil [22]. It consists of four modules: the flow field calculation, the particle trajectory and impingement calculation, the thermodynamic and ice growth calculation, and the modification of the geometry due to the ice accretion [8]. The airflow solver in LEWICE is based on a Douglas 2D potential flow panel method program developed by Hess and Smith [23] for an inviscid velocity field, and an integral boundary layer calculation for viscous effects in the boundary layer. Recently, a 3D version of the panel method has been developed to enable LEWICE to calculate the airflow in 3D [8]. The droplet impingement calculation in LEWICE is based on particle tracking techniques, in which a particle is launched from the free stream and is dragged by the airflow until hitting or not the body. These techniques are very efficient on 2D and simple geometries, but become computationally expensive in 3D and encounter major difficulties on complex geometries, especially when the appropriate launch area may not be known a priori. The thermodynamic analysis of
the icing surface is based on the Messinger’s thermodynamic model [24], which includes the equations of mass and energy balances in a control volume.

The mass balance equation evaluates the amount of the water mass entering and leaving the control volume and can be expressed as [8]:

\[
\dot{m}_{\text{imp}} + \dot{m}_{\text{in}} = \dot{m}_{\text{evap}} + \dot{m}_{\text{f}} + \dot{m}_{\text{out}} + \dot{m}_{\text{sh}} + \dot{m}_{\text{st}}
\]

The energy balance equation, obeying the first law of thermodynamics, evaluates the amount of the heat entering and leaving from the control volume and can be expressed as [8]:

\[
-k \frac{\partial T}{\partial \varphi}_{\varphi=0} = q_{\text{nc}} + q_{\text{evap}} + q_{\text{ke}} + q_{\text{lat}} + q_{\text{sens}}
\]

In LEWICE hot-air anti-icing simulations, a 1D heat conduction in the normal direction through the surface is assumed. This assumption states that the heat conduction through a solid surface is primarily 1D if anti-icing processes have a 3D internal hot airflow, and a primarily 2D external airflow and water collection. In their anti-icing model, the heat transfer coefficient on the solid internal surface was determined based on empirical correlations rather than directly solving the 3D internal jet flow from the CFD solver. The empirical correlations were designed to evaluate the local and average Nusselt numbers for arrays of circular holes impinging normally on a flat plate, which has a vaguely similar geometry to the piccolo tube of aircraft anti-icing system. The local and average Nusselt numbers can be evaluated and are primarily a function of the jet Reynolds number, the jet spacing, and the distance to the wall. The Reynolds number for circular cross-section holes can be calculated by using the mass flow rate per unit span as:

\[
\text{Re} = \frac{4 \dot{m}}{\pi \dot{N}_r \mu d}
\]
Different correlations have different ranges of application. A correlation developed by Goldstein et al. is used in LEWICE to evaluate the average Nusselt number of the piccolo tube impinging jet [25].

\[
\overline{Nu} = Re^{0.76} \frac{24 - \left( \frac{z_n}{d} - 7.75 \right)}{533 + 44 \left( \frac{r}{d} \right)^{1.285}}
\]  

(2.16)

\[
\overline{Nu_d} = Re^{0.76} \frac{24 - \left( \frac{z_n}{d} - 7.75 \right)}{533 + 44 \left( \frac{r}{d} \right)^{1.594}}
\]  

(2.17)

The application ranges of this correlation are:

\begin{align*}
61,000 & \leq Re \leq 124,000 \\
0.5 & \leq r/d \leq 32 \\
6 & \leq z/d \leq 12
\end{align*}

Wright [25] has implemented a numerical anti-icing simulation using LEWICE to a wing model with a piccolo tube system. This wing model has an average chord length of 1.52m with a 8° sweep and has a similar airfoil profile as the NACA23014 model. The surface temperatures and residual ice shapes have been compared between the Icing Research Tunnel experimental results and two numerical test cases with different ambient temperatures. The results show that the empirical correlation over-predicts the average surface temperature by about 20% (5.1°C difference) in one test case and under-predicts the average surface temperature by about 10% (4.3 °C difference) in another test case. The deviation of surface temperatures consequently results in the differences of residual icing shapes and locations from experimental results. The percent difference of temperatures is defined by the following equation:

\[
\% \text{diff} = \frac{|T_{\text{exp}} - T_{\text{LEWICE}}|}{T_{\text{exp, max}} - T_{\text{m}}}
\]

The major deficit of anti-icing simulations in LEWICE is that the correlation formula is designed just for predicting the heat transfer coefficients on the impinging surface and
may fail for other locations such as the inner liner, which is located at downstream of the tube to create the crossflow for increasing the flow velocity and heat transfer. However, this correlation model can still allow users to perform a fast preliminary analysis for an anti-icing system design. This is great hot-air anti-icing test case, but unfortunately it cannot be used to validate the numerical code developed in this thesis because of the deficient geometry data of the 3D wing model and the completely construction inside the wing.

2.2.2 Hot-Air Anti-Icing Simulation in CHT2D

The Advanced Aerodynamics group of Bombardier Aerospace has developed a 2D hot-air anti-icing numerical simulation tool named CHT2D [26], which couples two numerical modules: the ice prediction code CANICE [27][28] and the Navier-Stokes airflow solver NSU2D [29][30].

![Figure 2.2 CHT2D Anti-Icing Algorithm (From Reference [26])](image)

CANICE is an ice accretion simulation code, which can solve the external airflow, water droplets trajectories, the thermal balance in runback water region, and the heat conduction in the solid region, in almost the same way as LEWICE. The external airflow is calculated by a panel method and the integral boundary layer equations. The droplet impingement is solved by a Lagrangian method. In the runback water region, the mass
and energy conservative equations are solved to evaluate the ice accretion. The following
two equations and Figure 2.3 illustrate the energy balance in the runback water region.

\[ \dot{m}_{in} + \dot{m}_{imp} = \dot{m}_{ice} + \dot{m}_{evap} + \dot{m}_{out} \quad (2.18) \]

\[ Q_w = -\dot{m}_{ice} (c_w T_s - L_f) - \dot{H}_{evap} - \dot{m}_{out} c_w T_h + \dot{H}_{imp} + Q_{conv} + \dot{H}_{in} \quad (2.19) \]

![Figure 2.3 Water Film Region (From Reference [32])](image)

CANICE can also simulate the heat conduction in solid region by solving a 2D steady-state conduction equation as follow:

\[ \frac{\partial}{\partial x} \left( k_x \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k_y \frac{\partial T}{\partial y} \right) = 0 \quad (2.20) \]

The detailed description of the CANICE code can be found in [27][28]. The loose-coupling procedure in CHT2D is illustrated in Figure 2.2. CHT2D starts with the external airflow and droplet impingement calculations with CANICE, and the computation of NSU2D to solve the internal airflow with an initial temperature distribution on the internal wall. With the convective heat flux distributions from the internal and external airflow solutions, CANICE will start the iterative boundary condition exchange between the water film energy balance module and the solid conduction module until a converged local temperature distribution on the internal wall is obtained. Thereafter, this converged local temperature distribution will be used as the boundary condition to solve the internal airflow with NSU2D again in order to update the local heat flux on the internal wall and
give it back to CANICE. CHT2D will repeat the two levels of iterations until the maximum surface temperature variation is lower than 0.1K.

Morency et al. [31][32] had performed an anti-icing analysis in CANICE but in their simulation, the heat flux or heat transfer coefficient distributions had to be imposed on the internal surface of the solid region and the internal jet flow was not solved. Pueyo et al. [26] have carried out a 2D piccolo tube anti-icing simulation to a wing section of the Bombardier CRJ700 based on fully CFD-based calculations with CHT2D. Three test cases with dry air conditions has been simulated and compared with wind tunnel experimental results of the temperature distribution on the external wall. A maximum temperature difference of 7°C at the leading edge shows a good agreement between CHT2D simulations and experimental results. Another test case with icing condition was also implemented to show the influence to the ice accumulation by different anti-icing heat loads. CHT2D successfully simulated the full range of the anti-icing cases at the protected region and captured the runback water freezing due to incomplete evaporation. However, no experimental results for this test case are available for validation.

The above-mentioned CHT2D code is completely different from the CHT3D code developed in the present thesis: they do not have any connection except the similarity of their names.

2.3 Introduction of A Simplified Anti-Icing Experiment

The Laboratoire de Mécanique des Fluides (LMF) of Université Laval has built an experimental assembly for heat transfer research of aircraft hot-air anti-icing systems [33]. The purpose of their project is to create a database, which can be used to validate anti-icing numerical codes. This experiment is the only one available for validating the hot-air anti-icing simulation code CHT3D developed in this thesis because Laval University provided detailed information about the experiment, while the detail of any other hot-air anti-icing experiment is unobtainable due to either the complexity of implementing the entire hot-air anti-icing experiment or the confidentiality of geometrical or experimental data.
This anti-icing system has a hot-air jet flow with 2D properties in order to simplify the experiment and facilitate the numerical simulation. This experiment assembly is based on a wind tunnel with a jet generation device below it, and is shown in the following figure.

![Figure 2.4 The Construction of the Laval University Anti-Icing Experimental Assembly](From Reference [33])

The test section of the wind tunnel has an inlet with size 0.61m by 0.46m. The pressure gradient on the tunnel floor is similar to the one on the leading edge of a NACA2412 airfoil with a chord length of 2.5m and an angle of attack of 18 degrees. This pressure gradient is created by the convergent and divergent shapes of the wind tunnel ceiling. The wind tunnel floor is comprised of many pieces of aluminum flat plates and one with a bump surface that can be seen on the figure above. This curve plate is used to simulate the similar pressure coefficients at an airfoil leading edge. An aluminum flat plate with 150mm in length, 400mm in width and 6.35mm in thickness behind the curve plate is just located between the wind tunnel airflow and the impinging jet flow. It is isolated from other parts of the wind tunnel floor by adiabatic materials in order to guarantee that the heat conduction will not be able to transfer to other parts of wind tunnel floor. The
aluminum plate is extended from x/c=0.01 to x/c=0.07 in the flow direction and is equipped with 11 evenly spaced heat flux gauges. The x is measured from the kink on the wind tunnel floor and c=2.5m is the chord length of the NACA2412 airfoil. This design makes the experiment much easier to be implemented than a prototype of a real aircraft wing. Moreover, to install the measurement equipment on a planar surface is also much easier than on the curved surface of the wing leading edge.

A slot jet generation device with a rectangular box is installed and fixed just below the test aluminum plate. The position of jet inlet is 6.35mm lower than the lower surface of the aluminum plate, which is 4 times the width of the slot jet. The jet is a fully developed laminar flow at the inlet with a mass flow of 0.004736 kg/s. The jet center is aligned with the middle point of the aluminum plate located at x/c=0.04.

No water droplets are launched in this experiment, and thus only dry air results are available. Contrary to most aircraft anti-icing systems, the air at jet nozzle is not choked and the flow is essentially laminar. This experiment lasted two hours in order to establish the steady state of surface temperatures.

2.4 Review of Heat Transfer for A Single Slot Jet Impingement

Impinging jets have been widely used in engineering applications, such as aircraft anti-icing systems, turbine blade cooling, and electronic components cooling, due to their high heat/mass transfer rates. A typical device for the aircraft anti-icing system is the piccolo tube, which impinges the anti-icing hot air onto the internal surfaces of protected regions to prevent the ice buildup on the external surfaces. It’s very important to understand the flow and heat transfer characteristics of impinging jets before performing the numerical simulation of anti-icing systems.

Figure 2.5 illustrates a single jet flow developing from the nozzle exit with slot width W or diameter D and impinging on a flat plate surface. This flow pattern can commonly be divided into three characteristic regions: the free jet region, the stagnation or impinging region, and the wall jet region.
The free jet region, in which the jet flow is not affected by the impinging surface, can be divided into a potential core and a fully developed sub-region. With the increasing distance from the jet exit, the jet width becomes linearly broader and the velocity within the fully developed sub-region becomes smaller due to the momentum exchange between the jet and the surrounding air. The potential core, within which the nozzle exit velocities are retained, is diminishing in width with the jet growth.

In the impinging region, which starts relatively close to the impinging surface, the flow is decelerated in the vertical direction and transformed to the acceleration in the horizontal direction due to the influence of the impinging surface.

In the wall region, the flow starts to transform from the horizontal acceleration to the deceleration because of the shear stress from both the wall and the surrounding quiescent air.
2.5 Conclusions

As have been discussed above, many algorithms have been developed to solve conjugate heat transfer problems, which are generally divided into tight-coupling and loose-coupling methods. But most of them are developed just for solving dry air heat transfer problems and only very few numerical codes are able to simulate the hot-air anti-icing processes. As the complexity of the hot-air anti-icing heat transfer processes, the loose-coupling method is always suggested to couple different domains based on an explicit boundary condition exchange. The objective of this thesis is to develop a CFD-based conjugate heat transfer procedure, with loose-coupling, to simulate the 3D hot-air anti-icing processes including a 3D external fluid flow, a 3D droplet impingement, a 3D ice accretion, a 3D heat conduction, and a 3D internal jet flow. It would be able to simulate a full range of anti-icing cases and to assist in designing efficient anti-icing systems and in certifications.
CHAPTER 3: Mathematical Equations and Numerical Methods

The anti-icing simulation procedure developed in this thesis is based on the use of some elements of a second-generation 3D in-flight icing state-of-the-art package, FENSAP-ICE, to iteratively solve the fully 3D viscous fluid flow, the 3D droplet impingement, the 3D ice accretion, and the 3D heat conduction. This chapter will give a brief introduction of the mathematical models and numerical methods related to the CHT3D anti-icing simulations.

