NUMERICAL INVESTIGATION OF A WING HOT AIR ICE PROTECTION SYSTEM

A Thesis by

Alonso Oscar Zamora Rodríguez

Bachelor of Science, Wichita State University, 2007

Submitted to the Department of Aerospace Engineering
and the faculty of the Graduate School of
Wichita State University
in partial fulfillment of
the requirements for the degree of
Master of Science

December 2009
NUMERICAL INVESTIGATION OF A WING HOT AIR ICE PROTECTION SYSTEM

The following faculty members have examined the final copy of this thesis for form and content, and recommend that it be accepted in partial fulfillment of the requirement for the Master of Science with a major in Aerospace Engineering.

________________________________________________________________________
Michael Papadakis, Committee Chair

________________________________________________________________________
Scott Miller, Committee Member

________________________________________________________________________
Ikram Ahmed, Committee Member
ACKNOWLEDGEMENTS

I would like to thank Dr. Michael Papadakis for giving me the opportunity to work in the CFD/Icing lab and to participate on the research effort that gave way to this thesis. I would also like to acknowledge See-Ho Wong, Hsiung-Wei Yeong, See-Cheuk Wong, and all the graduate and undergraduate students in the CFD/Icing lab for their continuous help, support, and comradeship throughout these years.
ABSTRACT

Aircraft icing is a recurrent aviation safety concern. In the past eight years alone, eight icing accidents involving business jets and other aircraft have occurred. The accumulation of ice on critical aerodynamic surfaces, the primary cause of these accidents, leads to considerable performance degradation that compromises the safety of the passengers, the crew, and the vehicle.

A variety of surface-deformation and thermal systems provide icing protection for aircraft. Hot air anti-icing systems are the most common for airplanes with aluminum leading edges on wing and tail surfaces, and engine inlets. These surfaces are heated using bleed air redirected from the jet engine compressor and channeled through a piccolo tube located inside the leading edge. A series of hot air jets emanate from small holes on the piccolo tube (piccolo holes) and impinge on the internal surface of the leading edge skin, transferring heat, and increasing the skin temperature to prevent ice accumulation.

The design and optimization of hot air anti-icing systems involve both experimental and numerical studies. Computational Fluid Dynamics (CFD) is a cost-effective analysis tool for bleed air ice protection system design and evaluation. CFD analysis tools, however, require validation against experimental data to determine the accuracy of the numerical schemes, turbulence models, boundary conditions, and results obtained. The present thesis details a CFD methodology developed to simulate the performance of a wing hot air anti-icing system under dry air conditions (no water impingement).

Computational simulations were conducted with the commercial CFD code FLUENT to investigate the performance of a hot air anti-icing system installed in the leading edge of a 72-inch span, 60-inch chord business jet wing model. The analysis was performed with a full-span model (FSM) and a partial-span model (PSM). The FSM was used to model the entire length of the piccolo tube to investigate the development of spanwise flow inside the piccolo tube. The PSM was used to model a 2.44-in spanwise section of the wing in order to investigate the internal and external flow properties about the wing with the bleed air system in operation. Computational results obtained with the PSM model were compared with experimental data obtained from icing tests performed at the NASA Glenn Icing Research Tunnel (IRT) facility.
The work presented in this thesis includes extensive 2D axisymmetric computational studies performed with a subsonic, heated, turbulent jet impinging on a flat plate to evaluate the performance of five eddy-viscosity turbulence models available in the FLUENT code. The turbulence model studies showed that the Shear Stress Transport (SST) $\kappa-\omega$ formulation provided the most consistent prediction of recovery temperatures at the impingement wall.

Grid resolution and spatial discretization studies were completed with a three-dimensional version of the jet impingement scenario employed in the turbulence study, and first- and second-order upwind schemes. Three grid resolution levels were considered based on the number of nodes distributed around the nozzle exit circumference in order to apply the same distribution around the piccolo holes circumferences in the anti-icing system PSM.

A boundary condition study was performed with the anti-icing models (FSM and PSM). The PSM did not model the piccolo tube internal flow and, consequently, required inflow boundary conditions to be specified at the piccolo holes’ exits. The FSM was employed to analyze the flow inside the piccolo tube and to obtain the inflow boundary conditions for the PSM. The approaches applied to extract the boundary conditions were centerline and cell-averaged. Skin temperature results from the PSM were compared with available experimental data and showed that the cell-averaged approach provided the most accurate simulation.

Finally, a parametric study was conducted with the anti-icing models (FSM and PSM) to validate the computational methodology with a broad range of cases with variable internal and external flow parameters for which experimental data was available. The results for leading-edge skin temperature as well as piccolo flow properties demonstrated in all cases high-fidelity agreement with experimental data.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1 Background</td>
<td>1</td>
</tr>
<tr>
<td>1.2 Literature Review</td>
<td>8</td>
</tr>
<tr>
<td>1.3 Objectives</td>
<td>12</td>
</tr>
<tr>
<td>1.4 Technical Approach</td>
<td>12</td>
</tr>
<tr>
<td>2. COMPUTATIONAL METHODOLOGY</td>
<td>14</td>
</tr>
<tr>
<td>2.1 Computational Model</td>
<td>14</td>
</tr>
<tr>
<td>2.2 Grid Generation</td>
<td>18</td>
</tr>
<tr>
<td>2.3 Governing Equations</td>
<td>21</td>
</tr>
<tr>
<td>2.4 Boundary Conditions</td>
<td>23</td>
</tr>
<tr>
<td>2.5 Numerical Schemes</td>
<td>25</td>
</tr>
<tr>
<td>2.6 Viscous Flow Analysis</td>
<td>27</td>
</tr>
<tr>
<td>2.6.1 Jet Flow Turbulence Modeling</td>
<td>28</td>
</tr>
<tr>
<td>2.6.2 Turbulence Models</td>
<td>30</td>
</tr>
<tr>
<td>2.6.3 Transition</td>
<td>34</td>
</tr>
<tr>
<td>3. RESULTS AND DISCUSSION</td>
<td>37</td>
</tr>
<tr>
<td>3.1 Experimental Database</td>
<td>37</td>
</tr>
<tr>
<td>3.1.1 Wing Model Geometry</td>
<td>37</td>
</tr>
<tr>
<td>3.1.2 Model Instrumentation</td>
<td>38</td>
</tr>
<tr>
<td>3.1.3 Experimental Test Cases</td>
<td>41</td>
</tr>
<tr>
<td>3.2 Turbulence Model Study</td>
<td>42</td>
</tr>
<tr>
<td>3.3 Grid Resolution and Spatial Discretization Study</td>
<td>48</td>
</tr>
<tr>
<td>3.4 Transition Modeling</td>
<td>54</td>
</tr>
<tr>
<td>3.5 Boundary Conditions Study</td>
<td>59</td>
</tr>
<tr>
<td>3.6 Parametric Study</td>
<td>69</td>
</tr>
<tr>
<td>3.6.1 Effect of External Flow Conditions</td>
<td>71</td>
</tr>
<tr>
<td>3.6.2 Effect of Hot Air Temperature</td>
<td>71</td>
</tr>
<tr>
<td>3.6.3 Effect of Hot Air Mass Flow at High Temperature</td>
<td>72</td>
</tr>
</tbody>
</table>
### TABLE OF CONTENTS (continued)

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.6.4</td>
<td>Effect of Hot Air Mass Flow at Low Temperature</td>
</tr>
<tr>
<td>3.6.5</td>
<td>Effect of Piccolo Configuration</td>
</tr>
<tr>
<td>4.</td>
<td>CONCLUSIONS AND RECOMMENDATIONS</td>
</tr>
<tr>
<td>4.1</td>
<td>Conclusions</td>
</tr>
<tr>
<td>4.2</td>
<td>Recommendations for Future Work</td>
</tr>
<tr>
<td>REFERENCES</td>
<td></td>
</tr>
<tr>
<td>APPENDICES</td>
<td></td>
</tr>
<tr>
<td>A.</td>
<td>Partial Span Model Grid Description</td>
</tr>
<tr>
<td>B.</td>
<td>CFX Transition Model Formulation</td>
</tr>
<tr>
<td>C.</td>
<td>Effects of Transition Location on Skin Temperature</td>
</tr>
<tr>
<td>D.</td>
<td>Test Model Instrumentation</td>
</tr>
</tbody>
</table>
### LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Business, regional, and transport aircraft icing accidents since 1982</td>
<td>2</td>
</tr>
<tr>
<td>2. Geometric specification of computational models</td>
<td>16</td>
</tr>
<tr>
<td>3. FSM grid size information</td>
<td>19</td>
</tr>
<tr>
<td>4. PSM grid size information</td>
<td>20</td>
</tr>
<tr>
<td>5. Turbulence model selection criteria</td>
<td>27</td>
</tr>
<tr>
<td>6. Experimental test cases</td>
<td>42</td>
</tr>
<tr>
<td>7. Grid Resolution study information</td>
<td>50</td>
</tr>
<tr>
<td>8. PSM Piccolo hole boundary conditions</td>
<td>69</td>
</tr>
<tr>
<td>9. Computed and experimental skin temperatures</td>
<td>70</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>1</td>
<td>Wreckage of Bombardier CL-600-2A12, Montrose, Colorado, 2004.</td>
</tr>
<tr>
<td>2</td>
<td>Wreckage of Bombardier CL-600-2B16 in Moscow, Russia, 2007.</td>
</tr>
<tr>
<td>3</td>
<td>Wing leading edge separation bubble</td>
</tr>
<tr>
<td>4</td>
<td>Critical surfaces for ice protection on a business jet.</td>
</tr>
<tr>
<td>5</td>
<td>HAARP II wing model in icing tunnel test section</td>
</tr>
<tr>
<td>6</td>
<td>Full Span Model (FSM) geometry</td>
</tr>
<tr>
<td>7</td>
<td>Partial Span Model (PSM) geometry</td>
</tr>
<tr>
<td>8</td>
<td>Leading edge cross-section</td>
</tr>
<tr>
<td>9</td>
<td>Isometric view of Full-Span Model</td>
</tr>
<tr>
<td>10</td>
<td>Section view of Full-Span Model</td>
</tr>
<tr>
<td>11</td>
<td>Isometric view of Partial-Span Model</td>
</tr>
<tr>
<td>12</td>
<td>Section view of Partial-Span Model</td>
</tr>
<tr>
<td>13</td>
<td>Diagram of single jet impingement on a flat surface</td>
</tr>
<tr>
<td>14</td>
<td>Airfoil section (not to scale)</td>
</tr>
<tr>
<td>15</td>
<td>Bleed air system details</td>
</tr>
<tr>
<td>16</td>
<td>Spanwise locations of leading edge skin thermal instrumentation</td>
</tr>
<tr>
<td>17</td>
<td>Distribution of T-type thermocouples embedded inside the leading edge skin</td>
</tr>
<tr>
<td>18</td>
<td>Chordwise distribution of heat flux sensors attached to the skin interior surface</td>
</tr>
<tr>
<td>19</td>
<td>Thermocouples installed in diffuser and diffuser passages.</td>
</tr>
<tr>
<td>20</td>
<td>Thermocouples installed on back of inner-liner</td>
</tr>
<tr>
<td>21</td>
<td>Bleed air supply line and instrumentation</td>
</tr>
<tr>
<td>22</td>
<td>Axisymmetric grid – impinging jet</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>23.</td>
<td>Axisymmetric grid – jet region.</td>
</tr>
<tr>
<td>24.</td>
<td>Spalart-Allmaras results.</td>
</tr>
<tr>
<td>25.</td>
<td>Standard $\kappa - \epsilon$ results</td>
</tr>
<tr>
<td>26.</td>
<td>Realizable $\kappa - \epsilon$ results</td>
</tr>
<tr>
<td>27.</td>
<td>Standard $\kappa - \omega$ results</td>
</tr>
<tr>
<td>28.</td>
<td>Shear Stress Transport $\kappa - \omega$ results</td>
</tr>
<tr>
<td>29.</td>
<td>Jet centerline turbulence kinetic energy ($k$)</td>
</tr>
<tr>
<td>30.</td>
<td>Recovery factor distribution for turbulence intensity of 1%</td>
</tr>
<tr>
<td>31.</td>
<td>Recovery factor distribution for turbulence intensity of 3%</td>
</tr>
<tr>
<td>32.</td>
<td>Comparison of nozzle mesh for grid resolution study</td>
</tr>
<tr>
<td>33.</td>
<td>Comparison of grid resolution in 3D jet region</td>
</tr>
<tr>
<td>34.</td>
<td>Recovery factor comparison for three grid levels with 1st order discretization</td>
</tr>
<tr>
<td>35.</td>
<td>Recovery factor comparison for three grid levels with 2nd order discretization</td>
</tr>
<tr>
<td>36.</td>
<td>Coarse 3D jet recovery factor for 1st and 2nd order solutions</td>
</tr>
<tr>
<td>37.</td>
<td>Medium 3D jet recovery factor for 1st and 2nd order solutions</td>
</tr>
<tr>
<td>38.</td>
<td>Fine 3D jet recovery factor for 1st and 2nd order solutions</td>
</tr>
<tr>
<td>39.</td>
<td>HAARP wing model in wind tunnel test Section, AOA = 3 deg</td>
</tr>
<tr>
<td>40.</td>
<td>Contours of transition onset momentum thickness Reynolds number</td>
</tr>
<tr>
<td>41.</td>
<td>Boundary layer velocity profiles</td>
</tr>
<tr>
<td>42.</td>
<td>Boundary layer transition onset momentum-thickness Reynolds number profiles</td>
</tr>
<tr>
<td>43.</td>
<td>Laminar regions surrounding Partial-Span Model (PSM) at AOA = 3 deg</td>
</tr>
<tr>
<td>44.</td>
<td>Spanwise temperature on the piccolo tube outer surface, FSM (Run 26)</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>45.</td>
<td>Piccolo flow spanwise temperature, FSM (Run 26).</td>
</tr>
<tr>
<td>46.</td>
<td>Total pressure spanwise distribution at piccolo centerline, FSM (Run 26).</td>
</tr>
<tr>
<td>47.</td>
<td>Total temperature spanwise distribution at piccolo centerline, FSM (Run 26).</td>
</tr>
<tr>
<td>48.</td>
<td>Total pressure variation from piccolo centerline to jet exit, station B (Run 26).</td>
</tr>
<tr>
<td>49.</td>
<td>Total temperature variation from piccolo centerline to jet exit, station B (Run 26).</td>
</tr>
<tr>
<td>50.</td>
<td>Mach number variation from piccolo centerline to jet exit, station B (Run 26).</td>
</tr>
<tr>
<td>51.</td>
<td>Static pressure variation from piccolo centerline to jet exit, station B, Run 26.</td>
</tr>
<tr>
<td>52.</td>
<td>Normalized boundary conditions at the piccolo jet exit, FSM (Run 26).</td>
</tr>
<tr>
<td>53.</td>
<td>Piccolo hole cross section with total pressure contours, FSM (Run 26).</td>
</tr>
<tr>
<td>54.</td>
<td>Effect of piccolo hole inflow conditions on skin temperature, PSM (Run 26).</td>
</tr>
<tr>
<td>55.</td>
<td>Streamlines colored by total temperature in PSM (Run 26).</td>
</tr>
<tr>
<td>56.</td>
<td>External skin temperature, PSM (Run 26).</td>
</tr>
<tr>
<td>57.</td>
<td>Interior surface heat flux comparison, single- and double-jet stations, PSM (Run 26).</td>
</tr>
<tr>
<td>58.</td>
<td>Interior surface heat flux comparison, in-between jet stations, PSM (Run 26).</td>
</tr>
<tr>
<td>59.</td>
<td>Total temperature contours at 5 jet stations in PSM (Run 26).</td>
</tr>
<tr>
<td>60.</td>
<td>Total pressure contours at 5 jet stations in PSM (Run 26).</td>
</tr>
<tr>
<td>61.</td>
<td>Contours of Mach number for 0-deg jet in PSM (Run 26).</td>
</tr>
<tr>
<td>62.</td>
<td>Contours of total pressure for 0-deg jet in PSM (Run 26).</td>
</tr>
<tr>
<td>63.</td>
<td>Leading edge forward, upper, and lower regions.</td>
</tr>
<tr>
<td>64.</td>
<td>Effect of external flow conditions on LE skin temperature, PSM (Runs 26, 58).</td>
</tr>
<tr>
<td>65.</td>
<td>Effect of hot air temperature on LE skin temperature, PSM (Runs 26, 62).</td>
</tr>
<tr>
<td>66.</td>
<td>Effect of piccolo mass flow at high temperature on LE skin temperature, PSM (Runs 26, 42, 66).</td>
</tr>
<tr>
<td>67.</td>
<td>Effect of piccolo mass flow at low temperature on LE skin temperature, PSM (Runs 62, 76).</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>68.</td>
<td>Effect of piccolo configuration on LE skin temperature, PSM (Runs 26, 107).</td>
</tr>
<tr>
<td>69.</td>
<td>Comparison of total temperature at the piccolo centerline, FSM (Runs 26, 58).</td>
</tr>
<tr>
<td>70.</td>
<td>Comparison of total temperature at the piccolo centerline, FSM (Runs 26, 62).</td>
</tr>
<tr>
<td>71.</td>
<td>Comparison of the total temperature at the piccolo centerline, FSM (Runs 26, 42, 66).</td>
</tr>
<tr>
<td>72.</td>
<td>Comparison of the total temperature at the piccolo centerline, FSM (Runs 62, 76).</td>
</tr>
<tr>
<td>73.</td>
<td>Comparison of the total temperature at the piccolo centerline, FSM (Runs 26, 107).</td>
</tr>
<tr>
<td>74.</td>
<td>Comparison of temperature contours on the thermocouple face, PSM.</td>
</tr>
<tr>
<td>75.</td>
<td>Comparison of Mach number contours for a hot-air jet, PSM.</td>
</tr>
</tbody>
</table>
LIST OF ABBREVIATIONS