3.1 Airflow Model (FENSAP)

FENSAP is a Finite Element Navier-Stokes Analysis Package designed to solve the steady or unsteady, incompressible or compressible, inviscid or viscous, laminar or turbulent Newtonian fluid flows by solving the Navier-Stokes equations, the most general mathematical expression of fluid flows. Accurate evaluation of the convective heat fluxes at the fluid/structure interfaces from airflow calculations is essential for the anti-icing conjugate heat transfer simulations. The convective heat fluxes are strongly related to the turbulence models, the rough wall treatment for wet surfaces, and the method of calculating the convective heat fluxes in numerical simulations. Hence, more attention will be paid to these issues.

3.1.1 Governing Equations

A control volume analysis is used to establish the partial differential governing equations.

**Continuity equation:**

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0 \]  \hspace{1cm} (3.1)

**Momentum equations:** (for Newtonian viscous fluids ignoring the body forces)
\[
\begin{align*}
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} &= -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \\
(3.2)
\end{align*}
\]

where, \( \tau_{ij} \) is the viscous shear stress tensor

\[
\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right)
\]

\( \delta_{ij} \) is the Kronecker delta, i.e., \( \delta_{ij} = 1 \) if \( i=j \) and \( \delta_{ij} = 0 \) if \( i \neq j \).

Energy equation:

The equation of energy conservation follows from the first law of thermodynamics and states that the total energy of the system has to be conserved.

\[
\begin{align*}
\frac{\partial (\rho H - p)}{\partial t} + \frac{\partial (\rho u_j H)}{\partial x_j} &= \frac{\partial}{\partial x_j} \left( k \frac{\partial T}{\partial x_j} \right) + \frac{\partial (u_i \tau_{ij})}{\partial x_j} \\
(3.3)
\end{align*}
\]

The left hand side is the change of the total energy in a control volume. The first term in the right hand side is the heat transfer to the control volume and the second term is the work done by viscous forces.

For 3D problems, the above 5 governing equations have 9 unknowns, which are \( \rho, u_i, u_j, u_k, p, T, \mu, k, H \), hence 4 more equations are required to close the system as follows.

\( \mu \) is the dynamic viscosity of the fluid. For a viscous laminar flow, it can be calculated from Sutherland's law as:

\[
\frac{\mu}{\mu_\infty} = \left( \frac{T}{T_\infty} \right)^{3/2} \frac{T_\infty + 110}{T + 110} \\
(3.4)
\]

\( k \) is the thermal conductivity of the fluid and can also be calculated by Sutherland's law as:

\[
\frac{k}{k_\infty} = \left( \frac{T}{T_\infty} \right)^{3/2} \frac{T_\infty + 133.7}{T + 133.7} \\
(3.5)
\]
The equation of state for an ideal gas:

\[ p = \rho RT \]  

(3.6)

The definition of the total enthalpy:

\[ H = \frac{1}{2} \| \vec{\nabla} \|^2 + \frac{\gamma}{\gamma - 1} \frac{p}{\rho} \]  

(3.7)

In FENSAP, the governing equations are non-dimensionalized by introducing the following dimensionless variables.

\[ x_i^* = \frac{x_i}{l_\infty}, \quad u_i^* = \frac{u_i}{U_\infty}, \quad \rho^* = \frac{\rho}{\rho_\infty}, \quad p^* = \frac{p}{\rho_\infty U_\infty^2}, \quad T^* = \frac{T}{T_\infty}, \]

\[ \mu^* = \frac{\mu}{\mu_\infty}, \quad k^* = \frac{k}{k_\infty}, \quad c_p^* = \frac{c_p}{c_{p,\infty}}, \quad H^* = \frac{H}{H_\infty}, \quad t = \frac{t U_\infty}{l_\infty} \]

Omitting the star notation, the Navier-Stokes equations can then be rewritten in non-dimensional form as follows:

**Continuity equation:**

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \]  

(3.8)

**Momentum equations:**

\[ \frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = - \frac{\partial p}{\partial x_i} + \frac{1}{Re_\infty} \frac{\partial \tau_{ij}}{\partial x_j} \]  

(3.9)

**Energy Equation:**

\[ \frac{\rho c_p \partial T}{\partial t} + \frac{\partial (\rho c_p u_i T)}{\partial x_j} = (\gamma - 1) \frac{M_\infty^2 D_p}{D_{t}} + \frac{1}{Re_\infty Pr_\infty} \frac{\partial}{\partial x_j} \left( k \frac{\partial T}{\partial x_j} \right) + \frac{(\gamma - 1) M_\infty^2}{Re_\infty} \frac{\partial (u_i \tau_{ij})}{\partial u_j} \]  

(3.10)

### 3.1.2 Numerical Discretization

In FENSAP, the Navier-Stokes equations including the continuity equation, momentum equations, and energy equation are solved using the weak-Galerkin finite element discretization scheme. The weak-Galerkin finite element discretization is obtained by integrating the products of the governing equations in non-dimensional form and the
weight functions $W_i$, which are generally chosen as the shape functions $N_i$. In FENSAP solution processes, the continuity equation and momentum equations are first solved in a coupled way using the weak-Galerkin finite element method to calculate the incremental changes of the primitive variables $\Delta p$, $\Delta u_1$, $\Delta u_2$, $\Delta u_3$. The total enthalpy is then solved from the energy equation using the weak-Galerkin finite element method in term of the incremental change $\Delta H$. Finally, the variables $T$, $\rho$, $\mu$, $k$ are solved from the definition of total enthalpy, the gas equation of state, and Sutherland’s laws.

The linear system of equations is solved using an iterative matrix solver based on the generalized minimum residual (GMRES) method. For anti-icing simulations in this thesis, both external and internal airflows are considered as steady-state problems. In order to improve the performance of the iterative matrix solver, the steady-state problems are solved with a local time stepping approach in which the calculation starts from the unsteady-state and advances in time until the steady state of fluid flow is obtained. The choice of the local time step $\Delta t$ is based on the stability analysis of the explicit-Euler centered finite difference scheme, and can be expressed as:

$$\Delta t = CFL \cdot \Delta t_{stab}$$

$\Delta t_{stab}$ is the maximum theoretical time step from the stability analysis and varies according to the element size. Hence, $CFL \leq 1$ is required for system stability.

In order to overcome the numerical instabilities, artificial viscosity terms are added to the right hand sides of the continuity, momentum, and energy equations. Different artificial viscosity schemes have been implemented in FENSAP, such as the Streamline Upwind (SU) scheme, first order and second order schemes [34].

The fluid energy equation with the weak-Galerkin finite element discretization can be written as:

$$\iint W_i \frac{1}{Re_{\infty} Pr_{\infty} (\gamma_{\infty} - 1) M_{\infty}^2} \left( k_{eff} \frac{\partial T}{\partial x_i} \right) dS = \quad (3.11)$$
The thermal boundary conditions have to be defined in the boundary surface integral term in this discretized energy equation, which is the source term in the finite element computation. Hence, the thermal boundary conditions on the wall can be either the wall temperatures $T_w$ or the first derivative of wall temperatures, the convective heat fluxes, 

$$q_w = k_{\text{eff}} \frac{\partial T}{\partial x}.$$  

The energy equation calculates temperatures in the entire domain.

### 3.1.3 Turbulence Models

In FENSAP, the effect of turbulence on the mean flow is modeled using eddy viscosity models based on Boussinesq's assumption. The Boussinesq assumption states that the turbulent stress tensor is in the same direction as the laminar stress tensor, thus, all turbulence effects can be represented by an effective viscosity, i.e.

$$\mu_{\text{effective}} = \mu_{\text{Laminar}} + \mu_{\text{Turbulent}}$$

This effective viscosity is then used in the momentum equations. Similarly, the effective thermal conductivity is defined as:

$$k_{\text{effective}} = k_{\text{Laminar}} + k_{\text{Turbulent}}$$

The turbulent thermal conductivity is estimated from the turbulent viscosity $\mu_{\text{Turbulent}}$ and the turbulent Prandtl number $Pr_t$, which is most commonly taken as a constant value of 0.9 in the literature.

$$k_{\text{Turbulent}} = \frac{c_p \mu_{\text{Turbulent}}}{Pr_t}$$

Various turbulence models have been implemented in FENSAP to determine the turbulent viscosity. They can be divided into one-equation turbulence models such as the Spalart-Allmaras model and two-equation turbulence models such as the $k-\varepsilon$ and $k-\omega$ models. The Spalart-Allmaras and high-Reynolds number $k-\varepsilon$ turbulence models will be used in this thesis.
3.1.3.1 Spalart-Allmaras Turbulence Model

The Spalart-Allmaras model is a one-equation turbulence model. The turbulent kinematic viscosity $\nu_T$ is evaluated from a modified viscosity $\tilde{\nu}$, which is calculated from a non-dimensional transport equation as follows:

$$\frac{\partial \tilde{\nu}}{\partial t} + u_j \frac{\partial \tilde{\nu}}{\partial x_j} = c_{h1} S \tilde{\nu} + \frac{1}{\sigma \text{Re}_\infty} \frac{\partial}{\partial x_i} \left[ (\nu + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_i} + \frac{\partial \tilde{\nu}}{\partial x_k} \frac{\partial \tilde{\nu}}{\partial x_k} \right] - c_{w1} f_w \frac{1}{\text{Re}_\infty} \left( \frac{\tilde{\nu}}{d} \right)^2$$  (3.12)

$d$ is the distance to the nearest wall.

The magnitude of the modified vorticity is:

$$\tilde{S} = S + \frac{1}{\text{Re}_\infty} \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2}$$

The magnitude of the vorticity $S$ is:

$$S = \sqrt{2\Omega_y \Omega_{ij}}$$

$$\Omega_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)$$

The closure functions $f_{v1}$, $f_{v2}$, and $f_w$ are defined as

$$f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3} \quad f_{v2} = \frac{\chi}{1 + \chi f_{v1}} \quad f_w = g \left( 1 + c_{w3} \right)^{\frac{1}{5}}$$

where,

$$\chi = \frac{\tilde{\nu}}{\nu} \quad \text{and} \quad \nu \quad \text{is the laminar viscosity}$$

$$g = r + c_{w2} (r^6 - r) \quad r = \frac{\tilde{\nu}}{\kappa^2 d^2 \text{Re}_\infty S + \tilde{\nu} f_{v2}}$$

The closure coefficients are

$$c_{h1} = 0.1335 \quad c_{h2} = 0.622 \quad c_{v1} = 7.1 \quad \sigma = \frac{2}{3}$$

$$c_{w1} = \frac{c_{h1}}{\kappa^2} \frac{(1 + c_{h2})}{\sigma} \quad c_{w2} = 0.3 \quad c_{w3} = 2 \quad \kappa = 0.41$$

Now the modified viscosity can be solved from the transport equation. Hence, the turbulence kinematic viscosity $\nu_T$ can be determined from the modified viscosity as
3.1.3.2 k-ε Turbulence Model

The k-ε model is a two-equation turbulence model, in which the turbulence viscosity is calculated from the turbulence kinetic energy $k$ and its dissipation rate $\epsilon$ as follows:

$$\mu_\tau = \rho C_\mu \frac{k^2}{\epsilon}$$

where,

$C_\mu = 0.09$

The turbulent kinetic energy $k$ and the dissipation rate $\epsilon$ are solved from their own transport equations as follows:

$$\rho \frac{\partial k}{\partial t} + \rho u_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \rho \epsilon + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_\tau}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$  \hspace{1cm} (3.13)

$$\frac{\partial \epsilon}{\partial t} + \rho u_j \frac{\partial \epsilon}{\partial x_j} = C_1 \frac{\epsilon}{k} \tau_{ij} \frac{\partial u_i}{\partial x_j} - C_2 \rho \frac{k}{\epsilon} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_\tau}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right]$$  \hspace{1cm} (3.14)

where,

$$\tau_{ij} = 2\mu_\tau \left( S_{ij} - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \rho k \delta_{ij}$$

$$S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$

The closure coefficients are:

$C_1 = 1.44, \hspace{1cm} C_2 = 1.92, \hspace{1cm} \sigma_k = 1.0, \hspace{1cm} \sigma_\epsilon = 1.3$

For the high-Reynolds model, $k$ and $\epsilon$ are only solved from the two transport equations above beyond the first layer of elements on the wall. In the first layer of elements at the wall, Reichardt’s velocity profile or wall function is applied as follows:
\[ u^* = \frac{1}{\kappa} \ln(1 + 0.4 y^*) + 7.8 \left[ 1 - \exp\left( -\frac{y^*}{11} \right) - \frac{y^*}{11} \exp\left( -0.33 y^* \right) \right] \]  
\quad (3.15)

where
\[ \kappa = 0.41 \] is the Karman’s constant.

The definitions of \( u^* \) and \( y^* \) are:
\[ u^* = \frac{U}{u_*}, \quad y^* = \frac{\rho y u_*}{\mu} \]

where \( y \) is the normal distance from the nearest wall.

Substituting the above definitions into Reichardt’s velocity profile equation, the friction velocity \( u_* \) can be solved and then used to evaluate the turbulent kinetic energy \( k \) and its dissipation \( \varepsilon \) in the wall elements as follows:
\[ k = \frac{u_*^2}{\sqrt{C_{\mu}}}, \quad \varepsilon = C_{\mu}^{3/4} \frac{k_{3/2}}{k y} \]

Then, the values of \( k \) and \( \varepsilon \) at the first node off the wall are used as the boundary conditions of the \( k-\varepsilon \) turbulence equations.

### 3.1.4 Rough Walls Prediction in Spalart-Allmaras Model

The wall roughness is crucial for accurately predicting the turbulent heat fluxes on a surface with droplets or ice, which has a large influence on the growth, shape, and type of ice accretions. Increasing the wall roughness will increase the convective heat transfer, producing rime ice accretion. In contrast, decreasing the wall roughness will decrease the convective heat transfer, producing glaze ice accretion. A rough-wall treatment [35] has been included in the Spalart-Allmaras turbulence model in FENSAP and has proven successful for icing simulations [36]. For a roughness coefficient \( k_s \), the distance from the wall used in the turbulence model is increased as:
\[ d = d_{\text{min}} + 0.03 k_s \]

where \( d_{\text{min}} \) is the minimum distance from the wall.

As a result, this extension requires a non-zero wall boundary condition for \( \vec{v} \) and a mixed wall boundary condition.
Many factors may affect the roughness coefficient with complex physical aspects, such as the velocity, Liquid Water Content (LWC), temperature, droplet diameter, body geometry, and static pressure. An empirical correlation is used in FENSAP to evaluate the equivalent sand grain roughness height $k_s$, which is developed for LEWICE and is specifically valid for ice accretion [8]. It takes into account the effect of velocity, LWC, and static temperature and is calculated from the following correlation.

$$k_s = \left[ \frac{k_s}{c} \right]_{V\infty} \left[ \frac{k_s}{c} \right]_{LWC} \left[ \frac{k_s}{c} \right]_{T\infty} \left( \frac{k_s}{c} \right)_{base} c$$

(3.16)

where

Velocity

$$\left[ \frac{k_s}{c} \right]_{V\infty} = 0.4286 + 0.0044139(V_{\infty})$$

Liquid water content

$$\left[ \frac{k_s}{c} \right]_{LWC} = 0.5714 + 0.2457(LWC) + 1.2571(LWC)^2$$

Static temperature

$$\left[ \frac{k_s}{c} \right]_{T\infty} = 46.8384\left( \frac{T_s}{1000} \right) - 11.2037$$

Baseline value

$$(k_s/c)_{base} = 0.00117$$

and $c$ is the characteristic length.

3.1.5 Convective Heat Flux

In the classical approach, the surface heat fluxes are calculated from the wall temperature gradients. This is, however, a first-order accurate method and needs a very fine mesh in order to produce an accurate result. A consistent Galerkin finite element approach developed by Gresho et al. [37] has been implemented in FENSAP to evaluate more
accurate surface heat fluxes. This method is used to predict all the convective heat fluxes in this thesis. The Gresho heat fluxes are second-order accurate and are post-processed from FENSAP solutions. When the temperatures at the wall are imposed as Dirichlet boundary conditions, FENSAP evaluates the temperature at all nodes by solving the weak-Galerkin discretized fluid energy equation as follows:

\[
\oint_{\partial S} \left[ \frac{1}{\text{Re}_{\infty} \text{Pr}_{\infty} (\gamma_{\infty} - 1) M_{\infty}^2} \left( k_{\text{eff}} \frac{\partial T}{\partial x_i} \right) \right] dS = 0
\]

(3.17)

\[
\iiint_{\Omega} \left[ W_i \left( \frac{\rho c_p}{\gamma_{\infty} - 1} M_{\infty}^2 \frac{\partial T}{\partial t} - \frac{D\rho}{Dt} - \frac{1}{\text{Re}_{\infty}} \phi \right) + \frac{\partial W_i}{\partial x_i} \left( \frac{1}{\text{Re}_{\infty} \text{Pr}_{\infty} (\gamma_{\infty} - 1) M_{\infty}^2} \frac{\partial}{\partial x_i} \left( k_{\text{eff}} \frac{\partial T}{\partial x_i} \right) \right) \right] dV
\]

After obtaining a FENSAP solution, the heat flux at the wall can be evaluated in a reverse method with the same weak-Galerkin discretized fluid energy equation but replacing the temperature gradient term in the surface integral by the heat flux as follows:

\[
\oint_{\partial S} \left[ \frac{1}{\text{Re}_{\infty} \text{Pr}_{\infty} (\gamma_{\infty} - 1) M_{\infty}^2} q \right] dS = 0
\]

(3.18)

\[
\iiint_{\Omega} \left[ W_i \left( \frac{\rho c_p}{\gamma_{\infty} - 1} M_{\infty}^2 \frac{\partial T}{\partial t} - \frac{D\rho}{Dt} - \frac{1}{\text{Re}_{\infty}} \phi \right) + \frac{\partial W_i}{\partial x_i} \left( \frac{1}{\text{Re}_{\infty} \text{Pr}_{\infty} (\gamma_{\infty} - 1) M_{\infty}^2} \frac{\partial}{\partial x_i} \left( k_{\text{eff}} \frac{\partial T}{\partial x_i} \right) \right) \right] dV
\]

In this equation, the heat fluxes \( q \) at surface nodes are unknown and are solved for. The volume integral becomes a source term because the temperatures have been solved for by FENSAP, and are now imposed as boundary conditions. The variable \( q \) is only discretized for the faces at the wall, using 2D shape functions:

\[
q = \sum_{j=1}^{m} N_j q_j
\]

3.2 Droplets Impingement Model (DROP3D)

Supercooled impinging droplets are the cause of in-flight ice accretions. Hence, an accurate prediction for droplet impingement is crucial for icing/anti-icing/de-icing
simulations. DROP3D uses a fully 3D Eulerian approach, developed by Bourgault et al. [38] for airflows containing water droplets, as an alternative to the traditional Lagrangian particle tracking approach for which determining the launching areas for complex 3D geometries is cumbersome. It computes the droplet velocities and volume fractions of water at the nodes of the entire computational domain where the airflow variables are known from airflow solutions. The droplet volume fraction $\alpha(x,t)$ is the mean values of the ratio of the volume occupied by water over the total volume of the fluid element. Based on the fact that the icing droplets have a diameter less than $100\mu m$ and a volume fraction around $10^{-6}$, the following assumptions have been used in the DROP3D mathematical model [38].

- Droplets are spherical without any deformation or breaking;
- No droplet collision, coalescence or splashing;
- No heat and mass transfer between the droplets and the surrounding air;
- Neglect turbulence effects on droplets;
- The only forces acting on the droplets are drag, gravity, and buoyancy.

The following non-dimensional droplet continuity and momentum partial differential equations based on the above assumptions are used to solve the droplet impingement in DROP3D:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \bar{u}_d) = 0 \quad (3.19)$$

$$\frac{\partial \bar{u}_d}{\partial t} + \bar{u}_d \cdot \nabla \bar{u}_d = \frac{C_D \text{Re}_d}{24K} (\bar{u}_a - \bar{u}_d) + \left(1 - \frac{\rho_d}{\rho_a}\right)\frac{1}{Fr^2}g + \frac{\rho_a}{\rho} \left(\frac{\partial \bar{u}_a}{\partial t} + \bar{u}_a \cdot \nabla \bar{u}_a\right) \quad (3.20)$$

where

- $\bar{u}_a$ Non-dimensional velocity of air
- $\bar{u}_d$ Non-dimensional velocity of droplets
- $\rho$ Density of water
- $\rho_a$ Density of air
- $d$ Droplets diameter
\[ \text{Re}_d \quad \text{Droplet Reynolds number} \quad \text{Re}_d = \frac{\rho d U |\vec{u}_d - \vec{u}|}{\mu} \]

\[ \text{Fr} \quad \text{Froude number} \quad \text{Fr} = \frac{U}{\sqrt{\rho g_0}} \]

\[ K \quad \text{Inertia parameter} \quad K = \frac{\rho d^2 U}{18 l \mu} \]

\[ l \quad \text{Characteristic length} \]

The drag coefficient \( C_D \) for spherical droplets depends on the droplet Reynolds number and is defined as follows:

\[ C_D = \frac{24}{\text{Re}_d} \left(1 + 0.15 \text{Re}_d^{0.687}\right) \text{ for } \text{Re}_d \leq 1300 \]

\[ C_D = 0.4 \quad \text{for } \text{Re}_d > 1300 \]

The local collection efficiency \( \beta \) is introduced to represent the normalized flux of water on the aerodynamic surface. It is an important parameter for ice accretion and is post-processed from droplet solutions as follows:

\[ \beta = -\alpha \vec{u}_d \cdot \vec{n} \]

For a distribution of droplet diameters with \( p_i \) as the percentage of droplets in the \( i^{th} \) class, the overall collection efficiency would be:

\[ \beta = \sum_i p_i \beta_i \]

Then, the water flux at the airfoil surface can be calculated as:

\[ \dot{m}_w = LWC U_w \beta \]

LWC is the Liquid Water Content and is defined as the ratio of the bulk density of the droplets to the bulk density of the air.

A weak-Galerkin finite element method is used to discretize the droplets governing equations in non-conservative form, and the same Navier-Stokes solver and mesh from the airflow computation are used. The streamline upwinding (SUPG) terms are added as stabilization terms to remove possible oscillations in droplet calculations.
The first two assumptions in the above droplet impingement model are only valid for droplets with small sizes. However, for Supercooled Large Droplets (SLD), in which the volume median diameter (MVD) is greater than 50µm, droplet deformation, breakup, and splashing have to be considered. The SLD model has been recently implemented in DROP3D by Honsek et al. [39].

3.3 Water Film/ICE Accretion Thermodynamic Model (ICE3D)

ICE3D developed by Beaugendre et al. [40][41][42] is a fully PDE-based 3D ice accretion model based on the Messenger thermodynamic model [24], in which the conservation of mass and energy in the water film are defined.

In ICE3D, some assumptions are made as follows:

- Small water film thickness, usually less than 10µm;
- Linear velocity profile across the film thickness;
- No air is trapped inside the ice and no air is dissolved in water;
- Shear stress is the main driving force of the water film;
- Constant properties inside the control volume;
- Neglect the conduction between the surface and water;
- Neglect conduction between neighboring control volumes
- Neglect heating of droplets in the air;
- Constant temperature across the water film thickness.

ICE3D requires the solution of the turbulent Navier-Stokes equations and the droplet solution. The physical thermodynamic model can be expressed in the following general form in a water film control volume:

\[ \dot{m}_v + \dot{m}_r = \dot{m}_\beta + \dot{m}_{\text{evap}} + \dot{m}_{\text{ice}} \]  \hspace{1cm} (3.21)

where
The rate of change of liquid water within the control volume is:

$$\dot{m}_v = \frac{\partial}{\partial t} \int \int \rho_w d\tilde{x} dy = \int \int \rho_w \frac{\partial h_f}{\partial t} d\tilde{x}$$

The mass flux of runback water entering and leaving the control volume is:

$$\dot{m}_F = \int \int \rho_w \vec{u}_f (\bar{x}, y, t) \cdot \vec{n} (\bar{x}) dy d\tilde{x} = \frac{\rho_w}{2\mu_w} \int \int div (\tilde{T}_{wall} (\bar{x}, t) h_f^2 (\bar{x}, t)) d\tilde{x} = \int (\rho_w div (u_f h_f)) d\tilde{x}$$

The mass flux of incoming water due to impinging droplets is:

$$\dot{m}_\beta = U_w LWC \int \beta (\bar{x}, t) d\tilde{x} = U_w LWC \int \beta (\bar{x}, t) d\tilde{x}$$

The mass flux of water lost due to evaporation/sublimation is:

$$\dot{m}_{\text{evap}} = - \int \dot{m}_{\text{evap}} (\bar{x}, t) d\tilde{x} = - \int \dot{m}_{\text{evap}} (\bar{x}, t) d\tilde{x}$$

The mass flux of water lost due to ice accretion is:

$$\dot{m}_{\text{ice}} = - \int \dot{m}_{\text{ice}} (\bar{x}, t) d\tilde{x}$$

**Energy Balance Equation:**

$$\dot{Q}_v + \dot{Q}_F = \dot{Q}_\beta + \dot{Q}_{\text{evap}} + \dot{Q}_{\text{ice}} + \dot{Q}_{\text{conv}} + \dot{Q}_{\text{rad}}$$  \hspace{1cm} (3.22)

where

The rate of energy change within the control volume is:

$$\dot{Q}_v = \frac{\partial}{\partial t} \int \left( \int \rho_c c_{p,w} \tilde{T} (\bar{x}, t) dy \right) d\tilde{x} = \int \left( \frac{\partial}{\partial t} \left[ \rho_c c_{p,w} h_f (\bar{x}, t) \tilde{T} (\bar{x}, t) \right] \right) d\tilde{x}$$

The tilde above the T expresses the temperature in degrees Celsius.