Al   Aluminum
AFM  Aircraft Flight Manual
AMG  Algebraic Multigrid
Appendix C  FAA Icing Certification Envelope
AOA  Angle of Attack
CFD  Computational Fluid Dynamics
DES  Detached Eddy Simulation
DNS  Direct Numerical Simulation
FAA  Federal Aviation Administration
FAR  Federal Aviation Regulations
FAS  Full Approximation Storage
FSM  Full-Span Model
FSTI Freestream Turbulence Intensity
GA   General Aviation
HAARP Hot Air Anti-icing Research Program
HFG  Heat Flux Gage
ILS  Instrumented Landing System
IL   Inner-Liner
IPS  Ice Protection System
IRT  NASA Glenn Icing Research Tunnel
JAR  Joint Aviation Requirements
LE   Leading Edge
LES  Large Eddy Simulation
LWC  Liquid Water Content
NASA National Aeronautics and Space Administration
NS, N-S Navier-Stokes Equations
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>NTSB</td>
<td>National Transportation Safety Board</td>
</tr>
<tr>
<td>PSM</td>
<td>Partial-Span Model</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier-Stokes Equations</td>
</tr>
<tr>
<td>RK</td>
<td>Runge-Kutta</td>
</tr>
<tr>
<td>RTD</td>
<td>Resistance Thermal Detector</td>
</tr>
<tr>
<td>S-A, SA</td>
<td>Spalart-Allmaras Turbulence Model</td>
</tr>
<tr>
<td>SLD</td>
<td>Supercooled Large Droplet</td>
</tr>
<tr>
<td>SMC</td>
<td>Second Moment Closure</td>
</tr>
<tr>
<td>PSM</td>
<td>Partial Span Model</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport $\kappa-\omega$ Turbulence Model</td>
</tr>
<tr>
<td>TIPS</td>
<td>Thermal Ice Protection System</td>
</tr>
<tr>
<td>WSU</td>
<td>Wichita State University</td>
</tr>
</tbody>
</table>
LIST OF SYMBOLS

$\delta_{ij}$  Unit Tensor

$\varepsilon$  Turbulent Energy Dissipation Rate

$\kappa$  Thermal Conductivity, Turbulent Kinetic Energy

$\mu$  Molecular Viscosity

$\mu_t$  Turbulent Viscosity

$\theta$  Piccolo Orifice Circumferential Location

$\rho$  Density

$\tau_{ij}$  Stress Tensor

$\nu$  Turbulent Kinematic Viscosity

$\tilde{\nu}$  Modified Turbulent Kinematic Viscosity

$\omega$  Dissipation per unit Turbulent Kinetic Energy

$C_p$  Specific Heat at Constant Pressure

$C_v$  Specific Heat at Constant Volume

$D$  Jet Diameter

$D_{\omega}$  Cross-diffusion Term

$F$  External Force Term

$G_b$  Production of Turbulent Kinetic Energy due to Buoyancy

$G_k$  Production of Turbulent Kinetic Energy due to Mean Velocity Gradients

$G_{\omega}$  Production of Turbulent Dissipation due to Mean Velocity Gradients

$G_{\nu}$  Production of Turbulent Kinematic Viscosity

$h$  Enthalpy or Surface Heat Flux

$I$  Internal Energy

$m$  Piccolo Mass Flow
LIST OF SYMBOLS (continued)

 OD  Piccolo Outer Diameter
 P  Pressure
 P_0, P_{tot}  Total Pressure
 r  Piccolo Radius, Radial Direction, Recovery Factor
 Re_{at}  Transition Onset Momentum Thickness Reynolds Number
 S  Source Term
 s  Surface Distance along LE
 T_d^j  Jet Nozzle Dynamic Temperature
 T_{picc}  Total Temperature at Piccolo Tube Inlet
 T_s  Static Temperature
 T_s^j  Jet Nozzle Static Temperature
 T_{s,\infty}  Freestream Static Temperature
 T_{b, T_{tot}}  Total Temperature
 T_{L,\infty}  Freestream Total Temperature
 T_u  Turbulence Intensity
 x/c  Chordwise-normalized X-coordinate
 V_{\infty}  Freestream Airspeed
 y'  Distance in Wall Coordinates, \frac{\rho' \cdot y' \cdot H_i}{\mu}
 Y_M  Dilatation Dissipation
 Y_{\omega}  Destruction of Dissipation
 Y_{\omega}  Destruction of Turbulent Viscosity due to Wall Blocking, Viscous Damping
 y/c  Chordwise-normalized Y-coordinate
 z_n  Nozzle-to-plate Distance
CHAPTER 1
INTRODUCTION

1.1 Background

Aircraft icing remains a serious aviation concern as the number of commercial airlines, regional business jets, and general aviation aircraft continues to increase steadily around the world. Most aviation icing accidents occur due to ice accumulation on the aircraft aerodynamic surfaces such as wings and horizontal/vertical stabilizers. Ice accretion leads to considerable deterioration in performance and handling qualities, thus compromising safety.

According to The National Transportation Safety Board (NTSB), the last three commercial passenger air carriers that experienced an uncontrolled collision with terrain due to icing were a Yakovlev YAK-40 belonging to Vologodskiy Airlines in Moscow, Russia in 2000, a Bombardier CRJ-200LR belonging to China Eastern Airlines in Baotou, China in 2004, and a Bombardier CL-600-2B19 belonging to Belavia in Yerevan, Armenia [1]. Table 1 provides a brief summary of the most relevant aircraft accidents documented by the NTSB since 1982 in which ice accretion on an aerodynamic surface was one of the primary probable causes. The cases considered only include general aviation business jets and commercial air carriers. General aviation single-piston and single-turboprop airplanes are not included, although statistically they amount to the largest number of icing accidents [1].

The last accident in the United States studied and documented with a probable cause report by the NTSB occurred in February 2005 when a Cessna Citation 560 belonging to Circuit City Inc. crashed during an ILS approach in freezing rain to the Pueblo Memorial Airport in Pueblo, Colorado. According to the accident report [2], the probable cause was the formation of rime ice on the wing’s leading edges during descent and the crew’s failure to increase the referential speed while operating in icing conditions, contrary to company procedures and manufacturing guidelines. The report cites the Citation 560 Aircraft Flight Manual (AFM): “In icing conditions, a small amount of residual ice forms […]. This is normal, but can cause an increase in stall speeds. When any amount of residual ice is visible the stall speed increases by 8 knots” [2]. While on approach, the airplane’s left wing suddenly stalled causing the aircraft to roll left and auto rotate. The down-going wing experienced a local increase in angle of attack (AOA)
and went deeper into the stall. The up-going wing had a local decrease in AOA, increasing its stall margin. The airplane collided with the ground a few seconds later. The anti-icing systems were activated on the wing inboard leading edges and on the engine nacelle inlets. The deicing boots were activated on the wing outboard leading edges and the horizontal tail leading edge [2].

This was not the first time a Citation 560 was involved in an icing-induced collision. During flight testing, three Citations crashed during landing approach in a two-month interval: the first one in Wisconsin on Dec. 30, 1995, the second one in Germany on Jan. 2, 1996, and the third one in Austria on Feb. 19, 1996. The three aircraft accumulated ice ridges between 2 and 4 mm [2] on the wing leading edge and stalled. Two of them rolled abruptly to the left and one to the right. All three were uncontrollable and unresponsive to aileron input after stalling [2].

Table 1. Business, regional, and transport aircraft icing accidents since 1982 [1]

<table>
<thead>
<tr>
<th>Year</th>
<th>Site</th>
<th>Air Carrier</th>
<th>Aircraft</th>
<th>Phase</th>
</tr>
</thead>
<tbody>
<tr>
<td>1982</td>
<td>U.S.A.</td>
<td>Air Florida</td>
<td>Boeing 737–222</td>
<td>Takeoff</td>
</tr>
<tr>
<td>1985</td>
<td>Canada</td>
<td>Arrow Air</td>
<td>McDonnell-Douglas DC-8-63CF</td>
<td>Takeoff</td>
</tr>
<tr>
<td>1985</td>
<td>Russia</td>
<td>Aeroflot</td>
<td>Tupolev TU-134A</td>
<td>Landing</td>
</tr>
<tr>
<td>1987</td>
<td>U.S.A.</td>
<td>Continental Airlines</td>
<td>McDonnell-Douglas DC-9-14</td>
<td>Takeoff</td>
</tr>
<tr>
<td>1989</td>
<td>Canada</td>
<td>Air Ontario</td>
<td>Fokker F-28</td>
<td>Takeoff</td>
</tr>
<tr>
<td>1991</td>
<td>Russia</td>
<td>Aeroflot</td>
<td>Antonov AN-12B</td>
<td>Landing</td>
</tr>
<tr>
<td>1991</td>
<td>Russia</td>
<td>Tartarstan Airlines</td>
<td>Antonov AN-24</td>
<td>Landing</td>
</tr>
<tr>
<td>1991</td>
<td>U.S.A.</td>
<td>USAir</td>
<td>Fokker F-28</td>
<td>Takeoff</td>
</tr>
<tr>
<td>1993</td>
<td>Macedonia</td>
<td>Palair Macedonian</td>
<td>Fokker 100</td>
<td>Takeoff</td>
</tr>
<tr>
<td>1994</td>
<td>U.S.A.</td>
<td>American Eagle</td>
<td>ATR 72-212</td>
<td>Landing</td>
</tr>
<tr>
<td>1994</td>
<td>Russia</td>
<td>North Western Air Transport</td>
<td>Antonov AN-12B</td>
<td>Landing</td>
</tr>
<tr>
<td>1994</td>
<td>Italy</td>
<td>Romanian Banat Air</td>
<td>Antonov AN-24B</td>
<td>Takeoff</td>
</tr>
<tr>
<td>1997</td>
<td>U.S.A.</td>
<td>Comair Airlines</td>
<td>Embraer EMB120-Brasilia</td>
<td>Landing</td>
</tr>
<tr>
<td>1998</td>
<td>Canada</td>
<td>Private</td>
<td>CRJ-200LR</td>
<td>Takeoff</td>
</tr>
<tr>
<td>1999</td>
<td>Turkey</td>
<td>Turkish Airlines</td>
<td>Boeing 737-400</td>
<td>Takeoff</td>
</tr>
<tr>
<td>2000</td>
<td>Russia</td>
<td>Vologodskiye Airlines</td>
<td>Yakovlev YAK-40</td>
<td>Takeoff</td>
</tr>
<tr>
<td>2002</td>
<td>U.K.</td>
<td>Private</td>
<td>Bombardier CL-600-2B16</td>
<td>Takeoff</td>
</tr>
<tr>
<td>2004</td>
<td>U.S.A.</td>
<td>Private</td>
<td>Bombardier CL-600-2A12</td>
<td>Takeoff</td>
</tr>
<tr>
<td>2004</td>
<td>China</td>
<td>China Eastern Airlines</td>
<td>Bombardier CRJ-200LR</td>
<td>Takeoff</td>
</tr>
<tr>
<td>2005</td>
<td>U.S.A.</td>
<td>Private</td>
<td>Cessna Citation 560</td>
<td>Landing</td>
</tr>
<tr>
<td>2006</td>
<td>China</td>
<td>China PLA Air Force</td>
<td>KJ-200</td>
<td>Landing</td>
</tr>
<tr>
<td>2007</td>
<td>Russia</td>
<td>Private</td>
<td>Bombardier CL-600-2B16</td>
<td>Takeoff</td>
</tr>
<tr>
<td>2008</td>
<td>Armenia</td>
<td>Belavia</td>
<td>Bombardier CL-600-2B19</td>
<td>Takeoff</td>
</tr>
</tbody>
</table>
The type of stall pattern described for the Cessna Citation 560 is characteristic for most accidents involving ice accumulation on the wings of relatively small and mid-size business jets. In 2004, a Bombardier CL-600-2A12 crashed after takeoff in Montrose, Colorado (Figure 1). The attempted takeoff was executed during prevailing icing conditions, including snow showers. The NTSB Accident Report [3] specifies that the crew performed a takeoff with the wing anti-icing protection activated and in heavy snow with a wet runway. In this case, the aircraft lifted off, climbed to approximately 50 feet, stalled prematurely (with an angle of attack less than 24 degrees, the angle at which the stall warning activates with 20º flap deflection), and rolled left abruptly (roll rate over 50 deg/sec). Information provided by Bombardier Aerospace during the investigation suggested that the aircraft stall lift coefficient might have decreased by as much as 33% due to the ice/snow contamination on the wing upper surface. Other similar crashes involving a Bombardier CL-600-2B16 occurred in Birmingham, U.K. in 2002 [4], in Moscow, Russia in 2007 (See Figure 2) [5], and in Yerevan, Armenia in 2008 [6].

Figure 1. Wreckage of Bombardier CL-600-2A12, Montrose, Colorado, 2004. [1]
In general, ice accumulation on a wing produces a significant reduction in lift and an increase in drag. This translates into an increased stall speed. Business jet aircraft with supercritical wings are particularly susceptible to icing due to their asymmetric stall characteristics at low speeds [7], which make them roll abruptly. Supercritical wings are thin and have a suction peak very close to the leading edge. The wings in these aircraft, which in most cases do not have high-lift leading edge devices, experience a separation pattern in which a circulating bubble develops on the upper surface near the leading edge, as shown in Figure 3. As the angle of attack increases, the bubble bursts and the flow completely detaches from the wing, causing a sudden loss in lift that usually starts at the mid-span and rapidly spreads towards the wing tip inducing loss of aileron control. This process is not accompanied by a natural buffet that alerts pilots of an imminent stall as is the case with trailing edge separation (common in thick cambered airfoils or in thin airfoils with leading edge devices) [7, 8].

Figure 2. Wreckage of Bombardier CL-600-2B16 in Moscow, Russia, 2007. [5]
The Federal Aviation Administration (FAA) requires airplane manufacturers to demonstrate that their aircraft can fly safely in icing conditions as defined in the Federal Airworthiness Regulations (FAR/JAR) Part 23 and 25, Appendix C. This environmental envelope identifies a demanding variety of icing conditions for which aircraft must be certified. In-flight icing can occur in several scenarios such as ice crystal clouds, mixed conditions, and/or freezing rain, drizzle or fog. For example, ice accumulates on the airframe when an aircraft flies through clouds in which moisture has frozen to the solid or crystal stage. Another type of icing environment includes mixed conditions where super cooled droplets are suspended and the ambient temperature is below freezing. A particular type of droplet, called Supercooled Large Droplet (SLD), is very frequent in the event of freezing rain. This droplet, which splashes and freezes upon impingement, has received most attention in the past 20 years owing to the fact that it creates icing conditions outside the envelope specified in the Appendix C [2]. The Cessna Citation 560 that crashed in Colorado in 2005 is suspected to have encountered this type of droplets during descent [2].

Icing certification requires the design, analysis, and testing of several components including ice protection systems (IPS). Most general aviation (GA) and transport aircraft are protected against ice accretion on several critical surfaces such as wing, horizontal and vertical tail leading edges, and engine nacelle inlets as shown in red in Fig. 4. In general, two types of ice protection systems (IPS) are used on aircraft. Mechanical systems such as pneumatic boots, or electro-expulsive systems, which rely on
surface deformation to break the ice [9, 10]. These systems allow a small amount of ice to accrete prior to ice removal which is done in a cyclic fashion. Ice expelled from the leading edge (LE) can still accumulate behind the protected section forming a ridge in front of the flaps and ailerons.

Thermal ice protection systems (TIPS) such as hot air or electro-thermal systems use heat to prevent or minimize ice accretion on protected surfaces. Bleed air systems supply high temperature air to a piccolo tube installed inside the wing, tail, or inlet leading edge. Hot air jets from the piccolo are directed to the leading edge interior skin to supply the required heat for ice protection. The hot air is typically exhausted through vents on the wing tips [11].