The flux of energy lost due to water runback is:

$$\dot{Q}_F = \int \left( \int \rho_c c_{p,w} \tilde{T} (\bar{x}, t) \vec{u}_f (\bar{x}, y, t) \cdot \vec{n} dy \right) d\tilde{x} = \frac{\rho_w}{2\mu_w} \int \int div (h_f^2 \tilde{T}_{wall} c_{p,w} \tilde{T} (\bar{x}, t)) d\tilde{x}$$

The flux of incoming energy due to impinging water is:

$$\dot{Q}_\beta = U_w LWC \beta \left[ c_{p,w} \tilde{T}_{d,\infty} + \left[ \frac{\vec{u}_f^2}{2} \right] \right] d\tilde{x}$$
The flux of energy lost due to evaporation/sublimation is:

\[ \dot{Q}_{\text{evap}} = -k \left[ \frac{1}{2} \left( L_{\text{evap}}(T_e) + L_{\text{sub}}(T_e) \right) + c_{p,w}T_e \right] \] \[
\]}

\[ d\tilde{x} \]

\[ T_e = 273.15K \] is the freezing point of water.

The flux of incoming energy due to ice accretion is:

\[ \dot{Q}_{\text{ice}} = \int \left[ \dot{m}_{\text{ice}}(\tilde{x},t) \left( L_{\text{fus}}(T_e) - c_{p,w}T_e \right) \right] d\tilde{x} \]

The convective heat flux \( \dot{Q}_{\text{conv}} \) is calculated from the airflow solver and is used to evaluate the local heat transfer coefficient as follows:

\[ h_e = \frac{\dot{Q}_{\text{conv}}}{(T - T_w)} \]

The flux of energy lost due to radiation is:

\[ \dot{Q}_{\text{rad}} = \sigma \epsilon \left[ T_e^4 - \left( \tilde{T}(\tilde{x},t) + T_e \right)^4 \right] d\tilde{x} \]

or

\[ \dot{Q}_{\text{rad}} = \sigma \epsilon \left[ T_e^4 - T^4 \right] d\tilde{x} \]

where \( \sigma \) is the Boltzman constant and \( \epsilon \) is the emissivity of the solid.

The above mass and energy equations can be expressed in partial differential form with four compatibility relations as follows:

\[ \rho \frac{\partial h_i}{\partial t} + \text{div}(\bar{u}, h_i) = U_w LWC \beta - \dot{m}_{\text{evap}} - \dot{m}_{\text{ice}} \] \[ (3.23) \]

\[ \rho \frac{\partial h_f c_{p,w} \tilde{T}}{\partial t} + \text{div}(\bar{u}, h_f c_{p,w} \tilde{T}) = \left[ c_{p,w}T_{\text{daw}} + \frac{\|\bar{u}\|^2}{2} \right] \times U_w LWC \beta + \sigma \epsilon (T_e^4 - T^4) \] \[ (3.24) \]
\begin{align}
  h_f & \geq 0 \\
  m_{\text{ice}} & \geq 0 \\
  h_f \bar{T} & \geq 0 \\
  \dot{m}_{\text{ice}} \bar{T} & \leq 0
\end{align} \tag{3.25}

In the thermal model above, three unknowns $h_f, m_{\text{ice}}$, and $\bar{T}$ are solved from two partial differential equations. Hence, four compatibility relations are needed to close the system. These compatibility relations represent the physical observation of water/ice state in different temperature ranges, and divide them into three icing surfaces as follows:

1. Running wet (no ice) region: $\bar{T} > 0$, $\dot{m}_{\text{ice}} = 0$, $h_f \geq 0$
2. Glaze icing region: $\bar{T} = 0$, $\dot{m}_{\text{ice}} > 0$ and $h_f > 0$
3. Rime icing region: $\bar{T} < 0$, $\dot{m}_{\text{ice}} \geq 0$ and $h_f = 0$

The physical result must fall into one of these regions and obey the corresponding compatibility relations. In each region, one variable becomes known and hence, the system is closed. At each node, the calculation will start at one region, which is determined by its temperature from the calculation of previous time step. If the new results obey the compatibility relations of that region, the node will stay in that region, otherwise it will move to another region until a physical result is obtained.

A finite volume numerical method is used to solve the two partial differential equations based on a Roe scheme [43][44][45] for spatial discretization and a finite difference formula is used for time discretization. The two PDEs are solved on the two-dimensional wall surface in a 3D finite element mesh, the same mesh as that used for the airflow calculation.

The temperature on the wall $\bar{T}$ is solved from the energy equation based on the convective heat flux $\dot{Q}_h$ from airflow solutions and an adiabatic wall boundary condition.
3.4 Heat Conduction Model (C3D)

The heat conduction in the solid can be considered as a specific case of the fluid system. The solid is static and incompressible without volume forces and shear stresses. Therefore, the continuity and momentum equations disappear because all the terms go to zero. In the energy equation, by dropping the null terms, which are the viscous heat dissipation, the convective term, and the work done by forces, we get the solid conduction governing equation:

\[ \rho_s c_s \frac{\partial T}{\partial t} = \frac{\partial}{\partial x_i} \left( k_s \frac{\partial T}{\partial x_i} \right) \]  

(3.26)

It is also discretized by the weak-Galerkin finite element method and can be written in the following non-dimensional form as:

\[ \oint W_i \left[ \frac{1}{\text{Re}_w \text{Pr}_w (\gamma_w - 1) M_w^2} \left( k_s \frac{\partial T}{\partial x_i} \right) \right] dS = \]  

(3.27)

\[ \iiint W_i \left[ \frac{\rho_s c_s}{(\gamma_w - 1) M_w^2} \frac{\partial T}{\partial t} \right] + \frac{\partial W_i}{\partial x_k} \left[ \frac{1}{\text{Re}_w \text{Pr}_w (\gamma_w - 1) M_w^2} \frac{\partial}{\partial x_i} \left( k_s \frac{\partial T}{\partial x_i} \right) \right] dV \]

Boundary conditions are needed to make the surface integral term a source term. Either temperatures (Dirichlet BC) or heat fluxes (Neumann BC) at boundary surfaces have to be imposed as boundary conditions. Another possible boundary condition is the Robin boundary condition, in which an artificial parameter, the heat transfer coefficient \( h_e \), and a reference temperature \( T_{\text{ref}} \) are introduced and are imposed as boundary conditions. Then the heat flux in the surface integral term \( k_s \frac{\partial T}{\partial x_i} \) is rewritten as \( h_e (T - T_{\text{ref}}) \). The unknown temperature terms in the surface integral term are combined with the volume integral term and the temperatures are solved for by the finite element method.
CHAPTER 4: Algorithm for Hot-Air Anti-Icing Simulation

4.1 General Introduction of Hot-Air Anti-Icing Simulation

When an aircraft flies through clouds or in icy conditions, super-cooled water droplets at temperatures below the freezing point may impact on aircraft surfaces and result in ice accretion. For some important regions of the aircraft such as wings and nacelles, anti-icing hot air is applied to the internal surfaces of these regions and the thermal energy is conducted through the solid body to prevent ice accretion on the external surfaces. The entire anti-icing heat transfer process for a wing or an engine nacelle can be simply illustrated in the following figure.

A typical hot-air anti-icing model shown in the figure above has four domains that correspond to four heat transfer processes. From left to right, they are:

1. External cold airflow domain
In this domain, airflow around the airfoil at a temperature lower than the freezing point of water tends to freeze the impinging water droplets on the airfoil external surface by means of heat convection. The airflow solution including velocities, heat fluxes, and friction coefficients need to be determined first. The influence to convective heat flux due to the droplet impingement on the aerodynamic surface is taken into account by introducing a surface roughness height based on an equivalent sand grain roughness model, which increases the distance from walls in the turbulence model.

2. Impinging water film domain
   This domain is formed by impinging droplets. The local collection efficiencies of impinging droplets on the airfoil are solved by the droplet impingement model based on the airflow solution. In this domain, the energy balance consists of the external heat convection, the anti-icing heat conduction, the energy advected by impinging droplets, the evaporation, the sublimation, the radiation, the water runback and the ice accretion. The state of the water film in this domain can be used to indicate the anti-icing effect. The appearance of ice and residual water indicates an insufficient anti-icing heat flux; conversely, a clean surface indicates successful, fully evaporative anti-icing. Water runback in this domain is driven by the viscous force evaluated by the external airflow calculation.

3. Solid conduction domain
   The solid domain between the water film and the internal hot airflow is the medium through which the internal hot airflow energy is transferred to the external surface for anti-icing protection by means of heat conduction.

4. Internal hot air domain
   The internal hot air is bled from the engine compressors and impinges on the front internal surface of the airfoil through the holes of piccolo tubes. This internal convective heat flux is the source of the anti-icing thermal energy.
Hypotheses in hot-air anti-icing simulation:

- The airflow around the airfoil, the droplets impingement, the heat conduction, and the jet flow inside the airfoil are all considered at the steady state. Hence, large total times are usually chosen for these numerical simulations, especially for the heat conduction calculation, in order to obtain steady-state solutions. The time-marching steps between the coupled domains do not need to be matched.

- The ice accretion process is an unsteady process and the total simulation time must be specified. In the fully evaporative anti-icing case, it also becomes steady state.

- The boundary movement of the external surface due to ice accretions is ignored in anti-icing simulations because a great deal of ice accretion does not occur when the anti-icing system is engaged.

- The thin film theory makes the following assumptions across the thickness of the water film:
  - Linear velocity profile
  - Constant temperature profile

4.2 Overall Energy Balance in the Hot-Air Anti-Icing System

In any anti-icing thermal model, the overall energy conservation of the entire system has to be guaranteed and the following figure shows all of the thermal energy transfer mechanisms, which are taken into account in this CHT3D model.
Figure 4.2 The Overall Energy Balance in the Hot Anti-Icing System

The loose-coupling conjugate heat transfer approach is applied in the hot-air anti-icing simulation and hence the energy equations in four domains are solved separately and then are coupled through explicit boundary condition exchanges. The heat convection at both the external and internal surfaces of the water/solid system is determined from both the external and internal airflow calculations through a consistent Galerkin FEM post-processing approach developed by Gresho et al. The energy equation of the water film/ice accretion model is modified with an additional conductive heat flux term for anti-icing simulations and can be expressed as:

\[ \dot{Q}_v + \dot{Q}_p = \dot{Q}_\beta + \dot{Q}_\text{rad} + \dot{Q}_\text{evap} + \dot{Q}_\text{ice} + \dot{Q}_\text{conv} + \dot{Q}_\text{cond} \]

This equation indicates that the transfer of the energy in a water film control volume is caused by water film runback, droplet impingement, radiation, evaporation/sublimation, ice accretion, and heat conduction from the solid. The equation in partial differential form is:

\[
\rho_w \left[ \frac{\partial h_f c_{p,w} \bar{T}}{\partial t} + \text{div}(u_j h_f c_{p,w} \bar{T}) \right] = \left[ c_{p,w} \bar{T}_{d,m} + \frac{|| \bar{u}_d ||^2}{2} \right] \times U_w LWC \beta + \sigma \varepsilon (T_m^4 - T^4) \]  \quad (4.1)

\[
-0.5(L_{\text{evap}} + L_{\text{sub}} + c_{p,w} \bar{T})m_{\text{evap}} + (L_{\text{fus}} - c_{p,\text{ice}} \bar{T})m_{\text{ice}} + \dot{Q}_\text{conv} + \dot{Q}_\text{cond}
\]
The heat conduction term is obtained through post-processing of the temperature solution of conduction through the solid.

The energy equilibrium in the solid domain for steady state can be simply stated that the total heat flux entering the solid due to the convective heat flux of the internal jet flow equals to the total heat flux leaving the solid, which is the conductive heat flux from the solid to the water film:

\[ \sum \dot{Q}_{\text{cond}} = \sum \dot{Q}_{\text{conv}} \]

The local temperature in the solid is solved from the heat conduction equation in partial differential form as follows:

\[
\rho_s c_s \frac{\partial T}{\partial t} = \frac{\partial}{\partial x_i} \left( k_s \frac{\partial T}{\partial x_i} \right)
\]  

(4.2)

4.3 Methodology of Hot-Air Anti-Icing Simulation

The anti-icing model with the above four thermodynamic processes is a special conjugate heat transfer problem, as it also requires a droplet impingement solution and energy conservation within the water film. Using the implicit method to solve all of these heat transfer processes simultaneously in the same code is extremely complex. In comparison, the loose-coupling method has great flexibility to deal with more complicated heat transfer problems with any number of domains, because each domain is solved individually with respective solver. The loose-coupling method is independent with regards to the details of each solver and couples the domains by means of exchanging thermal boundary conditions at common interfaces instead of embedding the solid conduction, droplet impingement, and ice accretion models into the airflow solver. In the loose-coupling method, all of the heat transfer processes are solved sequentially and iteratively. Another advantage of using a loose-coupling method is the possibility to directly use the currently available CFD solvers for solving different heat transfer processes in the hot-air anti-icing model. The CHT3D algorithm is based on a FENSAP-ICE package, which includes the airflow solver FENSAP, the droplet impingement solver DROP3D, the ice accretion solver ICE3D and the heat conduction solver C3D.
Figure 4.3 demonstrates the methodology used in the CHT3D code for hot-air anti-icing simulations. Three interfaces have been identified in this figure to show the sequence of thermal couplings between four computational domains. DROP3D calculates the local droplet collection efficiency for ICE3D water film calculation and is updated with the external airflow solution. Actually, DROP3D does not participate in the heat flux boundary condition exchange because it does not solve any energy equation. The fundamental concept of the CHT3D algorithm is to couple energy equations in four computational domains and to enforce thermal continuity by exchanging boundary conditions iteratively until thermal equilibrium of the whole system is achieved. Details about the exchange of boundary conditions at each interface will be discussed later.

4.4 Algorithm of CHT3D Hot-Air Anti-Icing Model

4.4.1 Thermal Boundary Conditions for Each Solver

In order to implement the conjugate heat transfer simulation using the loose-coupling method, we must first clarify the thermal boundary conditions and the output of thermal parameters of each solver.

1. **FENSAP** - airflow solver for both the external and internal airflows
Solves the external convective heat transfer (free-stream air)
Solves the internal convective heat transfer (hot air)

The airflow solver can accept two types of thermal boundary conditions:
(a) Input: the temperature distribution at aerodynamic surfaces, $T_{\text{wall}}$ (Dirichlet B.C.)
    Output: the convective heat flux distribution at aerodynamic surfaces, $Q_{\text{wall}}$
(b) Input: the heat flux distribution at aerodynamic surfaces, $Q_{\text{wall}}$ (Neumann B.C.)
    Output: the temperature distribution at aerodynamic surfaces, $T_{\text{wall}}$

2. **ICE3D** – water film/ice accretion solver
   Input (B.C.):
   The convective heat flux distribution at the external surface of the water film caused by the external airflow
   The conductive heat flux distribution, $Q_{\text{ext}}$ at the external surface of the skin
   Output:
   The temperature distribution at the aerodynamic surface, $T_{\text{ext}}$

3. **C3D** - heat conduction solver
   Solves the conductive heat transfer process in the solid
   Input (B.C.): 3 types
   Dirichlet boundary: the temperature distribution at the wall, $T_{\text{wall}}$
   Neumann boundary: the heat flux distribution at the wall, $Q_{\text{wall}}$
   Robin boundary: the heat transfer coefficient distribution, $h_c$ and the reference temperature, $T_{\text{ref}}$ at the wall
   Output:
   Temperature distribution in the solid
Figure 4.4 shows the detailed coupling procedure of the thermal parameters through three interfaces in the whole hot-air anti-icing simulation. Because different solvers accept different thermal boundary conditions and output different thermal parameters, a method is required to be able to convert between different types of boundary conditions (that is,
temperatures or heat fluxes). In CHT3D, a local heat transfer coefficient $h_c$ and a reference temperature $T_{ref}$ are defined at each interface in order to implement these conversions.

\begin{align*}
h_c &= Q_w / (T_w - T_{ref}) \quad (4.3) \\
Q_w &= h_c (T_w - T_{ref}) \quad (4.4) \\
T_w &= Q_w / h_c + T_{ref} \quad (4.5)
\end{align*}

The reference temperature at each interface is defined at the beginning of a CHT3D calculation and is fixed for all CHT3D calculation cycles. The reference temperature $T_{ref}$ at each interface must be carefully chosen to avoid $T_w = T_{ref}$ during the CHT calculation. The standard choice is either the mainstream airflow temperature or the local wall temperature calculated by the airflow solver based on an adiabatic boundary condition. Equation (4.3) is used to evaluate the heat transfer coefficient distribution from the known temperature $T_w$ and heat flux $Q_w$ distributions at each interface. The calculation of the heat transfer coefficient $h_c$ distribution at each interface is updated in each corresponding iteration. Equations (4.4) and (4.5) are used to convert between the temperature distribution and the heat flux distribution at each interface based on the distribution of the heat transfer coefficient $h_c$.

4.4.3 Boundary Condition Exchange at Each Interface

In Figure 4.4, four computational domains are separated by three interfaces. From top to bottom, they are:

1. Interface 1 between the external cold airflow and the water film.
   This interface is used to couple and exchange boundary conditions between the external airflow solver, FENSAP, and the ice accretion solver, ICE3D. The reference temperature at this interface is chosen as the free stream temperature of
the external cold airflow, $T_{ref1} = T_w$. The external airflow solver, FENSAP, uses
the local wall temperature $T_{wi}$ calculated by ICE3D as the Dirichlet thermal
boundary condition, $T_{w_{-}ext} = T_{wi}$ to compute the local convective heat flux at
interface 1. The local convective heat flux from the FENSAP solution will be used
to calculate the new local heat transfer coefficient from the following equation,
and is used with the reference temperature $T_{ref1}$ as the Robin boundary condition
for ICE3D.

$$h_{c1} = \dot{Q}_{conv} / (T_{w_{-}ext} - T_{ref1})$$ \hspace{1cm} (4.6)

2. Interface 2 between the water film and the solid

This interface is used to couple and exchange boundary conditions between the ice
accretion solver, ICE3D and the solid conduction solver, C3D. The reference
temperature in this interface is also defined as the free stream temperature of the
external cold airflow, $T_{ref2} = T_w$. The local temperature $T_{w_{-}ext}$ on the solid
external surface from the C3D calculation will be converted to a local conductive
heat flux as the Neumann boundary condition of ICE3D using the following
equation.

$$\dot{Q}_{cond} = h_{c2} (T_{w_{-}ext} - T_{ref2})$$ \hspace{1cm} (4.7)

The local heat transfer coefficient $h_{c2}$ in this equation is obtained from the
previous iteration. The local wall temperature $T_{wi}$ calculated by ICE3D is used to
calculate the new local heat transfer coefficient at interface 2 and is used
afterwards with the reference temperature $T_{ref2}$ as the Robin boundary condition
at the solid external surface for the C3D conduction calculation. The new local
heat transfer coefficient is calculated by the following equation:

$$h_{c2} = \dot{Q}_{cond} / (T_{wi} - T_{ref2})$$ \hspace{1cm} (4.8)

3. Interface 3 between the solid and the internal hot airflow
This interface is used to couple and exchange boundary conditions between the solid conduction solver, C3D, and the internal airflow solver, FEANSAP. The local reference temperature \( T_{ref3} \) is the local wall temperature obtained by solving the internal airflow with an adiabatic wall boundary condition. The local temperature on the solid internal surface from the C3D solution is used as the Dirichlet thermal boundary condition for the internal hot airflow calculation. The internal airflow solver, FENSAP, calculates the local convective heat flux at this interface, which is used to calculate a new local heat transfer coefficient by the following equation.

\[
h_{c3} = \frac{\dot{Q}_{\text{conv}}}{(T_{w_{int}} - T_{ref3})}
\]  

(4.9)

This local heat transfer coefficient will be used with the local reference temperature \( T_{ref3} \) as the Robin boundary condition at the solid internal surface for the solid conduction calculation.

### 4.4.4 Implementation of Hot-Air Anti-Icing Simulation in CHT3D

Following the introduction of the coupling methods at each interface, we will discuss how the CHT3D works and couples the entire system to carry out the anti-icing simulation. All discussions will still refer to Figure 4.4. In order to start the CHT3D simulation, initial thermal boundary conditions must be assumed at each interface. Initial wall temperatures are guessed for both interface 1 and interface 3 and an initial conductive heat flux is guessed for interface 2. After that, CHT3D will start the calculation as follows:

1. Solve the internal airflow using FENSAP by imposing an adiabatic boundary condition at interface 3 to calculate the reference temperature distribution at interface 3, \( T_{ref3} \).

2. Solve the external airflow using FENSAP with the boundary condition of the temperature distribution at interface 1 to obtain a heat flux distribution at interface.
3. Evaluate the heat transfer coefficient distribution based on the temperature and heat flux distributions at interface 1 and prepare the Robin boundary condition for ICE3D at interface 1.

4. Solve the internal airflow using FENSAP with the boundary condition of the temperature distribution at interface 3 to obtain a heat flux distribution at interface 3.

5. Evaluate the heat transfer coefficient distribution based on the temperature and heat flux distributions at interface 3 and prepare the Robin boundary condition for C3D at interface 3.

6. ICE3D calculates the temperature distribution at interface 2 with the Robin boundary condition at interface 1 and the conductive heat flux distribution $\dot{Q}_{\text{cond}}$ at interface 2.

7. The heat transfer coefficient distribution at interface 2 is evaluated based on the temperature and heat flux distributions and pass a Robin boundary condition to C3D at interface 2.

8. C3D calculates the temperature distribution in the solid with Robin boundary conditions at interface 2 and interface 3 and evaluates the temperature distributions at interface 2 and interface 3.

9. Convert the temperature distribution at interface 2 to a heat flux distribution and pass it to ICE3D.

10. ICE3D and C3D start an inner loop by iteratively exchanging boundary conditions to each other at interface 2. Steps 6 to 9 are repeated until partial convergence is achieved or until a specific number of iterations of the inner loop have been completed.

11. ICE3D outputs the temperature distribution at interface 1 and passes it to the external airflow solver FENSAP, and C3D outputs the temperature distribution at interface 3 and passes it to the internal airflow solver FENSAP.

12. Both the external and the internal FENSAP calculations are updated based on the updated Dirichlet boundary conditions at interface 1 and interface 3. CHT3D repeats steps 2 to 11 above until the temperature distribution at each interface is converged.
4.4.5 Dual Surface Meshes

In thermal boundary conditions, the temperature is based on finite element nodes, but the heat flux and the heat transfer coefficient are based on surface cells, so the exchange of heat fluxes or heat transfer coefficients at each interface cannot be directly implemented. The heat transfer coefficient can also not be directly calculated from the face-based heat flux and the node-based temperature. A dual surface mesh is created at each interface to convert the values of heat fluxes or heat transfer coefficients from surface cells to finite element nodes before passing them at interfaces or converting them from finite element nodes to surface cells after passing them at interfaces. The dual meshes scheme was developed by Beaugendre and was implemented in the icing accretion simulation [42]. The dual surface mesh is obtained from the surface mesh by connecting each cell centroid to the midpoints of cell edges. The following figure shows an example of the dual meshes on a structured grid.

![Figure 4.5 Dual Meshes on Structured Grid](image)

Four surface cells, named A, B, C, and D, are drawn with solid lines in the figure above and I, J, K, and L are the corresponding centroids of these cells. Dual surface meshes are drawn with dash lines such as the dual surface mesh cell IJKL. The average heat flux or heat transfer coefficient over each cell of the dual surface mesh is evaluated and is then set at the corresponding node of the surface mesh. For example, the average heat flux or
heat transfer coefficient over the dual surface mesh cell IJKL will be set to the node M. The energy conservation has to be guaranteed for each surface cell when converting heat fluxes or heat transfer coefficients from surface cells to nodes. For example, the following relationship is used to calculate the heat flux at node M from the values on surrounding cells A, B, C, and D to ensure that the heat flow rate through the surface IJKL is conservative.

\[ \dot{Q}_M (S_{A4} + S_{B3} + S_{C1} + S_{D2}) = \dot{Q}_A S_{A4} + \dot{Q}_B S_{B3} + \dot{Q}_K S_{C1} + \dot{Q}_L S_{D2} \]

Hence,

\[ \dot{Q}_M = \frac{\dot{Q}_A S_{A4} + \dot{Q}_B S_{B3} + \dot{Q}_K S_{C1} + \dot{Q}_L S_{D2}}{(S_{A4} + S_{B3} + S_{C1} + S_{D2})} \]  

(4.10)

The same method is used to convert the face-based variables in the opposite way from nodes to cells.

### 4.4.6 Boundary Condition Exchange for Matching and Non-Matching Grids

In the current CHT3D code, the transfer of the thermal parameters at the common interfaces between different domains is only applicable for matching grids, which allows the direct transfer of parameters node-to-node and hence is more conservative and accurate than non-matching grids. But the mesh generation for matching grids is sometimes challenging, especially for industrial-level applications with complex 3D geometries, because the grids of different domains have to be generated at the same time and only structured grids can be used to guarantee that the nodes match at the common interfaces.

The non-matching grids are easier to generate because unstructured grids can be used and the grids of different domains can be generated separately. In addition, it is common to user a much finer mesh for a fluid domain than a solid domain. Hence, boundary condition exchange for non-matching grids can increase the flexibility of the CHT application and take advantage of mesh adaptation for fluid calculations, which is more efficient for unstructured meshes. However, it does not improve the quality of the CHT3D coupling procedure.
An interpolation method to transfer aerodynamic loads in a conservative manner at the fluid/solid interface for non-matching nodes has been developed by Lepage [46][47]. His work was originally developed for the purpose of aeroelasticity coupling. However, the method can also be applied to the CHT coupling problem and will soon be implemented into the current CHT3D code in order to increase the capability for complicated applications. In his method, a virtual grid is generated from the union of the fluid grid and the solid grid at the common interface. The virtual grid is initially created based on the fluid grid at the interface and is subdivided into triangular faces. The solid surfaces are then projected onto the fluid surfaces shown in Figure 4.6 (a). Some additional nodes and triangular faces are introduced in the final virtual grid by splitting the fluid edges by solid edges, which is shown in Figure 4.6 (b). The final virtual grid is built in such a way that both the solid and fluid grids at the interface are subsets of the virtual grid and any face of the virtual grid must be located completely inside a fluid face or solid face.