Anti-icing systems are usually classified as evaporative or running wet. Evaporative systems use high heat fluxes to evaporate the impinging droplets so that both the protected areas and the surfaces downstream remain dry and free of ice buildup. Surface temperatures are typically above 120ºF [12]. However, the new generations of high bypass ratio turbofan engines have difficulties providing sufficient bleed air for evaporative systems. In these cases, running wet systems are used. Running wet systems have lower thermal requirements and maintain surface temperatures between 40 and 50ºF [12]. They prevent ice formation, but do not evaporate all of the impinging water, thus runback ice may form behind the protected region.
The design and optimization of these systems is a complex endeavor due to the large number of geometric and air flow (external and internal) factors that affect system performance and efficiency [13, 14]. The parameters listed below are typically considered along other design constraints such as system weight and bleed air availability at various flight regimes and engine throttle settings.

- Piccolo tube design including the size and cross section of the piccolo, the number of piccolo holes, diameter of holes, pitch, circumferential placement and pattern.
- Piccolo tube vertical and horizontal placement within the leading edge.
- Leading edge geometry and diffuser bay shape.
- Wing skin thickness which affects heat distribution through conduction.
- Bleed air temperature.
- Bleed air mass flow.
- Heat losses through the system components (e.g. the inner liner and the wing skin in the chordwise direction).
- External flow icing conditions including angle of attack, air speed, air temperature and pressure, liquid water content (LWC), cloud droplet size and distribution.

Nowadays, the increasing availability of sophisticated and robust computer tools allows engineers to conduct multiple parametric studies on complex anti-icing system designs with improved quality and efficiency. Industrial practices include models for each physical phenomenon (heat transfer, water impingement, etc.) in an anti-ice bleed model suite of codes that are used in conjunction with empirical data from wind tunnel tests and flight testing. For example, references [14] and [15] summarize EMBRAER and AIRBUS practices, respectively.

Potential flow codes such as LEWICE [16], CANICE [17] or ONERA [18] are typically used to study ice accretion under various atmospheric conditions, whereas Navier-Stokes computer codes are employed to evaluate the performance of anti-icing systems under a wide range of internal and external flow conditions. CFD codes, however, need to be validated and calibrated against experimental data to support the development of 3D CFD tools for bleed air system analysis.
WSU and NASA with support from the Cessna Aircraft Company in Wichita, KS, initiated a multi-year research effort named Hot Air Anti-Icing Research Program II (HAARP II) in 2004 with the following key objectives [13, 19, 20].

1. Design and fabricate a two-dimensional wing model with a 60-in chord section representative of modern supercritical business aircraft for testing at the NASA Glenn Center Icing Research Tunnel (IRT).

2. Instrument the model to measure skin temperature and internal flow thermal and pressure properties.

3. Conduct a variety of parametric studies in the NASA IRT facility to evaluate system performance and to develop an experimental database for the validation of thermal analysis tools.

4. Conduct numerical simulations with commercial CFD software to develop a computational methodology for dry and wet simulations, and to validate the results against an experimental database.

This thesis presents a succinct compilation of the numerical investigation conducted as part of HAARP II [20]. The wind tunnel experiments are simulated with the state-of-the-art CFD software, FLUENT, in order to develop an appropriate computational methodology including grid resolution, spatial discretization, viscous flow analysis, and boundary conditions selection for bleed air system design and analysis. Experimental data available from icing tunnel experimental performed by WSU are used as a benchmark for comparison and evaluation of the analysis results.

1.2 Literature Review

Between the mid-1990’s and today, several computational studies of the physical mechanisms that affect the energy transfer from hot air jets to the wing skin in a bleed air system have been published. These efforts have investigated computational methodologies to support the design, development, and validation of CFD simulation tools. A brief review of some of these research efforts pertaining to two- and three-dimensional CFD simulations of bleed air anti-icing systems is provided below.

In 1997, Smith and Taylor [21] simulated in 2D a Jetstream 41 engine intake anti-icing system in dry (no water impingement) and wet conditions with the commercial code PHOENICS [22]. Source terms
were included in the energy equation to account for the cooling effect of water impingement. The melting and freezing processes were considerably simplified and heat transfer between the water impinging on the surface and the airflow was neglected. Nevertheless, comparison with flight test data demonstrated good agreement and the model was used to support airworthiness certification.

In 1998, Croce et al. [23] introduced FENSAP-ICE, a 2D/3D commercial finite-element based Navier-Stokes solver capable of modeling conjugate heat transfer, droplet impingement and ice accretion. A wing slat element and a nacelle inlet equipped with hot air anti-icing systems were analyzed in dry air conditions employing a standard $\kappa - \varepsilon$ turbulence model. The results were reported to be satisfactory, but no validation against experimental data was provided. In 2002, Croce, Beaugendre, and Habashi [24] expanded the usage of FENSAP-ICE to the dry and wet simulation of an engine nacelle reporting acceptable and validated temperature predictions for the dry simulation. Temperatures for the wet simulation were provided without any validation. In 2007, Habashi et al. [25] documented the development of a wet simulation for a wing hot air anti-icing system. In this simulation, the external flow around the aircraft, the impinging water film, and the internal hot air were calculated separately. The code employed a method called by the authors “loose-coupling conjugate heat transfer approach”, in which the energy equation in multiple computational domains were coupled and thermal equilibrium was reached through the iterative exchange of boundary conditions. The authors presented results qualitatively, but no quantitative results or experimental validation was provided at that time due to the proprietary nature of the engineering case illustrated.

In 2000, de Mattos and Oliveira [26] from EMBRAER used FLUENT [27] and the Spalart-Allmaras turbulence model to perform a 3D conjugate heat transfer analysis on an anti-icing wing slat in dry air conditions. The conjugate heat transfer approach consisted of the coupling of the internal and external convection along the slat wall surface, and the conduction through the solid wall. The effort also looked at the effect of different piccolo-hole configurations on the global efficiency of the system. The results demonstrated qualitatively the effect of hot air mass flow at the piccolo tube inlet holes on the surface heat transfer and skin temperature.

In 2001, Wang [28] conducted numerical simulation of anti-icing systems using 3 solvers: CFD-ACE [29], FLUENT, and SHARP [30], to study the effects of numerical schemes, grid resolution, near-wall
treatment, and turbulence models on skin heat transfer coefficients. His conclusions emphasized the necessity of using a high grid resolution near walls and selecting an appropriate turbulence model (either a low Reynolds number model such as Spalart-Allmaras or a two-layer zonal model such as $\kappa - \varepsilon$ and $\kappa - \omega$).

In 2004, Liu and Hua [31] analyzed a wing anti-icing system in dry conditions with FLUENT. The study was conducted with conjugate heat transfer through the wing skin. Surface temperatures compared acceptably against flight test data. The same authors modeled the unsteady development of the bleed air flow for the same system [32] in 2007. In this case, the comparison of the time accurate temperature distribution at half-span showed satisfactory agreement with flight test data.

In 2005, Planquart, Vanden Borre, and Buchlin [33] conducted 3D simulations with FLUENT to map heat transfer coefficients in a multi-jet anti-icing system. They employed the realizable $\kappa - \varepsilon$ turbulence model with high resolution near walls and reported finding satisfactory agreement between the numerical and experimental normalized surface heat transfer coefficients measured using infrared thermography, although no quantitative comparisons were provided.

In 2006, Lee, Rigby, Wright, and Choo [34] evaluated the applicability of NASA’s turbulent heat transfer Navier-Stokes code, GLENN-HT [35], by comparing results from this code, FLUENT, and a experimental icing assessment conducted in the IRT for a swept wing (NACA 23014 airfoil) with an inner-liner anti-icing system. The numerical simulations used the Spalart-Allmaras (FLUENT only) and standard $\kappa - \omega$ (both FLUENT and GLENN-HT) turbulence models. The turbulence intensity specified for the piccolo holes inflow boundary was estimated at 1%, although no particular reason for this selection was specified. The study also looked into differences between fully structured and unstructured grids concluding that tetrahedral grids were more dissipative and therefore needed higher resolution, especially in cases when the near-wall grid was not hexahedral or prismatic. The final objective was to develop a procedure to integrate the internal heat transfer predictions from FLUENT and Glenn-HT with LEWICE, the ice-accretion potential flow code developed by NASA. The same year, Rigby [36] studied with GLENN-HT alternative piccolo patterns to the diamond one to increase surface heat flux on the wing leading edge using a fully structured grid and the standard $\kappa - \omega$ turbulence model with 10% turbulence.
intensity for the inflow jet holes. The results showed a significant improvement in the minimum amount of heat transfer along the leading edge.

Papadakis and Wong [13] also conducted experimental and numerical parametric studies of a bleed air system in 2006. Wong [37] also reported results from the same research effort in 2004. Their study explored internal flow behavior and methods to enhance heat transfer to the wing skin including piccolo tube location, piccolo-hole pattern, and the inclusion of an inner liner. The numerical studies included a turbulence model comparison between the Spalart-Allmaras and the standard $\kappa - \omega$ turbulence models, and grid resolution evaluation for the hot air jet regions. The final parametric studies were conducted with the Spalart-Allmaras turbulence model. The numerical results were reported to be in good agreement with the experimental trends. A high grid resolution in the jet region was determined to be critical to the quality of the numerical results.

In 2007, Saeed and Al-Garni [38] also used FLUENT along with the realizable $\kappa - \varepsilon$ turbulence model to simulate surface heat transfer and record Nusselt number distributions on a wing skin for a variety of jet arrangements including a single array of jets and two staggered arrays at different angles. The geometry represented only the internal region of the wing. External cooling was not accounted for. Saeed and Al-Garni also studied the effect of an inner liner configuration concluding that it improved surface heat transfer substantially in agreement with the results reported by Papadakis and Wong [13].

Finally, in 2008, de Santos, Domingos, Maria, and Leal [14] presented a sensitivity analysis of a bleed air anti-icing system using a suite of CFD and panel method icing codes typically used at EMBRAER. They concluded that, based on a quadratic surface response, system efficiency is more sensitive to the hot air temperature than it is to the mass flow for a single flight condition.

Over the years, extensive numerical studies have been conducted to develop and validate thermal analysis tools including commercial CFD software packages for hot-air anti-icing modeling. In general, the preferred approach for numerical modeling involves the definition of a small section of the wing leading edge and an examination of the temperature locally at that spanwise station.

In most cases, there is a lack of unclassified or non-proprietary experimental data to support the results published. Only qualitative or normalized results can usually be presented without infringing trade
regulations. In addition, a review of different publications indicates a wide variety of approaches for the specification of boundary conditions and selection of turbulence models for the simulations.

1.3 Objectives

This thesis presents computational results from HAARP II obtained with the commercial CFD software FLUENT. The specific objectives of the work performed were as follows.

(1) Develop a computational methodology for the analysis of hot air ice protection systems in dry air conditions using the FLUENT computer code.

(2) Perform an evaluation of five eddy-viscosity turbulence models available in FLUENT to select a suitable turbulence model for accurate simulation of the external wall-bounded flow and the internal jet impinging flow based on comparisons with experimental data.

(3) Assess the effect of grid resolution, spatial discretization, and boundary conditions on the accuracy of the numerical results.

(4) Conduct a numerical parametric study to evaluate system performance for a range of external and internal flow conditions including freestream velocity and temperature, hot piccolo air temperature and mass flow, jet circumferential placement (inclination of the air jet with respect to the chord line).

(5) Compare the analysis results with experimental data from icing tunnel tests with a highly instrumented bleed-air wing model.

1.4 Technical Approach

The technical approach for the analysis of hot-air anti-icing systems included the development of suitable computational models and multiple preliminary studies to identify an appropriate turbulence model, grid resolutions level, spatial discretization scheme, and set of boundary conditions.

Two computational models were idealized to simulate an anti-icing system in a wing leading edge: a full-span model (FSM) and a partial-span model (PSM). The FSM was intended to model the entire span of a wing in an icing tunnel test section to investigate the development of spanwise flow inside the piccolo tube. On the other hand, the PSM was developed to model a 2.44-inch spanwise section of the wing with a finer grid resolution than the FSM in order to investigate internal and external flow properties at a particular spanwise station. The PSM was not intended to model the piccolo tube internal
flow; instead flow parameters at the piccolo holes exits in the FSM were used to specify inflow boundary conditions for the PSM. A boundary condition study was performed with the FSM and PSM in order to identify an appropriate method to extract inflow boundary conditions for the PSM. Two approaches were applied to obtain the boundary conditions from the FSM piccolo holes: centerline and cell-averaged. Chapter 2 provides a more detailed description of the anti-icing computational models.

The first preliminary study conducted was a viscous flow analysis. This study was used to identify a turbulence model that could provide consistent predictions of recovery temperatures at an impingement wall. A 2D axisymmetric, subsonic, heated jet impinging on a flat plate was the selected scenario to test five eddy-viscosity turbulence models available in FLUENT. Available experimental data was used to evaluate the results.

Next, a grid resolution and spatial discretization study was performed with a three-dimensional version of the jet impingement scenario employed in the viscous flow analysis, and first- and second-order spatial discretization schemes. Three grid resolution levels were considered based on the number of nodes distributed around the nozzle exit circumference. The conclusions of the grid resolution study were applied in the PSM grid generation.

Finally, a parametric study was conducted with the anti-icing models (FSM and PSM) to validate the computational methodology by simulating several cases from icing tests performed at the NASA Glenn Icing Research Tunnel (IRT) by WSU.
2.1 Computational Models

A numerical simulation of bleed air anti-icing configurations consists of multiple mathematical regions. The three basic ones are: the internal hot-air flow, the external cold-air flow (dry or wet), and the wing skin solid that separates the two fluid regions. A runback water region is also considered for wet external flow. Similarly, two additional regions (piccolo interior flow and piccolo tube skin solid) can be introduced to simulate spanwise piccolo heat loss.

The computational methodology used to simulate a bleed air anti-icing system operating in a dry external cooling condition (no water impingement) involves solving the conjugate heat transfer problem with a Navier-Stokes code. The flow properties that satisfy all the equations used to model the physical phenomena are determined through an iterative process. This is accomplished by coupling the fluid and solid solvers so that energy is conserved as it transfers among the regions inside the wing anti-icing system. Conduction and convection are the dominant heat transfer mechanisms.

Simulation of a bleed air anti-icing system operating in a wet external cooling condition is a complex task which requires evaluating the droplet collection rate and computing the water film that forms as the droplets coalesce on the wing surface, alongside solving the conjugate heat transfer problem. Simulating the thin water film requires the evaluation of mass flow and energy balances coupled with the flow solution. The energy balance must consist of external heat convection, anti-icing heat conduction, energy advected by impinging droplets, evaporation, sublimation, radiation, water runback, and ice accretion. Furthermore, during the anti-icing system operation time, portions of the water film turn into runback ice, thus making the modeling unsteady in time.

In general, there are two approaches to the simulation. The first one is a complete analysis of a full-length piccolo tube and all of its small-diameter jet holes. This approach allows an accurate geometric representation of the test article and also reduces the number of piccolo-related boundary condition to a single heat input value at the piccolo inlet. However, simulating a full-span wing anti-icing system to
obtain detailed skin temperature distributions requires using high resolution grids, which is computationally very expensive and is not often practical even with today's computing power.

An alternative approach is to model a spanwise segment containing only the interior hot air region, the external flow region, and the solid skin region. Local spanwise effects are assumed to be negligible. Symmetry or periodic conditions are applied at each span end boundary. Additional boundary conditions are required at the piccolo tube exterior surface and at the piccolo jet holes. This presents a challenge as temperatures and pressures at piccolo jet holes are not typically recorded experimentally to prevent any instrumentation from obstructing the jet development.

The CFD studies presented in this thesis used a cost-effective combination of the approaches described above. A 72-inch full-span model (FSM) and a 4.88-inch partial-span model (PSM) were used in the computations performed with the commercial CFD code FLUENT. An experimental test article, shown in Fig. 5, was reproduced. It is a 72-inch span and 60-inch chord business jet wing model located in the test section of the NASA Icing Research tunnel test section [13]. More details are provided in Section 3.1.

The geometric specifications for the two computational models are listed in Table 2. The FSM provided pressure and temperature conditions at the piccolo jet exits and at the piccolo tube exterior surface. Wing skin temperatures were not obtained from the FSM. The values extracted from the FSM were used in the PSM, allowing a considerably higher grid resolution (particularly in the jet regions), lower storage requirements, and reduced computational effort to obtain a converged solution.

The FSM, shown in Fig. 6, was modified to reduce the meshing efforts and the grid cell density. It included a partial tunnel (6-ft high x 9-ft wide x 2.33-ft long) and a truncated wing chord. The whole computational domain was terminated at a downstream plane normal to the end of the diffuser passage formed between the wing skin and the inner-liner skin. The circular 0.052-inch diameter piccolo holes in the IRT test model were converted to a set of 0.046-inch x 0.046-inch square holes to reduce meshing effort. The exit area of each square hole was the same as the area of the circular holes. The piccolo jet pattern was modified from a diamond type to three-in-line such that the +45°, 0° and -45° holes were at the same planes normal to the piccolo centerline axis, as shown in Fig. 6 (b). The spacing (pitch) in the
three-in-line hole arrangement was kept at 2.44-inch, which is the same distance between adjacent two-hole stations in the IRT wing model.