![Virtual Grid Construction](image)

Figure 4.6 Construction of the Virtual Grid for a Square Surface: (A) Fluid and Solid Grids at Fluid/Solid Interface; (B) Final Virtual Grid at Fluid/Solid Interface [47]

After the generation of the virtual grid at the non-matching fluid/solid interface, the variables can be transferred between the fluid grid and the solid grid. When transferring the temperatures, the node-based variables, from the solid grid to the fluid grid at the
interface, the temperatures at the nodes of the solid surface are first transferred to the matching nodes on the virtual grid. Then the temperatures on the other nodes of the virtual grid can be evaluated using linear interpolation over the triangular faces. Finally, the temperatures of the fluid nodes at the interface can be directly transferred from the matching nodes of the virtual grid because the temperatures at all the nodes of the virtual grid have already been evaluated.

However, when transferring the face-based variables, such as the heat fluxes, special attention has to be paid to ensure the transfer is in a conservative form. For transferring the heat flux from the fluid grid to the solid grid at the fluid/solid interface, the face-based heat fluxes at the fluid surface are first transferred to the corresponding faces of the virtual grid. Then, the heat balance across the CHT interface has to be conservative, i.e. the total heat flow though the solid surface must equal to the total heat flow through the fluid surface. It can be expressed as:

$$\int_{\alpha_i} q_s \, dA = \sum_{\text{v} \in S} \int_{\alpha_i} q_f^v \, dA$$

(4.11)

This equation represents the local heat flux conservation for each solid face and can be more easily understood from the following figure, which shows that the heat flow though a solid face equals the total heat flow through all the virtual faces inside the solid face. Hence, global conservation is automatically satisfied.

Figure 4.7 Evaluation of A Surface Integral on the Virtual Grid [47]: the Red Triangle Is A Solid Face and Blue Triangles Are Fluid Faces.
Since the heat fluxes are considered as constant over each fluid face and each solid face, the above integral equation for each solid face can be simplified as:

\[ q_S A(S) = \sum_{v \in S} q_f A(v) \]

Then, the heat flux though the solid face can be calculated as follow:

\[ q_S = \frac{\sum_{v \in S} q_f A(v)}{A(S)} \]

### 4.4.7 The Stability Analysis of Conjugate Heat Transfer Anti-Icing Simulation

A common problem for the loose-coupling approach is the stability of the calculation. Giles[13] has studied the stability of the loose-coupling approach for computing the temperatures and the heat fluxes in coupled fluid/structure interactions. From his study, using a Neumann boundary condition for the structure calculation and a Dirichlet boundary condition for the fluid calculation are suggested to achieve numerical stability. The reverse will cause numerical instability. Most of the CHT codes use a relaxation parameter to relax the local temperature on the solid/fluid interface before imposing it as the boundary condition to an airflow calculation because the fluid flow calculation is very sensitive to the boundary condition changes. This relaxation can be expressed in the following equation.

\[ T_{new}^{n+1} = T^{n+1}(1 - \beta) + \beta T^n \]

A higher value of the relaxation parameter \( \beta \), within the range of \([0, 1]\) corresponds to a higher relaxation. This method is an explicit method to reduce the instability of the system but also greatly increases the computing time. Usually, a relaxation scheme should be used in this method so that the initial relaxation is higher and less relaxation is subsequently used to increase the convergence speed. Extra efforts must be paid to find a proper relaxation scheme. Another method, which is used in the CHT3D simulation to relax the boundary condition exchange to the flow solver, is to impose the Robin boundary conditions to the structure conduction code instead of directly imposing the Neumann boundary conditions. In this method, a virtual heat transfer coefficient variable is introduced and is converted from the convective heat flux as has been discussed in section 4.4.3. The heat transfer coefficients are used in both interface 1 and interface 3.
and are calculated using the equation \( h_c = Q/(T - T_{\text{ref}}) \). In this implicit relaxation method, the relaxation is automatically adjusted according to the deviation of the heat flux from the final converged result. In addition, reducing the time step and iteration numbers in each domain’s calculation and increasing the total number of coupling loops of CHT3D will make the CHT3D coupling tighter and hence increase the stability of the calculation. Moreover, in the CHT3D algorithm, more frequent coupling between ICE3D and C3D in the inner loop will cause higher relaxation in the inner loop outputs, the temperature distributions at both interface 1 and interface 3 and will increase the stability of the CHT3D calculation as a result.

### 4.5 Conclusions

A loose-coupling conjugate heat transfer model, CHT3D, based on explicit boundary condition exchange has been developed to simulate the 3D heat transfer processes of aircraft hot air anti-icing systems. This model uses a modular approach and couples the external airflow domain, water film/ice accretion domain, solid conduction domain, and internal airflow domain, with currently available numerical solvers, FENSAP-ICE package. The temperature and heat flux distributions at each coupling interface are calculated and the anti-icing effect can be evaluated from the solution of the water film domain. This approach can also simulate any conjugate heat transfer problem with two different fluid domains and one solid conduction domain and can be easily expended to applications with any number of fluid/solid interfaces. The solver of each domain in the modular approach can be upgraded independently and separately. In the airflow solver, a more accurate local convective heat flux at the boundary surface, the so-called Gresho heat flux, is calculated with a consistent weak Garlerkin FEM post-processor instead of the classical method of temperature differences. A sand grain roughness model is used to model the surface roughness due to the droplet impingement, which has a significant impact to the convective heat flux at the surface. The different turbulence models in the airflow solver may be required for different types of airflows in order to predict the accurate convective heat fluxes at the fluid/solid interface. The verification and validation of the CHT3D model will be presented in the following chapters.
CHAPTER 5: Testing of CHT3D Code

5.1 Description of a 2-D Verification Hot-Air Anti-icing Test Case

The following 2D verification test case is used to demonstrate the capability of the CHT3D numerical model developed in this thesis for simulating different types of anti-icing results. In this test case, two ducts with different temperature airflows are separated by an aluminum flat plate as shown in the following figure:

![Figure 5.1 The Verification Test Case for CHT3D](image)

The duct above the flat plate has an airflow with an inlet free stream temperature of 260K. The lower duct has an airflow with a higher inlet free stream temperature of 350K. In this test case, a constant impinging droplet flux, without a runback velocity, is imposed on the upper surface of the flat plate in order to facilitate the analysis. The liquid water content is LWC=1.0^{-4} \text{kg/m}^3 and the collection efficiency is \beta=0.5. The mass flux of the impinging droplets to the upper surface of the flat plate can be calculated using

\[ m_w = LWC \cdot U \cdot \beta = 1.0 \times 10^{-4} \times 10 \times 0.5 = 5.0 \times 10^{-4} \text{kg} / (\text{m}^2 \cdot \text{s}) \, . \]

The inlet velocity of the upper airflow is 10 m/s. In order to simulate a full range of antiicing cases (iced, running wet, or fully evaporative), different inlet velocities are chosen for the lower airflow, which is equivalent to a change in the anti-icing hot-air mass flow.
All of the results are plotted for $-0.5 \text{m} < x < 0.5 \text{m}$ along the length of the aluminum flat plate.

This test case is designed for testing the functioning of the CHT3D code without comparing it to any experimental or numerical results. Therefore, only the coarser meshes are used in order to reduce the computational expense involved. This 2D test case is simulated using 3D meshes with 2 nodes in the $z$-direction and a periodic condition imposed on the two surfaces perpendicular to the $z$-axis.

5.2 Hot-Air Anti-Icing Simulations

5.2.1 Case 1: Hot-Air Anti-Icing Simulation with Ice Accretion

With the inlet velocity of the lower hot airflow at 5m/s, the anti-icing thermal energy is not enough to prevent the ice accretion on the entire upper surface of the aluminum flat plate. The iced anti-icing results obtained from CHT3D calculation are shown below.

![Figure 5.2 Water Film Mass Flow on Upper Surface of the Flat Plate](image1)

![Figure 5.3 Heat Flux Distribution on Upper Surface of the Flat Plate](image2)
In the range from $x=-0.5m$ to $x=-0.04m$ on the upper surface of the aluminum flat plate, the wall temperature decreases from 273.7K to 273.15K and no ice accretion is predicted. However, the anti-icing heat flux is not enough to evaporate all the impinging water; thus, the residual water film on the upper surface of the flat plate is still present. The decrease of the evaporative water flux in the airflow direction is due to a decrease of the surface temperatures.

In the range from $x=-0.04m$ to $x=0.5m$, the wall temperature on the upper surface becomes constant and equal to the freezing point 273.15K. As a result, the coexistence of the ice and the water film is present on the upper surface. The increase of the instantaneous mass of ice along the airflow direction is a result of a decrease of the anti-icing hot-air thermal energy. The mass flux of the evaporative water is almost constant because it primarily depends on the surface temperatures. The residual water film on the upper surface decreases due to an increase of ice.

The mass flow figure indicates the mass conservation of the water film. The sum of the evaporative mass flux, ice accretion mass flux, and the instantaneous mass of residual water on the upper surface of the flat plate equals the mass flux of the impinging droplet.
The decrease of the convective heat flux along the airflow direction on the upper surface is a direct result of decrease of temperatures. The decrease of the evaporative heat flux corresponds to a decrease in the evaporative mass flow.

5.2.2 Case 2: Hot-Air Anti-Icing Simulation with Running Wet

When increasing the inlet velocity of the lower hot airflow to 6m/s, the anti-icing heat flux is enough to prevent ice accretion on the upper surface of the flat plate. However, it is still not enough to evaporate all the impinging water. Hence, the running wet anti-icing result is obtained. The results of CHT3D calculation are illustrated in the following figures.

![Figure 5.5 Water Film Mass Flow on Upper Surface of the Flat Plate](image1)

![Figure 5.6 Heat Flux Distribution on Upper Surface of the Flat Plate](image2)
Along the airflow direction from \(-0.5\text{ m}\) to \(0.5\text{ m}\), the temperature decrease on both the upper and lower surfaces of the flat plate can be accounted for by the decrease in thermal energy. The temperature decrease leads to lowering the evaporative heat flux and the convective heat flux on the upper surface of the flat plate. No ice growth on the upper surface is present, due to temperatures greater than 273.15K. The mass flux conservation of the water film can also be seen from the plots.

### 5.2.3 Case 3: Hot-Air Anti-Icing Simulation with Full Evaporation

When further increasing the inlet velocity of the lower hot airflow to 10m/s, the anti-icing heat flux is enough to prevent ice accretion and evaporate all of the impinging water on the upper surface of the flat plate. Thus, fully evaporative anti-icing results are obtained and are shown in the following figures. The evaporative heat flux is constant, corresponding to the constant mass flux of evaporative water.
5.2.4  **Case 4: Hot-Air Anti-Icing Simulation Without Droplets Impingement**

Another test case is also performed with the same inlet velocity of the hot airflow as the above fully evaporative case (10 m/s), but without impinging water, in order to observe the influence of the impinging water. The following figures show that the temperatures and heat fluxes further increase on the flat plate surfaces because the hot-air energy is just used to heat up the solid temperatures without any loss to water evaporation.
5.3 Conclusion

From the above test cases 1 to 3, a full range of hot-air anti-icing cases were successfully simulated by gradually increasing the mass flow of the anti-icing hot air. It can be easily observed that the increase of the anti-icing thermal load leads to the increase of the temperatures on both surfaces of the flat plate. An increase in the thermal load also results in an increase of the mass fluxes and heat fluxes of the evaporative water until reaching the fully evaporation, a decrease of the instantaneous mass of ice accretion, a decrease of the instantaneous mass of the residual water film up to zero, and an increase of the convective heat fluxes on the upper surface. Case 4 shows the capability of the CHT3D model for solving the dry air conjugate heat transfer problem. In that case, the temperatures on both surfaces of the flat plate and the convective heat fluxes on the upper surface are greater than the same case with the droplet impingement.

All the results of the above verification test cases are completely in agreement with the expected qualitative analysis. Hence, the CHT3D procedure faithfully represents the physical anti-icing processes. However, additional quantitative validation test cases are still needed to validate the CHT3D code and will be presented in the following chapter.
CHAPTER 6: Validation of CHT3D Code

6.1 Validation of Turbulence Models in FENSAP for Impinging Slot Jet

For simulating hot-air anti-icing conjugate heat transfer problems, the most important and difficult part is to accurately predict the heat transfer coefficient distribution on the solid internal surface due to the jet impingement, which has a complex flow pattern with strong downstream vortices. The accuracy of the impinging jet solution greatly depends on the validity of the turbulence models in the airflow solver. In this thesis, an anti-icing experiment performed by Laval University is used to validate the CHT3D model. In that experiment, the anti-icing hot airflow is a fully-developed laminar impinging slot jet. However, no measurement results for this impinging jet is available for that experiment. For this reason, another simple impinging slot jet experiment is selected to first validate FENSAP’s capability to predict the heat transfer coefficient for an impinging jet flow, before moving to more complex anti-icing simulations.