![Figure 5. HAARP II wing model in icing tunnel test section [13]](image)

<table>
<thead>
<tr>
<th>Span length (inch)</th>
<th>AOA (deg)</th>
<th>Piccolo hole angular placement</th>
<th>Total number of piccolo holes modeled</th>
<th>Piccolo hole pattern</th>
<th>Piccolo interior/skin modeling</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Full-span model</strong></td>
<td>72</td>
<td>3</td>
<td>+45°/0°/-45°</td>
<td>84</td>
<td>3-in-line (28 x 3)</td>
</tr>
<tr>
<td><strong>Partial-span model</strong></td>
<td>4.88</td>
<td>3</td>
<td>+45°/0°/-45°, +70°/10°/-40°</td>
<td>6</td>
<td>Diamond (1/2-2-1-2-1/2)</td>
</tr>
</tbody>
</table>
The PSM (Fig. 7) consisted of a full-chord wing model and a longer tunnel length (4.88-inch high x 108-inch wide x 3,072-inch long). The tunnel height and the wing span in the PSM were set to 4.88-inch (twice the piccolo jet hole pitch). Six circular 0.052-inch diameter jet holes were modeled on the piccolo tube surface, arranged in a diamond pattern. Only half a hole was modeled at each spanwise end of the computational domain as required for symmetry boundary conditions (Fig. 7b). The interior of the piccolo tube was not modeled and inflow boundary conditions were specified at the piccolo holes. Figure 8 shows a cross-sectional diagram of the wing leading edge interior. The hot air domain in the PSM was terminated at the end of the diffuser passage that was formed between the wing and inner-liner skins while the 0.1-inch thick LE wing skin was extended to the trailing edge of the wing. Note that in all the experiments referenced, Teflon® inserts were used to reduce the heat transfer between the LE and the wing main body. As a result the wing skin in the PSM was divided into five parts to allow the specification of low thermal conductivity in the regions where the insulation inserts were located (Fig. 7a). These insert regions were positioned at between 13.5% and 14.5% chord of the wing on both upper and lower surfaces. Grid generation details are described in the following section.
2.2 Grid Generation

Gambit [39], a grid generation package, was used to create the computational grids presented. Triangular and quadrilateral elements (quads) are available in Gambit for surface meshing while hexahedral, tetrahedral, wedges, and pyramidal elements are used for volume meshing.
The three-dimensional grids for both the PSM and FSM included the IRT walls with the wing model centered inside the tunnel at AOA = 3º. For the FSM, a hybrid volume mesh consisting of hexahedra and wedges was generated by sweeping the triangles and quads of a 2D surface mesh across the span. The mesh, shown in Fig. 9, contained triangles and wedge/prism elements in the diffuser and external flow regions, and quads and hexahedral cells in the inner-liner passage, piccolo interior, wing skin, piccolo tube skin, and boundary layer regions. The total number of grid cells was significantly reduced by using large aspect ratio hexahedral and wedge elements stretched along the span, as shown in Fig. 10. A total number of 763 points were distributed across the 72-inch piccolo length with grid point clustering toward the jet planes. Five points were equally distributed across the width of each square piccolo jet hole. This coarse grid resolution was mainly intended to provide approximate flow parameters (total pressure and total temperature) for a boundary condition study and not to attempt to resolve all the physical features of the jet flow. The total number of cells, nodes and faces of the FSM are reported in Table 3.

Table 3. FSM grid size information

<table>
<thead>
<tr>
<th>Number of Cells</th>
<th>Number of Nodes</th>
<th>Number of Faces</th>
<th>$y_{average}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full-span model</td>
<td>15,156,180</td>
<td>14,360,738</td>
<td>44,225,796</td>
</tr>
</tbody>
</table>

For the PSM, two hybrid volume grids consisting of hexahedral, wedges, and tetrahedral elements were generated for the two different piccolo configurations: P45-0-M45 (Fig. 11) and P70-P10-M40. Grid resolution was selected based on studies reported in Chapter 3 and previous research efforts at Wichita State University [13, 37]. Tetrahedral elements were used mainly in the diffuser section and portion of the external flow region outside of the diffuser, while hexahedral and wedge elements were used in regions including the inner-liner passages and wing skin. Tetrahedral cells, sized at 0.003-in, were used to mesh the custom designed conical volumes containing the jets and jet impingement regions inside the diffuser. This provided a node distribution of 48 nodes along the circumference of each circular piccolo jet hole (or 18 nodes across the diameter). The size of the tetrahedral elements increased gradually to a maximum size of 0.05-inch in connecting the density jet regions to the rest of the diffuser.
bay volume. Grid density near walls was also high ($y^+_{\text{Average}} = 0.06$) to ensure proper resolution of the flow and of heat transfer properties. The PSM had 793,562 cells per cubic inch of diffuser volume. Figure 12 shows a cross-sectional view of the Partial Span Model at a spanwise station with one piccolo hole. The total number of cells, nodes and faces for the PSM grids are reported in Table 4.

Table 4. PSM grid size information.

<table>
<thead>
<tr>
<th></th>
<th>Number of Cells</th>
<th>Number of Nodes</th>
<th>Number of Faces</th>
<th>$y^+_{\text{Average}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Partial-span model</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(P45-0-M45)</td>
<td>13,039,161</td>
<td>5,055,499</td>
<td>30,251,163</td>
<td>0.06</td>
</tr>
<tr>
<td>Partial-span model</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(P70-P10-M40)</td>
<td>13,047,346</td>
<td>4,983,428</td>
<td>30,152,870</td>
<td>0.06</td>
</tr>
</tbody>
</table>

Figure 9. Isometric view of FSM.  
Figure 10. Section view of FSM.  
Figure 11. Isometric view of PSM.  
Figure 12. Section view of PSM.
2.3 Governing Equations

FLUENT is capable of modeling fluid flow and heat transfer in complex geometric domains. According to FLUENT's User Guide [27], it includes dynamic memory allocation, efficient data structures, and flexible solver control. Some of its main capabilities [27] are:

1. Two-dimensional and three-dimensional flow simulations over complex geometries
2. Incompressible or compressible flow conditions
3. Steady or transient analysis
4. Inviscid, laminar, or turbulent flow
5. Convective heat transfer, including natural or forced convection
6. Coupled conduction/convection heat transfer

The governing differential equations are the compressible Navier-Stokes equations, comprised of the conservation of mass, momentum, and energy as shown in tensor notation in Eqns. (1), (2), and (3), respectively.

Continuity equation:
\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = S_m
\] (1)

Momentum equation:
\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = - \frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i
\] (2)

Energy equation:
\[
\frac{\partial}{\partial t} (\rho E) + \frac{\partial}{\partial x_j} (u_j (\rho E + P)) = \frac{\partial}{\partial x_i} \left[ \kappa_{\text{eff}} \frac{\partial T}{\partial x_j} - \sum_j h_j J_{ij} + u_j (\tau_{ij})_{\text{eff}} \right] + S_b
\] (3)

The momentum equation includes terms for static pressure, the gravitational body force, and the external body forces. The stress tensor is given by:
\[
\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial u_i}{\partial x_i} \delta_{ij}
\] (4)

where \( \mu \) is the molecular viscosity and \( \delta_{ij} \) is the unit tensor.

The energy equation is used when modeling heat transfer and compressibility effects and includes terms for conduction, diffusion flux of species, and viscous dissipation. The energy term \( E \) is defined by:
\[ E = h - \frac{P}{\rho} + \frac{u_i^2}{2} = I + \frac{u_i^2}{2} \]  

(5)

where \( h \) is enthalpy and \( I \) is the internal energy.

The conservation equations provide 5 scalar equations for three-dimensional problems. The system of equations is closed by introducing thermodynamic relations and auxiliary relations for the coefficient of viscosity and thermal conductivity. The result is a closed system of 9 equations with 9 unknowns.

The relationships between the four thermodynamic variables \( (\rho, P, h, \text{ and } T) \) are obtained based upon the assumption of thermodynamic equilibrium, which specifies that the state of a substance can be described by means of only two equations of state. For a perfect or ideal gas, these equations are:

\[
\rho = f(P, T) = \frac{P}{RT} 
\]

(6)

\[
h = f(P, T) = \frac{P}{\rho} - c_v T 
\]

(7)

The three conservation equations have the same general form and can be expressed by a general conservation equation for transport of a scalar quantity as shown in Equation (8). The variable \( \phi \) represents any scalar flow property and \( \Gamma \) is the diffusion coefficient.

\[
\frac{\partial (\rho \phi)}{\partial t} + \text{div} (\rho \phi \mathbf{u}) = \text{div} (\Gamma \nabla \phi) + S_\phi
\]

(8)

The transport equation defines -from left to right- the rate of change term, the convective term, the diffusive term, and the source term, respectively, for the property \( \phi \).

FLUENT uses a finite volume approach to convert the governing transport equations to algebraic equations that can be solved numerically. Equation (8) is integrated over a 3D control volume and the Gauss’s Divergence theorem is used to yield Equation (9), which is then discretized and linearized. The linearized equation is solved to obtain updated values of the dependent variable aided by initial and boundary conditions of the flow problem.

\[
\frac{\partial \left( \int_{CV} \rho \phi \, dV \right)}{\partial t} + \int_A \mathbf{n} \cdot (\rho \phi \mathbf{u}) \, dA = \int_A \mathbf{n} \cdot (\Gamma \nabla \phi) \, dA + \int_{CV} S_\phi \, dV
\]

(9)
2.4 Boundary Conditions

Boundary conditions indicate the flow and thermal variables on the boundaries of the physical domain including walls, inflow and outflow planes, and symmetry or periodic faces.

Wall boundaries mainly require specification of shear and thermal conditions as well as material type and thickness (for thin-wall simulations). Additional options for special problems include moving walls, chemical reactions, and radiation, for example. The no-slip boundary condition is the default shear condition for both internal and external viscous flow, but FLUENT also allows a specified shear stress, specularity coefficient for multiphase granular flow, or Marangoni stress for shear stresses caused by the variation of surface tension due to temperature [27]. The thermal condition, which is required to solve the energy equation, can be specified through any of these parameters: heat flux, temperature, convective heat transfer coefficient, and radiation emissivity. Interface walls (fluid/solid walls) are considered as “coupled” for conjugate heat transfer.

Flow inlet and exit conditions are selected depending on the expected flow properties. FLUENT provides several options [27]:

- **Pressure Inlet**: Used when the inlet total pressure is known, but the flow rate or velocity are unknown. It requires fluid total pressure, static pressure (only if the flow is supersonic), and other scalar properties of the flows (e.g. total temperature and turbulence model parameters). It is suitable for both incompressible and compressible flows.
- **Pressure Outlet**: Used for any flow regime (subsonic or supersonic). It requires a static pressure if the flow is subsonic as well as other scalar parameters in case of backflow.
- **Mass Flow Inlet**: Used to prescribe a known mass flow rate or mass flux distribution for compressible flows. A static pressure is needed for supersonic flows, while the total pressure is calculated from the interior solution.
- **Pressure Far Field**: Used to model a free-stream compressible flow at infinity. It requires the free-stream Mach number and static conditions. It is based on Riemann invariants [27].
- **Velocity Inlet**: Intended for incompressible flows only. FLUENT computes the mass flow based on the inlet velocity, area, and temperature. This boundary condition should not be placed too close
to a solid obstruction, since this could cause the inflow stagnation properties to become highly non-uniform.

- **Outflow**: Used to model flow exits where the details of the flow velocity and pressure are not known prior to the solution of the problem. FLUENT extrapolates all the variables from the interior. This condition cannot be used if the computational model (1) has a pressure inlet, (2) models compressible flow, and (3) models unsteady flows with varying density. In addition, this outflow condition can only be specified in regions where fully developed flow is expected.

- **Inlet vent**: Used to model an inlet vent with a specified loss coefficient, flow direction, and ambient (inlet) pressure and temperature.

- **Outlet vent**: Used to model an outlet vent with a specified discharge (i.e. pressure loss).

The various boundary conditions employed in the FSM and PSM are described below. A more specific description is provided in Section 3.5 (Boundary Condition Study).

**Full-span model (FSM)**: Boundary conditions included adiabatic walls on the four sides of the 6-ft x 9-ft x 2.33-ft tunnel, pressure inlet and pressure outlet conditions at the tunnel inlet and outlet boundaries, respectively. A 2D total pressure profile (in the direction of the y-axis shown in Fig. 6) and a 2D total temperature profile were specified at the 6-ft x 9-ft inlet of the tunnel, while a 2D static pressure profile was specified at the tunnel outlet. At the inlet of the piccolo tube, a pressure inlet boundary condition was used in which the total pressure and total temperature of the hot air entering the tube were specified. At the closed end of the piccolo tube, an isothermal (constant temperature) wall boundary condition was employed. At the exits of the upper and lower diffuser passages, shown in Fig. 8, pressure outlet boundary conditions were used where the static pressure of the hot air leaving the system was specified. The inner-liner skin was modeled as a zero thickness wall surface with a 2D chordwise heat flux profile applied to the rear (downstream end) surface of the wall. The flux profile was computed from the experimental data obtained from heat flux gages. A heat flux profile was used since available experimental inner-liner skin temperatures (measured at two spanwise stations of the wing model) were not sufficient for a full-span simulation. All wing surfaces at the two spanwise ends of the computational model were defined as adiabatic wall boundaries. Flow properties at the piccolo tube exterior and interior
Partial-span model (PSM): Symmetry conditions were applied at both spanwise ends of the computational domain of the 4.88-inch span wing model placed inside a 4.88-inch x 108-inch x 3,072-inch tunnel. The two side walls of the tunnel, facing the upper and lower surfaces of the wing model (model was installed vertically in the IRT), were defined as adiabatic. Total pressure and total temperature were specified at the tunnel inlet (pressure inlet condition) while static pressure was specified at the tunnel outlet (pressure outlet condition). An isothermal wall boundary condition was used at the piccolo surface. Total pressure and total temperature (pressure inlet condition) were specified at the piccolo jet holes. At the exits of the upper and lower diffuser passages, shown in Fig. 8, the static pressure of the hot air leaving the system was specified (pressure outlet condition). A wall boundary condition was specified at the zero thickness inner-liner skin. The wall temperature at the rear (downstream) surface of the inner-liner skin was specified from experimental measurements performed by WSU. The wall portion of the LE skin that extended from the inner-liner to the insulation insert (see Fig. 8) was modeled with a temperature profile specified from the experimental data obtained.

2.5 Numerical Schemes

FLUENT provides two numerical schemes to solve the nonlinear system of equations (Equations 1, 2, 3), and the integral equations for the turbulent scalars: pressure-based solver [40, 41] and density-based solver [42]. Both methods use a control-volume based technique and an iterative solution process to obtain a converged solution. The computational domain is divided into discrete control volumes. Then, integration on individual control volumes yields a set of algebraic equations for the discrete flow variables. These equations are linearized (implicitly or explicitly) and solved.

The pressure-based formulation linearizes each discrete nonlinear governing equation implicitly with respect to that equation’s dependent variable. The result is a scalar system of linear equations with one equation per cell in the domain. These equations can be solved with a segregated or with a coupled algorithm. If the segregated algorithm is employed, the velocities are solved sequentially, followed by a pressure correction step. Next, the mass flow, pressure and velocity are solved again. Lastly, the energy, species, and turbulence scalars are resolved. This process is repeated until a converged solution is
achieved. On the other hand, if the coupled algorithm is employed, a system of equations comprising the momentum and pressure-based continuity equations are solved simultaneously, before the energy, species, and turbulence scalars are solved for. The rate of solution convergence is faster for the coupled algorithm, but at a larger memory requirement.

The density-based formulation can discretize the governing equations either implicitly or explicitly. On one hand, the implicit option linearizes each equation with respect to all dependent variables in a set. This results in a block system of N linear equations per cell. This approach solves for all coupled variables (e.g. $p$, $u$, $v$, $w$, $T$) in all cells at the same time. On the other hand, the explicit option linearizes the equations in such a way that the dependent variables are updated based on their existing value. This formulation solves for all coupled variables one cell at a time.

FLUENT employs a point implicit linear equation solver (Gauss-Seidel or Incomplete Lower Upper Decomposition) in conjunction with an algebraic multigrid (AMG) method [40, 43] to solve the implicitly linearized system of equations [27]. Multigrid schemes accelerate solver convergence by computing corrections on a series of coarser grid levels, thus accelerating the removal of low frequency (global) errors. The immediate benefit is a considerable reduction in the number of iterations and the CPU time required to obtain a converged solution. The concept behind the AMG methodology is that long wavelength (low frequency) errors on a fine level appear as short wavelength (high frequency) errors on a coarser level and can be damped out by a relaxation scheme more efficiently. Without a multigrid scheme, the solver may stall with a prohibitively low reduction rate [26] because the removal of low frequency errors would be completed at a rate inversely proportional to the grid size. Coarse level equations are generated algebraically and no additional independent mesh is required.

Explicitly linearized systems of equations are solved with a multi-stage Runge-Kutta (RK) method as the updated variables are functions of existing field variables. FLUENT offers the option of employing a Full Approximation Storage (FAS) multigrid scheme [40] to accelerate the explicit solver [27]. FAS is usually applied to coupled, time-marching codes. The governing equations are discretized at each grid level and therefore, the creation of coarser meshes is required, either independently or through the use of agglomeration [44].
The computational studies presented in this thesis were completed using an implicit pressure-based segregated solver with Green-Gauss node-based gradient option, AMG, and the SIMPLE algorithm for pressure velocity coupling.

2.6 Viscous Flow Analysis

Turbulent flows are characterized by small scale and high frequency fluctuations in momentum, energy, and species concentration (e.g. water droplets). In principle, the Navier-Stokes equations describe both laminar and turbulent flows without any additional information. However, turbulent flows at realistic Reynolds numbers span a large range of length and time scales. Resolving all these scales in a direct numerical simulation (DNS) would require expensive and unavailable computing power and storage space. The alternative to DNS is to time–average the Navier-Stokes equations generating Reynolds-Averaged Navier Stokes (RANS) equations that include an additional term, called turbulent viscosity, which is added to the molecular viscosity. Turbulence models are statistical formulations designed to calculate the value of this additional term. The choice of a turbulence model for a CFD simulation is based on a criteria framework (Table 5), outlined in [45], that considers the particular flowfield under study, the formulation of the turbulence models including near-wall treatment (low-Re or wall functions), the model sensitivity to boundary conditions, the spatial discretization accuracy, and comparison with empirical data.

Table 5. Turbulence model selection criteria

<table>
<thead>
<tr>
<th>Criterion</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cases Examined</td>
<td>Flowfield geometry and relevant physics. Established level of practice for the scenario.</td>
</tr>
<tr>
<td>Formulations Examined</td>
<td>Form of the turbulence models used (Standard Calibration) Near-wall treatment.</td>
</tr>
<tr>
<td>Model Implementation</td>
<td>Boundary conditions sensitivity and robustness. Numerical Stability</td>
</tr>
<tr>
<td>Numerical Accuracy</td>
<td>Discretization errors. Convergence criteria and grid resolution</td>
</tr>
<tr>
<td>Turbulence Model Sensitivities</td>
<td>Percentage difference between numerical and experimental data.</td>
</tr>
</tbody>
</table>
2.6.1 Jet Flow Turbulence Modeling

The impinging jet flow (shown in Fig. 13) encompasses three different regions, each with particular physical characteristics: free jet, stagnation jet, and wall jet.

The free jet region includes the jet potential core, which is estimated to be 6 nozzle-to-plate \((z/d)\) distances in length [46, 47, 48, 49] and the shear layers that bound it. The stagnation region involves rapidly decelerating flow and a consequent change in static pressure. Significant pressure gradients develop in this region, not only in the direction of impingement, but also radially outwards. The wall jet region is formed due to a favorable radial pressure gradient that accelerates the flow as it spreads away from the impingement region. The wall jet flow reaches a maximum velocity and quickly decelerates as its static pressure increases to equal the diffuser bay pressure. In short, there are two pressure gradients in the wall jet region, a favorable one until the maximum velocity is attained, and an adverse one afterwards. Both gradients affect the heat transfer from the impinging jet to the surface.

![Diagram of single jet impingement on a flat surface](image_url)

**Figure 13.** Diagram of single jet impingement on a flat surface [37]
The jet flow characteristics previously described difficult turbulence modeling in several ways because there is no available formulation that has been shown to perform acceptably in every region of the jet flow. A number of models (e.g. $\kappa - \varepsilon$, Realizable $\kappa - \varepsilon$) are optimized for free shear flows such as submerged jets. Other models (e.g. S-A, $\kappa - \omega$) perform best in boundary layer flows such as the wall jet region.

In addition, most models have simple and less accurate predictions for turbulent effects in a stagnation region. The two-equation models are based around an assumption about the low importance of pressure gradients and the isotropy of the Reynolds stresses (i.e. all the normal stresses are equal and the tangential stresses are zero – Boussinesq Assumption). However, experiments have shown that these assumptions do not apply in stagnation regions or, in general, in cases of near-wall turbulence [50]. The assumption of local isotropy becomes inconsistent in a stagnation region where mean rate of strain and rise of static pressure are significant [50, 51, 52, 53, 54, 55, 56]. Therefore, no two-equation eddy-viscosity model should be expected to provide heat transfer rates in the stagnation region with errors better than 15 to 20% [50].

Another aspect of turbulence modeling affecting heat transfer predictions, besides the basic assumptions, is the near-wall treatment. There are two approaches to this problem. The first one uses a high resolution grid near the wall ($y^+ \sim 1$) to resolve the entire viscous sub layer and turbulent boundary layer with turbulence equations intended for use at low cell Reynolds number. The second approach involves using algebraic equations to relate steady and fluctuating velocity and scalar profiles to wall distance and surrounding flow properties. These wall functions predict the flow properties in and above the viscous sub layer and require a grid with a $y^+ \sim 30$. Note that these functions are developed for high Reynolds number parallel flows and are not based upon impinging jet flows, so they may not reproduce correct velocity profiles near walls [50]. Furthermore, these functions are based on the absence of pressure gradients near or along the wall, a strong characteristic of the flow fields in the stagnation and in the wall jet regions.
2.6.2 Turbulence Models

As previously mentioned, a direct numerical simulation (DNS) using three-dimensional, time-dependent Navier-Stokes equations at full scale Reynolds number is computationally too expensive as is a Large Eddy Simulation (LES). An alternative for engineering purposes is to time-average the Navier-Stokes equations presented in Section 2.3. The resultant system is known as Reynolds-Averaged Navier Stokes (RANS) equations.

The time-averaged products of the fluctuation of the three flow velocity components about their mean values are called Reynolds stresses or turbulent shear stresses. These terms represent the time-averaged rate of transfer of momentum. Turbulence models attempt to replicate the physical characteristics of the turbulent flow field to approximate the Reynolds Stresses and close the RANS system of equations. FLUENT provides a wide array of RANS applications involving wall-bounded flows. It solves a governing equation (10) for the kinematic eddy (turbulent) viscosity by transporting a variable called modified turbulent kinematic viscosity, $\tilde{\nu}$, turbulence models. The ones considered are five eddy-viscosity models. Second-moment closure (SMC) models such as the 7-equation Reynolds Stress Model were not considered because they are considerably expensive computationally for a three dimensional simulations and have not shown yet satisfactory improvements in the prediction of jet impingement flows [50, 51].

The Spalart-Allmaras (S-A) [57] is a one-equation, low-Re model designed specifically for aerospace application involving wall-bounded flows. It requires only one transport equation (Eqn. 10) for the modified viscosity $\tilde{\nu}$ based on which the eddy viscosity is evaluated. In equation (10), $G_\nu$ is the production of turbulent viscosity, $Y_\nu$ is the destruction of turbulent viscosity in the near-wall region, and $\sigma_\nu$ and $C_{b2}$ are constants.

$$
\frac{\partial (\rho \tilde{\nu})}{\partial t} + \frac{\partial (\rho \tilde{\nu} u_i)}{\partial x_i} = G_\nu + \frac{1}{\sigma_\nu} \left( \frac{\partial}{\partial x_j} \left( \mu + \rho \tilde{\nu} \right) \frac{\partial \tilde{\nu}}{\partial x_j} \right) + C_{b2} \rho \left( \frac{\partial \tilde{\nu}}{\partial x_j} \right)^2 - Y_\nu + S_\nu \tag{10}
$$

The S-A model requires a high near-wall grid resolution that can resolve the viscous affected region of the boundary layer properly (i.e. $y+$ within the viscous sub-layer). The model simulates the wall jet region with accuracy since it exhibits good results for boundary layers subjected to adverse
pressure gradients. However, the S-A model was not designed for the simulation of free jet flows or for cases where the flow changes abruptly from a wall bounded to a free shear flow or vice versa. Nevertheless, the model tends to perform well for a wide variety of flows and has been used with fair degree of success and accuracy in jet impingement studies, particularly in anti-icing systems [13, 26, 31, 34].

The standard $\kappa - \varepsilon$ model [58] is a two-equation eddy-viscosity model widely used in industry applications due to its robust formulation. Two-equation models require the solution of two transport equations for the turbulent velocity and flow length scales. In a $\kappa - \varepsilon$ model, the transport variables are the turbulent kinetic energy, $\kappa$, and the turbulent energy dissipation rate, $\varepsilon$. The standard $\kappa - \varepsilon$ model transport equations are:

\[
\frac{\partial \left( \rho \kappa \right)}{\partial t} + \frac{\partial \left( \rho \kappa u_i \right)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_{\kappa}} \right) \frac{\partial \kappa}{\partial x_j} \right] + \frac{\partial Y_M}{\partial x_i} \kappa + G \kappa + G_b - \rho \varepsilon - Y_M + S_\kappa \quad (11)
\]

\[
\frac{\partial \left( \rho \varepsilon \right)}{\partial t} + \frac{\partial \left( \rho \varepsilon u_i \right)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_{\varepsilon}} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{\kappa} \left( G \kappa + C_{3\varepsilon} G_b \right) - C_{2\varepsilon} \rho \varepsilon^2 \frac{\varepsilon^2}{\kappa} + S_\varepsilon \quad (12)
\]

In equations (11) and (12), $G_\kappa$ and $G_b$ are the generation of turbulent kinetic energy due to the mean velocity gradients and buoyancy, respectively; while $Y_M$ is the contribution of the fluctuating dilatation in compressible turbulence. $S_\kappa$ and $S_\varepsilon$ are user-defined source terms, and $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are model constants.

The $\kappa - \varepsilon$ model employs wall functions. FLUENT provides the option of using standard, non-equilibrium, and enhanced wall functions. The enhanced wall functions have terms to account specifically for pressure gradient effects and require a high grid resolution near a wall ($y^+ < 5$).

It is widely acknowledged that the standard $\kappa - \varepsilon$ model produces poor results for impinging jets, an eventuality known as “round-jet anomaly” [58]. One of its limitations is that it calculates the Reynolds stresses as a direct function of the velocity gradients. Since the model is based on flow behavior at high Reynolds numbers, it is not suitable in regions where viscous effects are comparable in magnitude to turbulence effects. The model is designed for free shear flows and is not intended to provide
good results for wall jet flows where pressure gradients are significant. The standard \( \kappa - \varepsilon \) model also
generates erroneous turbulent kinetic energy levels in stagnation regions [50, 51]. This is related to the
assumption of isotropic turbulence and the use of wall functions that approximate near wall velocity
fluctuations inadequately. The standard \( \kappa - \varepsilon \) model has been used in some jet impinging problems with
marginally acceptable results [23, 28].

The Realizable \( \kappa - \varepsilon \) is a modification of the standard \( \kappa - \varepsilon \) model. Its transport equation for
the dissipation rate is derived from an exact equation for the transport of the mean-square vorticity
fluctuation. Besides, it allows variation of certain proportionality constant to avoid predicting negative
normal Reynolds stresses or excessively high Reynolds shear stresses which are not consistent with the
physics of turbulent flows. The governing equations are:

\[
\frac{\partial (\rho \kappa)}{\partial t} + \frac{\partial (\rho \kappa u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_{\kappa}} \right) \frac{\partial \kappa}{\partial x_j} \right] + G_{\kappa} + G_{b} - \rho \varepsilon - Y_M + S_{\kappa} \quad (13)
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_{\varepsilon}} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S_{\varepsilon} + \frac{\rho C_2 \varepsilon^2}{\kappa + \sqrt{\rho \varepsilon}} + C_1 \frac{\varepsilon}{\kappa} C_{3\epsilon} G_{b} + S_{\varepsilon} \quad (14)
\]

Since the transport equations in this model were originally derived under the assumption of a
large Reynolds number, they require empirical wall functions and low Reynolds number source terms for
the near-wall treatment. FLUENT provides an option to activate enhanced wall treatment. In that case,
the viscosity-affected near wall region is resolved to the viscous sub-layer and the mesh resolution
requires a \( y^+ \) less than 5.

The Realizable \( \kappa - \varepsilon \) model has been extensively validated [33, 38] since its conception and
has been found to be accurate in predicting the spreading rate of both planar and round jets, which shows
a considerable improvement over the standard \( \kappa - \varepsilon \) formulation. It also provides superior performance
for flows involving rotation, boundary layers under strong pressure gradients, separation, and
recirculation.

Another type of two-equation eddy viscosity model is the \( \kappa - \omega \) formulation [60]. This model has
a transport equation for the turbulent kinetic energy, in similar fashion to the \( \kappa - \varepsilon \) approach, and
another one for the dissipation per unit turbulent kinetic energy, \( \omega \). Its transport equations are:
In equation (16), $G_\omega$ is the generation of dissipation due to mean velocity gradients and $Y_\omega$ accounts for the turbulence dissipation. Sources terms have been omitted.

The standard $\kappa - \omega$ turbulence model incorporates modification for low-Reynolds number effects, compressibility, and shear flow spreading that allow it to generate good predictions of flow properties in wall jets, both in the sub layer and logarithmic region of the boundary layer. It outperforms the standard $\kappa - \varepsilon$ model in jet impingement problems, but it is not equally accurate in free jet flows [49, 61]. This model has been used in impinging jets and anti-icing system simulations [13, 34, 62] with satisfactory results.

The Shear Stress Transport (SST) $\kappa - \omega$ model was developed by Menter [63, 64] to effectively blend the robust and accurate formulation of the standard $\kappa - \omega$ model proposed by Wilcox [60] in the near-wall region with the free-stream independence of the $\kappa - \varepsilon$ model in the far field. This hybrid model has, in essence, the same transport equations as the standard $\kappa - \omega$ formulation, but includes a cross-diffusion term ($D_\omega$) in the $\omega$-equation (17) that monitors the transport of shear stresses. The formulation also includes a limiter for the turbulent kinetic energy production term.

$$
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho \omega u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + D_\omega + S_\omega \tag{17}
$$

In this equation, $G_\omega$ is the generation of dissipation due to mean velocity gradients and $Y_\omega$ accounts for the dissipation of $\omega$.

The SST $\kappa - \omega$ model includes a modified definition of the turbulence viscosity to account for the transport of turbulent shear stress. FLUENT provides an option to operate the SST $\kappa - \omega$ model as a low Reynolds number turbulence model. Under this condition, the calculations receive enhanced wall treatment and the required mesh must be fine with a $y^+$ within the viscous sub-layer. This model has
been successfully validated for heat transfer applications [65], heated high-speed subsonic and supersonic air jets [66, 67, 68, 69], separated flow regions over airfoils [70], transonic flow over an airfoil [71], and transition modeling for general application in aeronautics [72].

A turbulence model study is presented in Section 3.3 to select a suitable formulation for the anti-icing simulations based on the criteria previously outlined. All the models described above were included in the study and the focus was on identifying the model with the best results for impinging jets heat transfer computations.

2.6.3 Transition

The nature (laminar or turbulent) of a boundary layer affects the heat transfer from the external flow to the wing skin. In an icing tunnel, the cold airflow over the wing produces a large thermal load that cools the wing leading edge at a substantial rate. The amount of cooling increases as the boundary layer transitions from laminar to turbulent. In the IRT facility, this issue becomes more important due to the high freestream turbulence intensity (FSTI) of approximately 1% in dry wind tunnel flow [73, 74]. Transition modeling in this case is important due to the influence of the boundary layer on the convective heat transfer properties, a phenomenon that is further enhanced by the presence of pressure gradients [75, 76] over the wing surfaces.

Transition mechanisms are often categorized depending on the turbulence level of the flow, the pressure gradient along the boundary layer, the geometry of the body in the flow field, the surface roughness, and the freestream Mach number. The most common transition mechanism is Natural transition. It occurs due to the linear growth of flow disturbances in the boundary layer called Tollmien-Schlichting waves [77]. Another important mechanism is Bypass transition, which occurs when there are high levels of turbulence in the freestream [72]. Examples of this transition mechanism are a body that encounters the wake from an upstream device or a model in an icing tunnel encountering the wakes and flow angularities produced by the spray nozzle system located upstream of the test section [73, 74].

As Menter [72] summarizes, there are three conventional methods typically used in industrial applications to predict boundary layer transition: low-Re turbulence models, \( e^N \) method, and empirical correlations. In low-Re turbulence models, the calibration of the wall functions is based on reproducing
the viscous sub layer behavior, not on predicting transition. Therefore, the simulation is for practical effects fully turbulent and it can only be applied to bypass transition cases.