Past literature has revealed that the heat transfer property of an impinging jet is very sensitive to the parameters of the jet: Reynolds number, nozzle-to-plate distance, nozzle geometry, jet velocity profile at the nozzle, and roughness of the impinged wall. Therefore, this validation experiment has to be carefully chosen in order to have similar flow conditions and geometry as the impinging jet of the Laval university experiment.

6.1.1 Description of Impinging Slot Jet Experiment

Very few impinging jet experiments for low Reynolds number slot jets are available in the literature. The experiment developed by Sparrow and Wong [48] is the only one that is very well defined and has jet flow conditions and geometry similar to the Laval university experiment. The detailed geometry of Sparrow and Wong’s slot-jet impingement test case is shown in the following figure.
Instead of local heat flux measurement, a naphthalene sublimation technique is applied in this experiment to determine the local mass transfer parameter (Sherwood number) at the impinging surface. The experimental data obtained from this technique is more accurate than the heat transfer measurement because the local mass transfer due to the naphthalene sublimation can be accurately measured with a sensitive depth indicator and the boundary condition of the uniform wall concentration is obtained by keeping the test surface isothermal. In contrast, the heat transfer measurement has many uncertainties that may cause inaccuracy, examples would be the complexity of the heat flux gauge, heat flux losses, calibration error of the gauge, difficulty of truly local measurement due to the size of the gauge, and difficulty of obtaining the desired thermal boundary condition due to heat losses and edge effects.

In this experiment, the local Sherwood Number, a dimensionless concentration gradient at the impinged surface is determined. It is used to express the convection mass transfer capability at the surface and is defined as:

\[ Sh = \frac{h_m L}{D_{AB}} = + \frac{\partial C^*_A}{\partial y^*} \bigg|_{y^*=0} \]

where, \( h_m \) is a convection mass transfer coefficient, \( D_{AB} \) is a binary diffusion coefficient.
To compare numerical and experimental results, the local Sherwood number \( \text{Sh} \) obtained experimentally is converted into a convection heat transfer parameter, Nusselt number \( \text{Nu} \). The Sherwood number can be converted to a Nusselt number by using the following heat-mass transfer relationship:

\[
\text{Nu} = (Pr/Sc)^n \text{Sh}
\]  

(6.1)

Where,

- \( n \) is usually suggested for slot jet impingement [48][49]. 0.42 is used in this test case.
- \( Pr \) is the Prandtl number, which can be expressed as:

\[
Pr = \frac{\nu}{\alpha}
\]  

(6.2)

- \( \nu \) is the kinematic viscosity [m\(^2\)/s]
- \( \alpha \) is the thermal diffusivity, which is defined by:

\[
\alpha = \frac{k_f}{\rho C_p}
\]  

(6.3)

- \( k_f \) is the thermal conductivity of the jet
- \( \rho \) is the density of the jet [kg/m\(^3\)]
- \( C_p \) is the specific heat capacity of the jet [W.s/(kg.K)]
- \( Sc \) is the Schmidt number, which is defined as:

\[
Sc = \frac{\nu}{D_{AB}}
\]

- \( D_{AB} \) is a property of the binary mixture known as the binary diffusion whose value can be found in any thermo-physical properties tables.

The heat transfer coefficient obtained from the numerical calculation can be converted to a Nusselt number by using the following relationship:

\[
\text{Nu} = \frac{hL}{k_f}
\]  

(6.4)

where \( L \) is the slot width \( W \) in this jet impingement test case and \( k_f \) is the thermal conductivity of the fluid at the jet inlet.
Figure 6.1 shows the 2D dimensions in X-Y plane of the test case. In numerical simulations, the length in Z direction is 200mm. In this mass transfer experiment, both the jet temperature and impinging wall temperature equal to room temperature 295.15°K. In the numerical simulation, the temperature on the impinging wall is chosen as 305.15°K in order to evaluate the Nusselt number. The boundary conditions for the numerical simulations are listed in the following table.

<table>
<thead>
<tr>
<th>Ambient conditions</th>
<th>$T_w = 295.15 , ^\circ K$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$P_w = 101325 , Pa$</td>
</tr>
<tr>
<td>Inlet</td>
<td>$T_{jet} = 295.15 , ^\circ K$</td>
</tr>
<tr>
<td></td>
<td>$Re_{w} = 650$ (Bulk velocity of $\overline{U}_{jet} = 7.66 m/s$) based on slot width</td>
</tr>
<tr>
<td></td>
<td>or $Re_{D} = 1200$ based on hydraulic diameter</td>
</tr>
<tr>
<td></td>
<td>Fully developed laminar velocity profile</td>
</tr>
<tr>
<td>Impinging Wall</td>
<td>$T_w = 305.15 , ^\circ K$</td>
</tr>
<tr>
<td>Other lower walls</td>
<td>$Q_w = 0$ (adiabatic wall)</td>
</tr>
<tr>
<td>Upper walls</td>
<td>$T_w = 295.15 , ^\circ K$</td>
</tr>
<tr>
<td>Exit</td>
<td>$P_{exit} = 101325 , Pa$</td>
</tr>
</tbody>
</table>

Table 6.1 Boundary Conditions for Slot Impinging Jet Test Case

The jet flow and geometry are symmetric to the Y-Z plane at location X = 0 mm, so only half the geometry is solved by FENSAP to reduce the computational load and the symmetric boundary condition is applied to this plane.

6.1.2 Numerical Implementation with Spalart-Allmaras Turbulence Model

First, the flow is solved by FENSAP using the Spalart-Allmaras turbulence model. The test case uses a structured hexahedral grid and has total 11800 nodes with 118 nodes in the X-direction, 50 nodes in the Y-direction, and 2 nodes in the Z-direction. The
following figure shows the mesh in the X-Y plane. The 2 nodes in the Z direction are periodic; hence, this 2D problem is solved with a quasi-3D mesh.

![Mesh with 11800 Nodes for Spalart-Allmaras Turbulence Model](image)

Figure 6.2 Mesh with 11800 Nodes for Spalart-Allmaras Turbulence Model

![Y+ of the First Layer Nodes Away From Impinging Wall](image)

Figure 6.3 $Y^+$ of the First Layer Nodes Away From Impinging Wall

Figure 6.3 shows the $Y^+$ of the first node off the impinging wall solve by FENSAP using the Spalart-Allmaras turbulence model. The variation of $Y^+$ is present because of the variations of the sheer stress and the shear velocity along the X-direction. The maximum value $Y^+ = 1.58$ occurs at $X = 7.4\text{mm}$ where the maximum shear stress is present due to the jet starting transit from the impinging region to the wall jet region. The average $Y^+$ in the test region ($X \leq 63.5\text{mm}$) is less than 0.9.
The above figure shows the Mach number distribution from the numerical simulation with the Spalart-Allmaras turbulence model. In order to ensure the results are mesh-independent, we also solve the same problem with a much finer mesh. The finer mesh has a total of 30080 nodes with 188 nodes in the X-direction, 80 nodes in the Y-direction, and 2 nodes in the Z-direction. The Nusselt number results from the two different meshes are nearly identical, proving the coarser mesh is already fine enough for this test case. In the fine mesh result, the average $y^+$ of the first node off the wall for the test region ($X \leq 63.5\text{mm}$) is less than 0.4.

6.1.3 Numerical Implementation with $k-\varepsilon$ Turbulence Model

The test case was also solved using FENSAP’s two-equation $k-\varepsilon$ turbulence model with wall functions. The computational grid shown in the following figure has total 5880 nodes with 98 nodes in the X-direction, 30 nodes in the Y-direction, and 2 nodes in the Z-direction. The average $y^+$ of the first node off the impinging wall in the test region is less than 6, which is smaller than the usual wall function range but FENSAP can still solve the near wall gradients. The maximum $y^+$ value is 8 and occurs at the position $X = 7.5\text{mm}$. 
The following figure illustrates the Mach number distribution from the numerical simulation with the k-ε turbulence model.

6.1.4 Comparison and Conclusion of Impinging Slot Jet Simulations

Numerical and experimental Nusselt number distributions are compared in Figure 6.7. Wilcox's study [50] shows that the Spalart-Allmaras turbulence model over-predicts the plane jet or slot jet spreading rate by more than 40%, compared to the measured results. However, the spreading rates predicted by the $k-\varepsilon$ model are close to the measured values for the slot jet. For a slot jet, round jet, and radial jet, the spreading rate is the value of $y/x$ where the velocity is half its centerline value ($y$ is the half of the jet width and $x$ is the distance to the jet nozzle). From the above Mach number distribution figures
of the Spalart-Allmaras model and the $k-\varepsilon$ model, it is easier to observe that the spreading rate of the jet predicted by Spalart-Allmaras model is higher than the one predicted by $k-\varepsilon$ model. Thus, the higher energy dissipation of the slot jet in the Spalart-Allmaras turbulence model will weaken the heat transfer of the jet impingement, especially at the impinging region. FENSAP results are in agreement with Wilcox’s conclusion, with a maximum Nusselt number predicted by the Spalart-Allmaras turbulence model 30% lower than the one predicted by the $k-\varepsilon$ turbulence model. The solution from the $k-\varepsilon$ turbulence model is very close to the experimental result. From this test case, we can conclude that the $k-\varepsilon$ turbulence model in FENSAP is superior to the Spalart-Allmaras turbulence model in predicting the heat transfer coefficient of an impinging slot jet with a fully-developed laminar velocity profile.

![Figure 6.7 Comparison of Local Nusselt Number between k-\varepsilon Model, Spalart-Allmaras Model, and Experimental Result]

6.2 Validation of CHT3D Code with a Simplified Hot-Air Anti-Icing Test Case

6.2.1 Description of the Test Case

The details of this simplified anti-icing experiment performed by Laval University have been introduced in the literature review. The following figure shows the geometry used for the test case to validate the CHT3D model.
This 2D experimental setup is a simplified model of an aircraft hot-air anti-icing system. The test section of the wind tunnel has an inlet with size of 0.61m×0.46m; the pressure gradient on the floor is similar to the one on the suction side of a NACA2412 airfoil with the chord length of 2.5m and an angle of attack of 18 degrees. The pressure gradient on the floor is obtained by changing the ceiling height of the wind tunnel.

The aluminum flat plate is equipped with 11 heat flux gages evenly spaced along the flow direction. No water impinges on the flat plate, and thus only dry air results are available.
The following figure shows the detailed geometry of the impinging jet flow enclosed within an aluminum box. Contrary to most aircraft anti-icing systems, the air at the jet nozzle is not choked and is essentially a fully-developed laminar flow.

6.2.2 Initial Wind Tunnel Airflow Simulation

Mesh:
This figure shows the partial mesh used for the wind tunnel airflow calculation with FENSAP. It is a quasi-3D hexahedral mesh and has a total of 77120 nodes with 2 periodic nodes in the z direction.

Figure 6.12 The Partial Mesh of the Wind Tunnel Airflow

**Boundary conditions for wind tunnel airflow:**

| Inlet                  | Turbulence velocity profile as shown in Figure 6.13 ($\bar{U} = 9.5\text{m/s}$)
|------------------------|-------------------------------------------------------------------------------------
|                         | $T_{in} = T_w = 296.65^\circ K$                                                   |
| Upper surface           | Initial temperature $T_w = 300^\circ K$                                           |
| of the flat plat        |                                                                                   |
| Other lower walls       | $Q_w = 0$ (adiabatic wall)                                                         |
| Upper walls             | $Q_w = 0$ (adiabatic wall)                                                         |
| Exit                   | $P_\infty = 101325\text{Pa}$                                                      |

Table 6.2 Boundary Conditions for the Wind Tunnel Airflow

The temperature of $T_w = 300^\circ K$ on the upper surface of the flat plate is only used for the initial FENSAP calculation of the wind tunnel airflow and will be updated during CHT3D iterations. The inlet velocity profile with a bulk velocity of 9.5 m/s is provided by the experiment.
Results:

FENSAP solves the wind tunnel airflow with the Spalart-Allmaras turbulence model. The following figure shows the Mach number distribution of the initial airflow solution. The airflow recirculation found on the second half of the wind tunnel floor is the same as the observation from the experiment.

Figure 6.14 The Mach Number Distribution of the Wind Tunnel Airflow

The following figure shows the comparison of the pressure coefficients on the wind tunnel floor between the numerical simulation and the experimental result. The x-axis
coordinate is divided by the chord length of the NACA2412 airfoil and the coordinate origin is at the kink on the wind tunnel floor. Agreement is good except a slight discrepancy after x/c=0.6. The possible reason of this discrepancy is that the height of the straight duct starting from x/c=0.6 to the exit of the wind tunnel obtained from the CAD file provided by Laval University is slightly smaller than the one described by Girard [33]. However, we cannot rebuild the CAD because of the scarcity of the detail description in the geometry of the experimental assembly. The only information of the geometry we can count on is the CAD file from Laval University. A smaller height of the tunnel will cause a smaller pressure of the airflow. As a result, a smaller numerical pressure coefficient on the wind tunnel floor after x/c=0.6 is shown in the following figure. However, this discrepancy will not have an effect to the CHT3D simulation because only the heat convection on the upper surface of the aluminum flat plate is important. The flat plate is located from x/c=0.01 to x/c=0.07 where the pressure coefficients are well matched between the numerical simulation and the experimental results.

![Figure 6.15 Comparison of the Pressure Coefficient on the Wind Tunnel Floor with Experimental Result](image)

Figure 6.15 Comparison of the Pressure Coefficient on the Wind Tunnel Floor with Experimental Result
6.2.3 Initial Impinging Jet Simulation

Mesh:

This figure shows the mesh used for the impinging jet simulation with FENSAP. It is a quasi-3D hexahedral mesh and has total 54065 nodes with 2 periodic nodes in z direction.

![Figure 6.16 The Mesh of the Impinging Jet flow](image)

**Boundary conditions for impinging jet flow:**

<table>
<thead>
<tr>
<th>Inlet</th>
<th>$m = 0.004736 \text{kg} / \text{s}$ (fully developed laminar flow)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$T_{\text{jet}} = 319.85^\circ K$</td>
</tr>
<tr>
<td>Lower surface of the flat plat</td>
<td>Initial temperature $T_w = 306.45^\circ K$</td>
</tr>
<tr>
<td>Other walls</td>
<td>$Q_w = 0$ (adiabatic wall)</td>
</tr>
<tr>
<td>Exit</td>
<td>$P_\text{a} = 101325\text{P}_a$</td>
</tr>
</tbody>
</table>

Table 6.3 Boundary Conditions for the Impinging Jet Flow

The initial temperature of $T_w = 306.45^\circ K$ on the impinging surface of the flat plate is only used for the initial FENSAP calculation of the impinging jet flow and will be updated in CHT3D iterations. The fully-developed laminar jet at the inlet has an average velocity of 7.66m/s.
This impinging jet flow is solved by FENSAP with the Spalart-Allmaras turbulence model. The following figure shows the Mach number contour lines from the initial FENSAP simulation, giving an idea of the flow pattern. Large vortices with very small velocity occur at the extensive downstream region. At box corners and near the walls, small secondary recirculations can also be observed.

![Mach Number Contour Lines of Jet Flow](image)

Figure 6.17 The Mach Number Contour Lines of Jet Flow

From the validation test case of the impinging slot jet in section 6.1, we were convinced that the k-ε turbulence model in FENSAP is more accurate than the Spalart-Allmaras turbulence model in predicting the heat transfer of a laminar impinging slot jet. However, due to the complexity of the jet flow in this anti-icing test case, a lot of effort still fails to achieve a converged result from the k-ε turbulence model in FENSAP. Furthermore, no experimental measurements, except the mass flow rate at the inlet, are available for this jet flow. Hence, this impinging jet flow is also simulated using FLUENT's commercial flow solver in order to compare the heat transfer coefficient distribution on the bottom side of the flat plate with the FENSAP calculation. Figure 6.18 shows calculation results from FLUENT with Spalart-Allmaras turbulence model and k-ε turbulence model and the
calculation result from FENSAP with Spalart-Allmaras turbulence model. The heat transfer coefficients predicted by both flow solvers are very close in most areas, except that the maximum value predicted by FENSAP is 30% higher than that predicted by FLUENT at the impinging region.

![Figure 6.18 Comparison of Heat Transfer Coefficient Distribution on Impinging Surface](image)

Without experimental results, it is hard to tell which the best result is. The difference between the two flow solvers results may be caused by the fact that FENSAP’s mesh was used for the FLUENT calculation and may not be adapted to FLUENT’s needs. In addition, it is possible that the artificial viscosity scheme is too dissipative in FLUENT. Globally, it seems that FENSAP predicts higher heat flux than FLUENT. That means FLUENT cannot give a better result as what we expect and is even worse than the result from the FENSAP calculation with the Spalart-Allmaras turbulence model.

### 6.2.4 CHT3D Numerical Simulation

In the Laval University anti-icing experiment, the material properties of the aluminum flat plate are not specified. Hence, the following thermal properties of pure aluminum are chosen for the flat plate:
- Density: 2702 kg/m$^3$
- Thermal conductivity: 200 W/m.K
- Specific heat: $C_p=900$J/kg.K
Although not really needed for the dry air calculation, DROP3D and ICE3D numerical codes are still involved in the CHT3D iterative loop in order to validate all the coupling schemes of the CHT3D code. For the DROP3D solver, a liquid water content near zero is used and hence the ICE3D solver is run with no impinging water mass. Both the DROP3D and ICE3D calculations use very small iteration numbers to reduce total calculation time of the CHT3D simulation because they do not affect the result of the dry air simulation. The mesh of the flat plate shown in the following figure has total 5520 nodes with 2 nodes in Z direction.

![Figure 6.19 The Mesh of the Aluminum Flat Plate](image)

After all the preliminary work above, the proposed CHT3D procedure is able to run and iteratively solve temperature and heat flux distributions in different domains until the temperature distribution at each interface reaches a steady-state. In this CHT3D simulation, 5 iterations are chosen for the inner loop, in which ICE3D and C3D are coupled together by iteratively exchanging boundary conditions. 80 iterations are chosen for the outer loop, in which FENSAP updates the calculations of both the external and internal airflows. The following figure shows the temperature convergence history of the CHT3D calculation at the interface between the external airflow and the water film, which is just a fictitious domain in this dry air test case. The residue value is defined as the average temperature difference between two iterations, and converges to a final value of $9.6 \times 10^{-4}$ K at the last iteration. The maximum temperature difference at this interface is $4.6 \times 10^{-30}$ K. The average temperature residue at the interface between the aluminum flat plate and internal jet flow at the last iteration is $4.4 \times 10^{-40}$ K. The average residue at the interface between the water film and aluminum flat plate at the last iteration is $3.6 \times 10^{-50}$ K. In the follow figure, the residue is already small enough and does not obviously decrease any more after 15 CHT3D iterations. In the literature review, the residue of the CHT2D simulation of Bombardier test case ends up below $0.1$ K. This convergence criterion is
achieved in just 4 iterations in this CHT3D simulation. This result shows very good convergence capability of the CHT3D calculation.

Figure 6.20 CHT3D Convergence History of the Temperature on the Flat Plate Upper Surface

Figure 6.21 and Figure 6.22 show the result of heat transfer coefficient distributions and heat flux distributions on both the upper and lower surfaces of the aluminum flat plate from the CHT3D calculation. In both figures, very large values can be seen at the impinging region of the lower surface of the plate due to the high heat transfer rate of the jet impingement. A steep decrease occurs when moving from the impinging point to both ends of the flat plate. However, the variations of the heat transfer coefficient and heat flux on the upper surface of the plate are much more gradual than those on the lower surface because the strong 2D heat conduction in the thin aluminum flat plate smoothes the temperature variation in horizontal direction. The heat flow conservation can be observed in Figure 6.22. The total heat flow entering to the lower surface of the flat plate equals the total heat flow leaving from the upper surface.
Figure 6.21 Heat Transfer Coefficients on Both Surfaces of Aluminum Flat Plate

Figure 6.22 Convective Heat Fluxes on Both Surfaces of Aluminum Flat Plate

The temperature distribution in the aluminum flat plate is shown in Figure 6.23. Figure 6.24 shows the temperature distribution on both the upper and lower surfaces of the flat plate. In this figure, the temperature variation is less than one degree along the x direction.
of the plate, due to the high thermal conductivity of aluminum. Even if the variation in temperature is small, the heat flux is strongly 2D. For example, the heat flux along the $x$ direction is approximately:

$$Q = -k \frac{dT}{dx} \approx -k \frac{\Delta T}{\Delta x} = -200 \times \frac{303 - 302.2}{0.06} = -2666 \text{ W/m}^2$$

This is more than twice the heat flux from impinging jet side to wind tunnel side.

![Figure 6.23 Temperature Distribution in the Aluminum Plate (The Thickness of the Plate Is Scaled for Better View)](image)

![Figure 6.24 Temperature Distributions on Both the Upper and Lower Surfaces of Aluminum Flat Plate](image)
Heat flux measurements across the upper surface of the flat plate are available. Predicted heat fluxes with the CHT3D code are compared to the experimental ones in Figure 6.25. Near the impinging region, the results agree with the experimental measurements, but discrepancies increase as the distance increases from the impinging point. The overall predicted heat fluxes from the CHT3D simulation are lower than the experimental ones.

![Figure 6.25 Comparison of Heat Fluxes on Upper Surface of the Aluminum Flat Plate between CHT3D Calculation and Experimental Result with Error Bar](image)

Owing to the strong 2D heat conduction in the flat plate, it is hard to tell what causes this discrepancy. The possible reasons of discrepancies from experimental results can be:

- The Spalart-Allmaras turbulence model under-predicts the heat transfer rate for the impinging slot jet, which has been confirmed from a simple test case at the beginning of this Chapter. It is the most possible reason that the overall heat flux from the CHT3D simulation is lower than the experimental measurements.

- Only the error of heat flux measurement is considered in the experiment. Some of the adiabatic wall conditions we used at the ends of the flat plate may not reflect the experimental setup. The flat plate is isolated from other wind tunnel floor plates, but heat flux leaking may occur from the sides of the plate.
• Heat conductivity of the flat plate is not specified in the experiment and it will affect the heat flux results. The higher heat conductivity will transfer more heat flux to the ends of the flat plate from the impinging region due to the strong heat conduction in horizontal direction of the flat plate.

Nevertheless, the overall CHT3D procedure does not seem to be responsible for the discrepancy. Investigations are still needed before we can conclude the study and further validation may be carried out by Bombardier on its proprietary geometries and data.

### 6.3 Conclusions

The CHT3D procedure has been validated against a 2D dry air anti-icing experimental test case with partial success. It can successfully simulate and solve hot-air anti-icing problems. The discrepancies between the numerical simulation and the experimental measurement do not seem to be the responsibility of the coupling procedure in the numerical code but the turbulence model used to solve the jet flow or the experimental setup. The numerical result of the impinging jet shows how an anti-icing system simulation can rapidly grow in complexity, even in 2D. Further investigations and efforts may be still needed to improve the results of the numerical simulation.
Conclusions and Future Work

Due to the great impact on flight safety by in-flight ice accretion, effective ice protection measures have to be applied against in-flight ice buildup. To design efficient hot-air anti-icing systems is very important in order to reduce the penalties due to the bleeding air from engines. CFD tools are continuously being developed to assist in anti-icing system design and in certification of aircraft. In this thesis, a 3D conjugate heat transfer module (CHT3D) has been developed to simulate hot-air anti-icing processes based on a loose-coupling procedure, which has great flexibility in solving complicated conjugate heat transfer problems. Different numerical methods for the general CHT and hot-air anti-icing simulations have been discussed in the literature review. The CHT3D module uses the currently available 3D CFD package FENSAP-ICE to iteratively solve the external cold airflow, water film/ice accretion, solid conduction, and internal hot airflow based on iteratively exchanging thermal boundary conditions at common interfaces of different domains. The mathematical equations and numerical methods used for solving different domains have been presented in chapter 3 and the CHT3D algorithm has been elaborated in chapter 4. Robin boundary conditions are imposed at most of the interfaces as a relaxation scheme for CHT3D coupling and have been proved to be robust in eliminating the numerical instability. Currently, all domains coupled by CHT3D must have matching grids at their common interfaces, which can accurately exchange boundary conditions but is difficult for industrial level simulations with complex geometries. A virtual grid technique for boundary condition exchanges to non-matching grids has been introduced and can be implemented into the current CHT3D code.

In chapter 5, CHT3D code has been tested to successfully simulate the full range of anti-icing cases. The verification test cases are simple but comprehensive enough to test the full capabilities of the CHT3D code. The results of CHT3D simulations for these test cases are in agreement with the physical observations.

In chapter 6, CHT3D has been validated against a simplified 2D hot-air anti-icing experiment performed by Laval University. This experiment mimics a real hot-air anti-
icing system with flow around a wing, internal impinging jet flow and solid conduction, but without droplet impingement. This experiment is the most suitable we can find to validate the CHT3D code. The simulation results are partially successful in comparison with the experimental results of heat fluxes. It does not seem that the discrepancies are related to the CHT3D procedure but most likely the shortcomings of the turbulence model for impinging jets and to the errors in experimental setup.

To sum up, the CHT3D procedure developed in this thesis has been proven successful in simulating the hot-air anti-icing processes. However, its applications are not only limited in anti-icing simulation. It can also apply to any CHT problems with two air/fluid flows separated by a solid with/without the water film on the external surface of the solid.

Some future work may involve the following:

- Further effort to solve the impinging jet in Laval test case with k-ε turbulence model.
- Add the capability into the current CHT3D code of exchanging boundary conditions for non-matching grids.
- 2D and 3D experiments including water runback and evaporation effects would be needed to further validate the CHT3D code.
Bibliography