The $e^N$ method is based on local, linear stability and the parallel flow assumption [78]. Transition is assumed to start when the linear disturbance amplitude ratio exceeds a user-defined parameter $N$, which is typically set to 7 or 9 for low turbulence freestreams. Even though this method has been shown to be very accurate for predicting transitions on airfoils under low turbulence conditions [79], it cannot predict transition due to non-linear effects, such as high freestream turbulence or surface roughness. In addition, it has not been successfully applied to predict transition on three-dimensional models [72].

Experimental correlations usually relate freestream turbulence intensity and local pressure gradients to the transitional momentum thickness Reynolds number, $Re_{\theta t}$ [80]. It is reported to be an accurate method as long as it is applied in cases that correctly resemble the experimental setups used to determine the correlations. However, it is difficult to integrate numerically into CFD simulations [72].

Transition modeling in CFD is a complex task. FLUENT does not model transition. A simulation can be either fully turbulent (low-Re turbulence models approach) or fully laminar. However, FLUENT allows defining certain zones of the flow domain as laminar where the turbulent scalars are not solved. A sub-domain can be created to simulate the laminar portion of a boundary layer around a leading edge, while the rest of the external flow computational domain remains fully turbulent. The only necessary inputs are the approximate transition locations on the upper and lower surfaces of the airfoil.

The process followed to determine the transition locations is reported in Chapter 3 and was conducted with the CFD software CFX [81]. This three-dimensional finite-volume based Navier-Stokes code includes a proprietary local correlation-based transition model developed by Menter and Langtry [72, 82] that relies on two transport equations for $Re_{\theta t}$ and Intermittency. The first equation relates empirical correlations to the onset criteria, whereas the second equation triggers transition by interacting with the SST $\kappa - \omega$ turbulence model equations. The details of the numerical formulation can be found in [72, 82, 83] and in Appendix B. Validation of this model for natural and bypass transition have been conducted for a variety of aeronautics and turbomachinery applications including flow over an aircraft airfoil, a wind turbine airfoil, a slat-flap configuration, a swept wing in transonic flow, and a helicopter airframe [72, 83]. The analysis with CFX was completed exclusively for the wind tunnel flow around the
wing model, not for the anti-icing system internal flow, although this is envisioned as a future goal. The transition locations obtained from CFX were used to create laminar sub-regions for the simulation of the PSM in FLUENT. The study is detailed in Section 3.4.
CHAPTER 3
RESULTS AND DISCUSSION

3.1 Experimental Database

The anti-icing system studies presented use data from the experimental database compiled during wind tunnel testing at the NASA Glenn Icing Research tunnel. The test model and instrumentation for the icing tests are briefly described in this section. A list of the test cases selected for the numerical parametric study is also provided. More detailed documentation can be found in [19, 20].

3.1.1 Wing Model Geometry

The two-dimensional wing model used in the icing tunnel tests had a constant airfoil section, shown in Fig. 14, representative of business jet wings with no twist, no sweep, and no taper. The model span was 72-inch and the chord was 60-inch. The leading edge, which was fabricated out of Al 6061-T6 aluminum and had a thickness of 0.1-inch, was equipped with an inner-liner type bleed air anti-icing system consisting of a piccolo tube and an inner-liner skin designed to direct the hot air flow close to the wing interior skin surface.

Figure 15 (a reproduction of Fig. 8) shows the leading edge cross-section and a diagram of the two diamond piccolo-hole configurations fabricated for the test: P45-00-M45 and P70-P10-M40. The piccolo tube had an outer diameter of 1.25-inch and a diamond hole-pattern. The diameter of each piccolo jet-hole was 0.052 inches and the spacing (pitch) was 2.44-inches. The single hole in the diamond pattern was placed halfway between stations containing two holes. The heated region extended to 10% chord (6.041 inches) on both upper and lower surfaces.

Two piccolo tubes were fabricated for the tests P45-00-M45 and P70-P10-M40. The only difference between these two piccolo tubes was the circumferential placement of the piccolo jet holes. For the P45-00-M45 tube the two holes were placed at +45° and -45°, and the single holes at 0° with respect to an axis through the center of the piccolo and parallel to the airfoil chord line. For the P70-P10-M40 tube, the two holes were placed at +70° and -40° and the single holes at +10°.
3.1.2 Model Instrumentation

The model thermal leading edge extended from the tunnel floor to the ceiling. Leading edge skin temperatures and bleed air system thermal and pressure data were obtained with T-type thermocouples, resistance temperature detectors (RTDs), heat flux gages (HFGs), and pressure transducers.

Leading edge skin temperatures were measured at four spanwise stations as shown in Fig. 16. The station selected for computational modeling was Station B. It was located 30.175-inch above the tunnel floor and was instrumented with 32 T-type thermocouples. The thermocouples were embedded inside the leading edge skin approximately halfway between the inner and outer skin surfaces. In addition, 12 Heat Flux Gages were located on a spanwise station 27.725 inches above the tunnel floor, close to station B. Figures 17 and 18 show the location of the leading edge thermocouples and heat flux gages.
Figure 16. Spanwise locations of leading edge skin thermal instrumentation.

Figure 17. Distribution of T-type thermocouples embedded inside the leading edge skin.

Figure 18. Chordwise distribution of heat flux sensors attached to the skin interior surface.
Figures 19 and 20 show the instrumentation installed in the diffuser and diffuser passages formed by the inner liner and the leading edge skin. Pressure taps installed on the inner liner skin were used to monitor pressures and T-type thermocouple were used to measure hot air temperature. Another set of thermocouples and four heat flux gages were installed on the back of the inner-liner skin to quantify heat loss through the skin. Inner-liner and diffuser pressure taps and thermocouples were installed at spanwise locations corresponding to Stations A and B. Heat fluxes were placed at Station B only.

In addition to the instrumentation described above, thermocouples and total pressure ports were installed inside the piccolo at four spanwise locations: 6.92-inch, 30.18-inch, 42.43-inch, and 65.59-inch above the tunnel floor to provide piccolo centerline values. A flow meter, two thermocouples and a pitot-static probe were used to monitor bleed air mass flow properties upstream of the piccolo inlet. Figure 21 shows the instrumentation installed upstream of the piccolo inlet underneath the tunnel floor.
3.1.3 Experimental Test Cases

This section presents selected cases from the WSU experimental database [84] used for validation of the computational methodology. All cases correspond to icing tests performed at the NASA Glenn Icing Research Tunnel (IRT) by WSU. Table 6 lists run numbers and test conditions. For all cases, the wing AOA was 3 degrees and the tunnel freestream Reynolds and mach numbers were approx. 6.8 million and 0.18. The piccolo inlet was located at the same level of the tunnel floor, as shown in Fig. 16, whereas the bleed air supply was underneath the tunnel floor. The piccolo inlet mass flows listed were measured in the experiment upstream of the piccolo inlet (See Fig. 21). In a similar fashion, the total pressure listed is an average of the readings obtained at the centerline of the piccolo tube along the span (the total pressure variation along the span was negligible in all cases). The total temperature values correspond to the first reading along the piccolo centerline inside the wing model. This approach was used because preliminary simulations with the FSM indicated that there were not significant changes in total temperature along the first 10 inches of the piccolo tube measured from the floor.
Notice that there were three distinct levels of mass flow and two levels of total temperature. Runs 26, 58, 62, and 107 had mass flows on the range of 4.26 to 4.68 lbm/min. In similar fashion, Runs 66 and 76 had low temperatures on the range of 257.5 to 263.5°F, whereas the other runs had high levels at 362.9 to 383.7°F. These small ranges, which were due to experimental uncertainties, did not pose an impediment for comparing the results under the assumptions of nearly similar mass flow rates and piccolo inlet total temperature. Anti-icing studies are presented beginning in Section 3.5.

Table 6. Experimental test cases.

<table>
<thead>
<tr>
<th>Run</th>
<th>Freestream Speed (kts)</th>
<th>Freestream Temperature (°F)</th>
<th>Piccolo Inlet Mass Flow (lbm/min)</th>
<th>Piccolo Inlet Total Pressure (psia)</th>
<th>Piccolo Inlet Total Temperature (°F)</th>
<th>Piccolo Configuration</th>
</tr>
</thead>
<tbody>
<tr>
<td>026</td>
<td>115</td>
<td>20</td>
<td>4.27</td>
<td>28.3</td>
<td>373.8</td>
<td>P45-0-M45</td>
</tr>
<tr>
<td>042</td>
<td>115</td>
<td>20</td>
<td>5.31</td>
<td>34.3</td>
<td>383.7</td>
<td>P45-0-M45</td>
</tr>
<tr>
<td>058</td>
<td>110</td>
<td>-22</td>
<td>4.68</td>
<td>28.2</td>
<td>367.7</td>
<td>P45-0-M45</td>
</tr>
<tr>
<td>062</td>
<td>115</td>
<td>20</td>
<td>4.12</td>
<td>26.2</td>
<td>263.5</td>
<td>P45-0-M45</td>
</tr>
<tr>
<td>066</td>
<td>115</td>
<td>20</td>
<td>1.83</td>
<td>17.6</td>
<td>362.9</td>
<td>P45-0-M45</td>
</tr>
<tr>
<td>076</td>
<td>115</td>
<td>20</td>
<td>10.7</td>
<td>63.3</td>
<td>257.5</td>
<td>P45-0-M45</td>
</tr>
<tr>
<td>107</td>
<td>115</td>
<td>20</td>
<td>4.26</td>
<td>31.1</td>
<td>377.9</td>
<td>P70-P10-M40</td>
</tr>
</tbody>
</table>

3.2 Turbulence Model Study

A turbulence model study was conducted in order to find the most suitable model to appropriately estimate the physical characteristics of the external and internal flow fields in an anti-icing simulation. As described in Section 2.6 (Viscous Flow Analysis), the choice of a turbulence model for a CFD simulation was based on a criteria framework that considered other aspects besides the physical representation of the flow. Special consideration was given to the sensitivity of the models to turbulence intensity. In addition, a preliminary assessment was conducted considering the availability (standard form in different solvers), robustness, and versatility (appropriate near-wall treatment) of different turbulence models, albeit to a lesser extent.

The turbulence models presented in Chapter 2 were evaluated in an axisymmetric jet impingement scenario. The results were compared with experimental data collected by Goldstein et al. [85] for a circular jet impinging on a flat surface located at four diameters from the nozzle (z/d = 4). In Goldstein’s experiment, the test surface was part of a solid textolite plate (1.6-mm thick) that was
insulated at the back with 5 cm of Styrofoam. Goldstein employed thermocouples buried in the plate to measure the radial distribution of recovery temperature from which recovery factors, \( r \), were computed [85]. Recovery factor is defined in equation (18) where \( T \) is the measured temperature by the thermocouples, \( T^j_s \) is the nozzle static temperature, and \( T^j_d \), the dynamic temperature.

\[
r = \frac{T - T^j_s}{T^j_d}
\]  

(18)

The multi-block quadrilateral grid used, shown in Figs. 22 and 23, had a nozzle-to-plate distance of four. The computational domain had a size of 105\( d \) x 100\( d \) in the jet longitudinal and transverse directions, respectively, and included a solid region behind the impingement wall. The resultant mesh consisted of 311,000 quadrilateral cells. In Fig. 23, the grid is shown with overlaid colors in the background to better illustrate the grid resolutions used in the different regions of a jet impinging. The grid resolution in the jet region had 350 nodes in the direction perpendicular to the nozzle and 300 nodes along the nozzle radius and the wall next to it. The mesh had an average \( y^+ \) of 0.04 at the impingement surface to ensure a fine near-wall grid resolution.

Figures 22 and 23 also show the boundary conditions used. The axisymmetric simulation was conducted for a jet flow with a total pressure of 1.8 psig and total temperature of 300K at the nozzle using a Pressure Inlet boundary condition. The nozzle-exit-diameter-based Reynolds number (\( Re_j \)) was 124,000 and the average Mach number was 0.47. The nozzle turbulence boundary conditions were specified with turbulence intensity and hydraulic diameter. Three different turbulence intensities were evaluated: 1%, 3%, and 5%. Note that no information was provided in [85] regarding the turbulence intensity of the flow. Flow outlets with zero gage static pressure were used to simulate atmospheric conditions. An inlet boundary condition with 0.0001 psig total pressure and a total temperature of 300K was also specified at the opposing edge of the test plate to define ambient flow. This condition is not shown in Fig. 23. The nozzle wall was adiabatic, whereas the impingement wall was a thermal interface between a solid and a fluid domain. The simulations were conducted with the default pressure-based (segregated) first-order scheme and the following turbulence models: S-A, Standard \( k-\varepsilon \), Realizable \( k-\varepsilon \), Standard \( k-\omega \), and Shear Stress Transport \( k-\omega \). Figures 24 through 28 show the resultant recovery factors.
Figure 22. Axisymmetric grid – impinging jet.

Figure 23. Axisymmetric grid – jet region.
Figure 24. Spalart-Allmaras results.

Figure 25. Standard $\kappa - \epsilon$ results.

Figure 26. Realizable $\kappa - \epsilon$ results.

Figure 27. Standard $\kappa - \omega$ results.
In general, the results identify the flow structure and shape of the recovery factor distribution. At stagnation, the heat transfer is the highest and a peak in the recovery factor forms. Vortices around the impingement bubble near the region where the shear layers meet the impingement wall capture energy from the flow producing a decrease in recovery factor, seen at $r/OD=1$.

The results for the Spalart-Allmaras model (Fig. 24) presented considerable fluctuation in the recovery factor distribution for variable turbulence intensities. For the 3% and 5% values, it predicted recovery factors greater than 1.0, which implied that the temperature recorded at the wall was higher than the total temperature of the nozzle flow. According a literature survey, air entrainment is not significant for the nozzle-to-plate spacing of this problem ($z_n/d = 4$) [86, 87, 88]. This premise was not verified. The results indicated excessive sensitivity in the S-A model to turbulence intensity levels. A possible explanation for this phenomenon was that the S-A model was intended and, therefore, calibrated for external low-turbulence (Max ~1%) wall-bounded flows typical of freestream air around an airfoil or a wing. Indeed, the curve for 1% turbulence intensity predicted acceptable recovery factors that progressively degraded as the curve spread away from the impingement point.

The results for the Standard $\kappa-\epsilon$ model (Fig. 25) were insensitive to the nozzle flow turbulence intensity. However, this model consistently predicted excessively high recovery factors. This phenomenon, known as the "jet anomaly", was attributed to the excessive generation of turbulence kinetic
energy, $\kappa$, near the impingement point [45, 49, 50, 59, 89, 90, 91]. The Realizable $\kappa-\varepsilon$ results (Fig. 26) presented significant sensitivity to the jet flow turbulence intensity, but not as dramatic as the Spalart-Allmaras model did. The results were reasonable ($r < 1.0$) for turbulence intensities of 1% and 3%, but excessive for 5%, indicating that this model was not suitable for highly-turbulent jet flows. Nevertheless, there was improvement over the standard $\kappa-\varepsilon$ model both in accuracy and spreading rates. The curve that best fit the experimental set of data was the 1% curve, even though it under-predicted the minima of the recovery factor distribution. The results of the standard $\kappa-\omega$ model (Fig. 27) compared favorably to the Realizable $\kappa-\varepsilon$ in predicting the minimum recovery factor. However, its predictions of the peak recovery factors were over estimated in all cases indicating excessive turbulence kinetic energy, $\kappa$. On the other hand, the spreading rates appeared to be uniform for the three turbulence intensities tested, but compared adversely to the results from the Realizable $\kappa-\varepsilon$ model in predicting the wall jet region starting at two diameters away from the centerline.

The Shear Stress Transport $\kappa-\omega$ results (Fig. 28) provided the best results at the center of the impingement region for all three turbulence intensity levels (1%, 3%, and 5%). The impingement point recovery factor was predicted with a very acceptable degree of accuracy and the spreading rates were also satisfactory, indicating that the levels of turbulent kinetic energy were well controlled and appropriately limited. The minimum recovery factor appeared to be under-predicted, but it remained within a determined range ($r = 0.88 - 0.90$).

As pointed out in Section 2.6, in two-equation eddy-viscosity models without production limiters, the Boussinesq assumption leads to high turbulent kinetic energy levels in stagnation regions [45, 50, 51, 89, 90, 91]. Figure 29 shows a comparison of jet centerline turbulence kinetic energy for the four two-equation formulations compared. As can be seen, the SST $\kappa-\omega$ model limited the kinetic energy levels at the stagnation region, thus improving the accuracy of the heat transfer calculations. An evaluation demonstrated that the best results were obtained for turbulence intensities of 1% and 3%. Figures 30 and 31 compare recovery factors of all five models for this two turbulence intensities.

The SST $\kappa-\omega$ model satisfied all the selection criteria outlined in Section 2.6 and was selected for the remaining studies. It was found to be a suitable formulation for the impinging jet problem, predicting acceptable recovery factors at different levels of turbulence intensity, and controlling the levels
of turbulence kinetic energy. It was also adequate for external flow simulations based on the findings from previous research efforts [64, 70, 72].

3.3 Grid Resolution and Spatial Discretization Study

Figure 29. Jet centerline turbulence kinetic energy ($k$).

Figure 30. Recovery factor distribution for turbulence intensity of 1%.

Figure 31. Recovery factor distribution for turbulence intensity of 3%.
In a computational simulation, error due to numerical diffusion can be alleviated either by increasing grid resolution or by raising the order of accuracy used in the numerical discretization scheme. A finer grid requires more computer memory and increases the computational time. A high-order spatial discretization scheme can reduce the diffusion error by it also introduces instability to the solver.

The order of discretization in a numerical solution typically indicates the level of reduction of the truncation errors that arise when partial differential equations are discretized. First order schemes usually provide acceptable and cost-effective solutions as long as high resolution grids are employed. Use of high order schemes is recommended when grid resolution is coarse. Ideally, the best scenario is to employ grids with fine resolution and high order schemes. However, this practice is frequently not cost-effective and seriously affects the versatility and efficiency of CFD for providing numerical solutions in a fast-paced industrial environment. Hence, it is necessary to find an adequate balance between the grid resolution and the spatial discretization scheme used to obtain solutions efficiently and with an acceptable accuracy for engineering applications.

Discretization in FLUENT employs an upwind scheme to approximate cell face value, required for convection terms, from cell-centered values. First order and second order discretization schemes are available. In a first-order upwind scheme, the face value \( \phi_f \) is set equal to the cell-centered value of \( \phi \) in the upstream cell. In the second-order scheme, \( \phi_f \) is obtained through a Taylor series expansion of the cell value about the cell centroid such that \( \phi_f = \phi + \nabla \phi \cdot \Delta s \) where \( \nabla \phi \) is the gradient of \( \phi \) in the upstream cell, \( \Delta s \) is the displacement vector from the upstream cell centroid to the face centroid. As reported in the literature review, previous research efforts [13, 37] at Wichita State University have explored different levels of grid resolution and spatial discretization for a simplified piccolo-hole anti-icing system. Both studies concluded that first order discretization was acceptable for a sufficiently fine grid resolution.

Grid resolution and spatial discretization studies were conducted using a jet impingement scenario. The experimental data from Goldstein et al. [85], previously used in the turbulence model study, was employed again. Three-dimensional grids were generated and used instead of the axisymmetric grid.
generated for the turbulence model study since it was important that the conclusions from the grid resolution study were applicable to 3D simulations of anti-icing systems.

Three grids were created for this study: coarse (Fig. 32a and 33a), medium (Figs. 32b and 33b), and fine (Figs. 32c and 33c). In these meshes, the jet-to-plate distance was set to four nozzle diameters, and a solid region was defined for the test plate. The computational domain had a size of $200d \times 200d \times 105d$, where $d$ was the nozzle diameter. The grid resolution for each mesh was based on the number of nodes distributed around the circumference of the nozzle exit. Hexahedral cells were used to keep the grid resolution constant within the 3D jet region. Table 7 summarizes the details of the three grid resolution levels considered including the following metrics: number of nodes around the nozzle circumference, uniform node spacing, and spacing-to-circumference ratio (i.e. $1/\text{nodes}$). The number of 3D cells and the cell density in the jet region are provided in Table 7.

Table 7. Grid resolution study information.

<table>
<thead>
<tr>
<th>Grid Resolution</th>
<th>Nodes around Nozzle</th>
<th>Node Spacing (inch)</th>
<th>Spacing / Circumference Ratio</th>
<th>Cells in Jet Region</th>
<th>Grid Density (cells/in$^3$) in Jet Region</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>12</td>
<td>0.2620</td>
<td>0.0834</td>
<td>1,121</td>
<td>2,855</td>
</tr>
<tr>
<td>Medium</td>
<td>48</td>
<td>0.0650</td>
<td>0.0207</td>
<td>20,661</td>
<td>52,626</td>
</tr>
<tr>
<td>Fine</td>
<td>192</td>
<td>0.0164</td>
<td>0.0052</td>
<td>885,885</td>
<td>2,256,457</td>
</tr>
</tbody>
</table>

![Figure 32. Comparison of nozzle mesh for grid resolution study.](image)
Figure 33. Comparison of grid resolution in 3D jet region.
Note that in all cases, the near wall grid was sufficiently fine (average $y^+ = 0.08$) in order to satisfy turbulence modeling requirements and to increase the fidelity of the heat transfer calculations. All computations were conducted with the Shear Stress Transport $\kappa - \omega$ turbulence model. The same jet boundary conditions used in the turbulence model study were employed: a total pressure of 1.8 psig and total temperature of 300K at the nozzle. The default pressure-based (segregated) solver was applied to advance the solution in time.

Figures 34 and 35 show the distribution of recovery factors between the first and second order solutions for all three grids. There was similarity in the trends and values between the first order and second order solutions for the medium and fine grid resolutions. The solution obtained with the coarse grid presented considerable fluctuation from the other two solutions. The centerline ($R/D = 0$) recovery factor was well predicted with both medium and fine resolution grids. The error was on average between 0.2% and 0.3% with respect to the experimental data available for both grid resolutions. For the coarse resolution, the error was 2.1% for the first-order solution and 0.5% for the second order solution. The results show that the use of high resolution near the impingement wall increased the fidelity of the heat transfer calculations in all three cases. Significant improvement due to higher-order discretization was achieved in the coarse grid, whereas the improvement in the medium and fine grids was marginal.
Figures 36, 37, and 38 show a direct comparison of discretization schemes for the grid resolutions considered. As can be seen, the medium and fine resolutions presented the least level of difference between first and second order solutions for the recovery factor. The results show that the advantage of using a higher order discretization scheme was more pronounced for to the coarse grid case.

In summary, the results indicated that both medium and fine resolution grids were appropriately dense to minimize errors with either first or second order discretization schemes. Consequently, the grid in the jet regions of the PSM used for the anti-icing studies had a resolution comparable to the medium grid: 46 nodes distributed uniformly around each piccolo hole circumference and a spacing/circumference ratio of 0.022 for a node spacing of approx. 0.003-inch. Also note that, as reported in Section 2.2, the piccolo jet hole regions in the PSM were meshed with two-dimensional triangular cells and three-dimensional tetrahedrons.

The first-order upwind scheme combined with medium and fine resolution grids, yielded results similar to the second-order scheme. In addition, the first order scheme exhibited better numerical convergence behavior that the higher order scheme.
3.4 Transition Modeling

A two-dimensional analysis was conducted using a panel method code, XFOIL [91], and the $e^N$ method to estimate a referential transition location on the wing upper surface under low freestream turbulence. The $e^N$ method is a semi-empirical formulation based on the local linear stability theory and the parallel flow assumption [77]. The results from XFOIL with the default $N=9$ (FSTI $\sim 0.05\%$) for the HAARP airfoil predicted full transition on the upper surface boundary layer at $x/c=7.4\%$. The $e^N$ method predicts appropriate transition locations, but its accuracy decreases under non-linear effects such as high freestream turbulence [78]. The freestream turbulence intensity of the Icing Research Tunnel (IRT) was estimated at 1% based on data from [72] and [73]. Therefore, it was expected that the transition on the upper surface of the wing in the IRT would be further upstream than the location indicated by XFOIL.

A second transition study with the HAARP wing model was conducted using CFX, a CFD code with a transition model (see code details in Section 2.6.3 and Appendix B), to determine its upper surface transition location. Figure 39 shows the wing model. The mesh on the tunnel floor (perpendicular to the wingspan direction) was the same used in the PSM for the symmetry planes. The total height and width of the tunnel test section (6-ft x 9-ft) were modeled and the wing was positioned at an angle of attack of 3 degrees. A pressure inlet boundary condition was specified for the tunnel inflow condition ($P_{\text{Tot}}=0.375$...
psig, $T_{Ro}=22.98^\circ$F, and FSTI=1%), whereas a pressure outlet was specified for the tunnel outflow condition with static pressure of zero psig. The freestream Reynolds number was 6.8 million and the mach number 0.18. The SST $\kappa-\omega$ turbulence model was employed along with a correlation-based transition model available in CFX (see Section 2.6.3 and Appendix B). The wing surface was modeled as an adiabatic wall.

![Figure 39. HAARP wing model in wind tunnel test Section, AOA = 3 deg.](image)

The semi-empirical correlation-based transition model in CFX was designed to calculate a transition onset momentum thickness Reynolds number ($Re_{\theta,t}$) in relation to pressure gradient and turbulence intensity in the flow. By definition, large increments in $Re_{\theta,t}$ indicate favorable pressure gradients and/or turbulence intensity decay. Conversely, large decreases in $Re_{\theta,t}$ point to adverse pressure gradient and/or increasing turbulence intensity [80, 82]. Increases in turbulence enhance the convective heat transfer of a boundary layer [92, 93] and subsequently increase the amount of cooling on the wing surface due to the external flow. A sudden drop in $Re_{\theta,t}$ indicates boundary layer transition from laminar to turbulent.
Figure 40 shows a two-dimensional view of the contours of \(Re_{\theta t}\) around the wing at the half-span section \((z=36\text{ inches})\). Notice that \(Re_{\theta t}\) increased due to the favorable pressure gradient over the upper and lower surfaces (up to 2.97% on US and 63% on LS). Over the upper surface, a sudden decrease in \(Re_{\theta t}\) occurred near the leading edge at approximately \(x/c=2.97\%\) indicating an increment turbulence intensity in the flow and the presence of an adverse pressure gradient, which denoted the occurrence of boundary layer transition. On the other hand, the transition at \(x/c=63\%\) was gentler on the lower surface and occurred primarily due to an adverse pressure gradient.

![Transition Onset Momentum Thickness Reynolds Number (\(Re_{\theta t}\))](image)

Figure 40. Contours of transition onset momentum thickness Reynolds number.

Further analysis demonstrates that the selected transition location \((x/c=2.97\%)\) was situated at an inflection point for the profiles of pressure gradient distribution and \(Re_{\theta t}\), respectively. Figures 41 and 42 show velocity profiles and \(Re_{\theta t}\) as a function of the ratio between the distance normal to the wing surface \((y_{local})\) and the boundary layer thickness \((\delta_{BL})\) at seven chordwise stations ranged between 0.74% and 10% \(x/c\). Three locations were upstream of the transition point \((x/c=0.74\%, 1.48\%, \text{ and } 2.23\%)\) and another three were downstream of transition \((x/c=4.45\%, 5.93\%, \text{ and } 10\%)\). Note that the boundary layer thickness is estimated to be the orthogonal distance from the wall surface to where the normalized velocity magnitude was nearly unchanged approximated the local boundary layer thickness at seven chordwise stations.
Figure 41. Boundary layer velocity profiles.

Figure 42. Boundary layer transition onset momentum-thickness Reynolds number profiles.
Figure 41 shows that for stations upstream of the transition location (broken-line curves), the local velocity magnitude increased due to the presence of a strong favorable pressure gradient. The acceleration stopped around $x/c=2.97\%$ (transition location) as the flow encountered an adverse pressure gradient. For the stations downstream of transition (solid curves), pressure recovery caused the local flow velocity magnitude to decrease. Figure 42 shows that $Re_{\theta t}$ increased near the surface as the flow moved downstream. At $x/c=2.97\%$, $Re_{\theta t}$ experienced a sudden decrease indicating transition in the boundary layer. At $x/c=4.45\%$, turbulence had propagated further into the boundary layer, but the inner sub-viscous region had not transitioned yet. At $x/c=5.93\%$, constant $Re_{\theta t}$ stopped increasing near the surface implying that the inner region of the boundary layer was on the process of transition. Finally, at $x/c=10\%$, the boundary layer was fully turbulent as indicated by the lower $Re_{\theta t}$ near the surface.

The transition locations obtained from CFX was used in FLUENT to establish a laminar boundary layer zones extending from the wing LE to the transition locations on the upper and lower surfaces. Figure 43 shows the grid topology of the PSM with the laminar boundary layer regions in turquoise around the leading edge and portions of the lower surface. Appendix C provides data on the evaluation of the effect of transition on skin temperature.

![Laminar Flow Regions](image)

Figure 43. Laminar flow regions surrounding PSM at AOA = 3 deg.
3.5 Boundary Conditions Study

The 72-inch full-span model (FSM) and the 4.88-inch partial-span model (PSM) were used to perform simulations for the warm hold case corresponding to experimental Run 26. Computational results from the FSM are presented to demonstrate the flow development inside the piccolo. The results consist of piccolo centerline total temperature and pressure distributions, and piccolo exterior surface temperature. For the CFD analysis performed with the PSM, the boundary conditions at the exit of the piccolo jet holes, and at the piccolo surface were specified based on the results obtained with the FSM.

The tunnel operational pressure in the FSM and PSM simulations was set to 14 psi. Boundary conditions for the FSM computations are as follows:

- Piccolo inlet: Pressure inlet at the opened end of the piccolo tube with a total pressure of 14.09 psig and total temperature of 373.8°F. This provided a mass flow rate similar to the experimental value.
- Hot air outlets: Pressure outlet at the upper and lower exits of the diffuser passage formed between the wing skin and the inner-liner skin with a gauge static pressure of 0.47 psig. This value was obtained from the experimental measurements.
- Inner-liner surface: No-slip wall condition with a chordwise heat flux profile based on measurements from the experiment.
- Tunnel inlet: Pressure inlet at the upstream end of the truncated tunnel (see Fig. 12) with a fixed total temperature of 22.98°F and a total pressure profile obtained from a preliminary CFD analysis with the wing in the full-size IRT test section (6-ft x 9-ft).
- Tunnel outlet: Static pressure profile at the upper and lower tunnel outlet surfaces (note the truncated tunnel shown in Fig. 12 is divided by the wing into two parts). Profile data was obtained from a preliminary CFD analysis with the wing in the full-size IRT test section. The tunnel airspeed computed from the pressures specified at the tunnel inlet and outlet boundaries was approximately 115 knots.
- Tunnel floor, ceiling, and side walls: No-slip, adiabatic wall conditions.

The regions listed below were defined for the simulation performed with the FSM.

- Piccolo tube: Modeled as a solid region.
• Wing skin: Modeled as a solid region.

• Hot air: Included the flow region between the piccolo inlet and the hot air outlets including the piccolo jets and the diffuser.

• External flow: Modeled fully laminar due to simplified geometry (only leading edge).

The FSM simulation was conducted with the default pressure-based segregated solver, first-order discretization scheme, and the SST $\kappa - \omega$ turbulence model.

Selected FSM results from the warm hold simulation are shown in Figs. 44 – 53. Figures 44 and 45 show the spanwise total temperature contours of the piccolo outer surface and the piccolo interior flow, respectively. Both the piccolo skin and interior flow temperatures decreased with spanwise distance from the piccolo inlet. The total temperature drop was approximately 110°F between the inlet and the closed end of the piccolo tube. The surface temperature of the piccolo tube was approximately 50°F lower than that of the fluid inside the tube at all spanwise stations.

The spanwise distribution of the total temperature along the centerline of the piccolo tube is plotted against experimental data in Fig 46. In this plot, the $x$-distance is measured from the piccolo inlet which was close to the tunnel floor. Figure 46 shows that the drop in the hot air temperature from the piccolo tube centerline was nonlinear and exhibited a steep decline near the closed-end of the piccolo tube. The hot air total pressure along the piccolo centerline remained nearly constant as shown in Fig. 47. In general, the computed piccolo centerline total temperature and pressure distributions were in good agreement with the experimental data.
Figures 48 – 51 show the radial variations of several flow properties along the jet axis passing through the 0-deg piccolo hole located at $z=30.52$-in of the FSM (a spanwise location close to Station B in the icing tunnel model). The start of the $x$-axis in Figs. 48 – 51 is at the piccolo centerline ($r=0$) and the end is at the exit of the piccolo jet hole located on the surface of the piccolo tube ($r = 0.5*OD$). The start of the piccolo hole is indicated in the figures with dashed lines. Figure 48 demonstrates that the hot air total pressure dropped by approximately $1.4$ psig across the piccolo hole. The hot air total temperature decreased by approximately $10^\circ F$ from the center of the piccolo tube to the exit of the piccolo jet hole as shown in Fig. 49. The small increase in total temperature observed between $r/OD$ of 0 and 0.25 is not physically possible and it is attributed to grid resolution in this region. The decrease in total temperature between $r/OD$ of 0.25 and 0.5 is the result of external cooling caused by the flow in the diffuser cavity containing the piccolo tube. The radial Mach number and static pressure distributions are shown in Figs. 50 and 51, respectively. As the flow was redirected to the piccolo holes it accelerated considerably from a low speed Mach number at the piccolo center axis to a nearly choked condition at the piccolo jet hole exit.
Flow properties across the diameter of the piccolo hole (0-deg jet at z=30.52-inch) are examined in Figs. 52 – 53. Static pressures, total pressures, and total temperatures at the exit plane of the piccolo hole were sampled at five equally spaced points (see Fig. 53 for locations). The sampled properties were subsequently normalized with their respective jet centerline values. Figure 52 shows that the pressure and temperature profiles were symmetric about the jet axis as expected. Total pressure and temperature were highest at the center and decreased towards the edge of the hole. Furthermore, the total temperature distribution exhibited a large variation across the diameter of the piccolo hole (a 40%
change). Both the jet centerline \( \left( T_{\text{Total}} = 358.18^\circ\text{F}, \ P_{\text{Total}} = 12.61 \ \text{psig}, \ \text{and} \ P_{\text{Static}} = 2.15 \ \text{psig} \) and averaged \( \left( T_{\text{Total}} = 349.41^\circ\text{F}, \ P_{\text{Total}} = 10.15 \ \text{psig}, \ \text{and} \ P_{\text{Static}} = 2.15 \ \text{psig} \) flow properties at the piccolo hole exit are listed in the legend of Fig 52.

![Normalized Boundary Conditions Distribution](image)

**Figure 52.** Normalized boundary conditions at the piccolo jet exit, FSM (Run 26).

![Piccolo hole cross section with total pressure contours](image)

**Figure 53.** Piccolo hole cross section with total pressure contours, FSM (Run 26).

The FSM simulation was followed by two PSM simulations which were used to predict the wing skin temperatures at spanwise Station B. Two sets of boundary conditions at the piccolo jet holes were investigated in the PSM simulations: one for the jet centerline and another one for the averaged flow properties. The conditions at the piccolo holes as well as other boundary surfaces of the PSM included:

- Piccolo holes: Pressure inlet with two different sets of boundary values.
  1. Jet centerline values \( \left( P_{\text{Total}} = 12.61 \ \text{psig}, \ T_{\text{Total}} = 358.18^\circ\text{F}, \ \text{and} \ P_{\text{Static}} = 2.15 \ \text{psig} \)
  2. Average values \( \left( P_{\text{Total}} = 10.15 \ \text{psig}, \ T_{\text{Total}} = 349.41^\circ\text{F}, \ \text{and} \ P_{\text{Static}} = 2.15 \ \text{psig} \)
- Hot air outlets: Pressure outlet with a gauge static pressure of 0.47 psig at the upper and lower exits of the diffuser passages.
- Inner-liner surface: No-slip wall condition with a chordwise temperature profile based on experimental data.
• Piccolo tube: wall condition with a fixed temperature of 303°F. The temperature at the wall was obtained by averaging the piccolo exterior surface temperatures computed with the FSM at $z=30.52$-in.

• Tunnel inlet: pressure inlet with a total pressure of 0.375 psig and a total temperature of 22.98°F as recorded in the experiment ($Re_c=6.8$ million, $M_\infty=0.18$).

• Tunnel outlet: pressure outlet with a zero gauge static pressure. The resulting tunnel speed from the pressure difference between the tunnel inlet and outlet was approximately 115 kts.

• Symmetry conditions at both span ends of the 4.88-in segment.

• No-slip adiabatic walls at the interior surface of the wing skin downstream of the Teflon® inserts.

• No-slip adiabatic walls at the two side walls of the tunnel.

• Wall conditions along portion of the wing LE skin that extended from the inner-liner exit to the insulating insert with a chordwise temperature profile specified from the experimental data.

The PSM simulation was conducted with the pressure-based segregated solver and first-order spatial discretization. The SST $\kappa-\omega$ turbulence model was applied to compute the turbulent internal and external flows. A laminar region was specified around the wing leading edge in accordance to the results from the transition study (Section 3.4). The Teflon® inserts used to reduce the heat transfer between the LE and the main wing body in the experiments was modeled through the specification of low thermal conductivity regions on the wing skin where the inserts were attached.

Figure 54 demonstrates the effects of using averaged (across the jet hole) and centerline piccolo jet hole boundary values on the LE skin temperature. The experimental data shown in Fig. 54 corresponds to the LE skin temperatures measured at station B. The computed temperatures were taken from the PSM at $z=1.83$-inch which corresponded to a spanwise distance halfway between the single- and double-jet stations. The skin temperatures were extracted halfway across the leading edge skin thickness to emulate the readings from the embedded thermocouples in the experimental setup. The surface distance ($s$) in Fig. 54 is the wrap distance measured chordwise along the wing LE surface from the furthermost thermocouples on the lower surface (see Fig. 17 for locations of all the wing LE thermocouples). Figure 54 shows that both averaged and centerline piccolo jet hole boundary values predicted the correct experimental trend. Skin temperatures in both cases were higher near the leading
edge and decreased gradually away from it. Lower temperatures were observed along the upper surface of the wing compared to the temperatures measured along the wing lower surface.

In general, using the averaged boundary values resulted in better agreement with the experimental data than when the centerline values were used. For the averaged boundary values case, the temperatures at the leading edge and along the upper surface were computed accurately with little discrepancy between the computational and the experimental data. But when the centerline values were used, the leading edge temperature was over-predicted by as much as 10°F (5%). Figure 54 also shows that the wing skin temperatures computed using averaged and centerline properties were practically the same at the downstream end of the upper and lower surfaces. This was the result of imposing the same temperature profiles at the aft portion of the LE interior skin (-2.09-inch < s < 0 and 13.81-inch < s < 15.76-inch).

In summary, using averaged piccolo jet hole boundary values from the FSM to determine the inflow boundary conditions in the PSM provided better agreement with the experimental data. This approach was subsequently used for all other simulations to further validate its range of applicability.

Figure 54. Effect of piccolo hole inflow conditions on skin temperature, PSM (Run 26).
Other selected results from the PSM are shown in Figs. 55 – 61. The streamlines shown in Fig. 55 indicate that the external flow accelerated over a longer distance on the upper surface downstream of the stagnation point. The accelerated flow subsequently lowered the static temperature and increased the cooling effect on the upper surface. The figure also shows that the hot air from the piccolo holes impinged on the wing skin and circulated inside the diffuser region before leaving through the upper and lower diffuser passages formed between the inner-liner and the wing skin. In Fig. 56, skin temperature contours show the locations of the hot air jet impingement regions. The contours correspond to an interior surface created halfway between the exterior and interior surfaces of the wing skin.

Figures 57 and 58 show the surface heat flux on the interior of the wing skin at five spanwise stations. Figure 57 corresponds to a double-jet station (z=1.22-inch) and a single-jet station (z=2.44-inch), whereas Fig. 58 illustrates three additional station equally spaced between the single and double jet stations (z=1.525-inch, 1.83-inch, ad 2.135-inch). Experimental data recorded at 27.725-inch above the tunnel floor (a spanwise station nearly halfway between the single and the adjacent double-jet stations) are also provided in both Figs. 57 and 58 for comparison. At each spanwise jet station, the computed heat flux was highest at the location of jet impingement and decreased rapidly away from the jet. Figure 58 shows that the heat flux distribution computed at each spanwise station was not symmetrical along the upper and lower surface. The -45º jet contributed a higher local heat flux than the +45º jet as seen in the results taken at z=1.525-inch. This is attributed to higher wing skin temperatures along the lower surface.

In general, the experimental data were best correlated with the computed surface heat flux taken at z=1.83-inch, the station halfway between a single- and a double-jet stations. The surface heat flux values computed at spanwise station z=1.83-inch were similar to the recorded experimental data (i.e. between 5 and 15 W/in²). Note that the accuracy of the surface heat flux measurements in the experiment depended on both the thickness of the sensors and the thickness of the glue layer between the sensors and the wing interior surface.

Figures 59 and 60 show sectional contours of total temperature and gauge total pressure, respectively, at five spanwise stations containing the hot air jets in the PSM. Figure 61 shows that the 0-deg jet impinging on the concave surface of the wing skin was non-axisymmetric. More flow was directed towards the lower surface due to the inclination of the wing surface with respect to the jet axis. Figure 62
shows that the total pressure remained high throughout the jet core and over the impingement region. The jet shear layers that bounded the jet core are also visible in the contour maps in Figs. 61 and 62.

Figure 55. Streamlines colored by total temperature in PSM (Run 26).

Figure 56. External skin temperature, PSM (Run 26).
Figure 57. Interior surface heat flux comparison, single- and double-jet stations, PSM (Run 26).

Figure 58. Interior surface heat flux comparison, in-between jet stations, PSM (Run 26).

Figure 59. Total temperature contours at 5 jet stations in PSM (Run 26).

Figure 60. Total pressure contours at 5 jet stations in PSM (Run 26).

Figure 61. Contours of Mach number for 0-deg jet in PSM (Run 26).

Figure 62. Contours of total pressure for 0-deg jet in PSM (Run 26).
3.6 Anti-Icing System Parametric Study

A total of seven additional cases were analyzed with FLUENT to investigate the effects of external flow conditions, hot air flow conditions, and piccolo circumferential placement on LE skin temperatures. The experimental flow conditions used in the FSM simulations are listed in Table 6 (Section 3.1.3). The boundary conditions obtained from the FSM results along with the mass flow (per hole) reported in the PSM simulations are presented in Table 8. The mass flows were found to be proportional to the piccolo inlet mass flow used in the experiments.

Table 8. PSM Piccolo hole boundary conditions.

<table>
<thead>
<tr>
<th>Test Runs</th>
<th>Piccolo Hole Boundary Conditions</th>
<th>Piccolo Tube Surface Temperature °F</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Total Temperature °F</td>
<td>Total Pressure psig</td>
</tr>
<tr>
<td>26</td>
<td>349.41</td>
<td>10.15</td>
</tr>
<tr>
<td>42</td>
<td>365.86</td>
<td>14.85</td>
</tr>
<tr>
<td>58</td>
<td>346.50</td>
<td>10.02</td>
</tr>
<tr>
<td>62</td>
<td>252.88</td>
<td>8.65</td>
</tr>
<tr>
<td>66</td>
<td>306.49</td>
<td>2.38</td>
</tr>
<tr>
<td>76</td>
<td>253.86</td>
<td>38.03</td>
</tr>
<tr>
<td>107</td>
<td>356.22</td>
<td>12.36</td>
</tr>
<tr>
<td>119</td>
<td>342.26</td>
<td>11.78</td>
</tr>
</tbody>
</table>

The effects of each system parameter on the wing skin temperature are presented next. The computed and experimentally recorded temperatures at three sub-regions of the LE skin, namely the forward region, the upper region, and the lower region, were averaged and are reported in Table 9 along with percentage errors. Figure 63 illustrates the location and extent of these regions. The forward LE region was located between surface distances \( s \) of 6.59-inch and 6.87-inch. This region contained the two thermocouples installed near the most forward portion of the wing LE (see also Fig. 17 in Section 3.1.2) and was important because it is typically the location with highest ice accumulation. The upper region was defined as 9.61-inch \( \leq \) \( s \leq \) 13.81-inch and corresponded to the portion of wing upper surface positioned above the inner-liner passage. Similarly, the lower region was defined as 0-inch \( \leq \) \( s \leq \) 4.13-inch. These regions were significant because runback ice builds there. In the experiment, the upper and
lower regions, as defined above, contained six thermocouples each installed inside the wing LE skin. The experimental values reported in Table 9 were obtained from the thermocouple data. The CFD values were computed from the PSM simulations. The percent errors of the computed temperatures with respect to experimental data (Eqn. 19) are also provided in Table 9.

\[
\frac{|T_{\text{Avg,CFD}} - T_{\text{Avg,Exp}}|}{T_{\text{Avg,Exp}}} \times 100
\]  

(19)

Table 9. Computed and experimental skin temperatures.

<table>
<thead>
<tr>
<th>Test Run</th>
<th>Forward Region 6.59 ≤ s ≤ 6.87-inch</th>
<th>Upper Region 9.61 ≤ s ≤ 13.81-inch</th>
<th>Lower Region 0 ≤ s ≤ 4.13-inch</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(T_{\text{Avg,CFD}}) (^{\circ}\text{F})</td>
<td>(T_{\text{Avg,Exp}}) (^{\circ}\text{F})</td>
<td>% error</td>
</tr>
<tr>
<td>26</td>
<td>192.63</td>
<td>192.90</td>
<td>0.14</td>
</tr>
<tr>
<td>42</td>
<td>214.17</td>
<td>214.50</td>
<td>0.15</td>
</tr>
<tr>
<td>58</td>
<td>168.07</td>
<td>171.26</td>
<td>1.86</td>
</tr>
<tr>
<td>62</td>
<td>141.38</td>
<td>140.62</td>
<td>0.54</td>
</tr>
<tr>
<td>66</td>
<td>122.35</td>
<td>129.98</td>
<td>5.88</td>
</tr>
<tr>
<td>76</td>
<td>178.54</td>
<td>176.97</td>
<td>0.88</td>
</tr>
<tr>
<td>107</td>
<td>194.95</td>
<td>200.79</td>
<td>2.91</td>
</tr>
</tbody>
</table>

Figure 63. Leading edge forward, upper, and lower regions.

The computed skin temperatures shown in Figs. 64 – 68 were extracted at the spanwise station \(z=1.83\)-inch (halfway between the single- and double-jet spanwise stations) and halfway across the leading edge skin thickness. The CFD results are compared with experimental data recorded at Station B.
of the wing model. Note that the surface distance, s, in Figs. 64 – 68 is the wrap distance measured chordwise along the wing LE surface from the furthermost downstream thermocouple on the lower surface.

In addition, Figs. 69 – 73 show piccolo centerline total temperature distributions obtained from the FSM compared against values recorded experimentally. Contours of skin temperature (Fig. 74) and hot air jet Mach number (Fig. 75) are also presented to further illustrate the differences among the cases considered.

3.6.1 Effect of External Flow Conditions (Runs 26, 58)

Computed and experimental wing skin temperatures were compared for a cold hold case (Run 58) and a warm hold case (Run 26). The cold hold case was characterized by external airspeed of 110 knot, and -22ºF static external flow temperature. The wing skin temperatures computed from the two PSM simulations performed are compared with corresponding experimental data in Fig. 64. The results in Fig. 64 show that for a similar piccolo hot air temperature and mass flow rate, the wing skin temperature decreased as the external cold air temperature was decreased (increased external cooling). The drops in skin temperature due to the acceleration of the flow were approximately 39ºF and 36ºF at the downstream ends of the upper and lower wing regions, respectively, and 24.5ºF at the forward region. The percent errors in Table 9 for all three wing regions indicate that in general, the CFD results for the warm hold case (Run 26) were in better agreement with the experimental data compared to the corresponding results for the cold hold case (Run 58).

The external flow conditions did not have any discernible effect on the total temperature spanwise distribution at the piccolo centerline as shown in Fig. 69.

3.6.2 Effect of Hot Air Temperature (Runs 26, 62)

The effect of piccolo hot air temperature on wing skin temperatures is shown in Fig. 65. The piccolo inlet total temperatures in Runs 26 and 62 were set to 373.8ºF and 263.5ºF, respectively. In the simulations performed with the PSM, the total temperature specified at the piccolo jet holes in Run 26 was 96.5ºF higher than the one used in Run 62. For similar external flow conditions and piccolo inlet mass flow rates, increasing piccolo hot air temperature raised the skin temperature at the forward region of the LE by approximately 52ºF. The gains in wing skin temperatures were most prominent at the forward
region, followed by the lower region and lastly by the upper region. The percent difference between the computed temperature obtained from the PSM simulations and the experimental data at the forward, upper and lower regions for Runs 26 and 62 are listed in Table 9. In general, the observed differences were small.

The spanwise total temperature distribution was computed satisfactorily for both cases as shown in Fig. 70.

### 3.6.3 Effect of Hot Air Mass Flow at High Temperature (Runs 26, 42, 66)

For similar external flow conditions and piccolo inlet temperature (on the order of 370ºF), wing skin temperatures obtained for three cases with different piccolo mass flow rates are compared in Fig. 66. The mass flow rates investigated included 1.83 lbm/min (Run 66), 4.27 lbm/min (Run 26), and 5.31 lbm/min (Run 42). Figure 66 shows that skin temperatures over the protected area increased as the piccolo mass flow rate was increased. In all cases, good correlations between the numerical and experimental data were observed at the upper region with a maximum percent error of 2.53% corresponding to Run 66 as shown in Table 9. The highest discrepancy was observed at the forward region of the wing LE for the low mass flow case (Run 66) where the computed skin temperature was 7.6ºF (or 5.88%) lower than the experimental data.

In addition, as Fig. 71 shows, the run with the lowest mass flow rate (Run 66 – 1.83 lbm/min) experienced more spanwise losses in piccolo centerline total temperature as compared with Runs 26 and 42. Finally, Figs. 74 and 75 show that Run 66 had the lowest skin temperature of all the cases examined. For this case the jet flow Mach number was 0.5 as shown in Fig. 75. For the other cases investigated the jet exit Mach number was greater than 0.8.

### 3.6.4 Effect of Hot Air Mass Flow at Low Temperature (Runs 62, 76)

The effect of hot air mass flow was also evaluated at a lower piccolo inlet temperature (approximately 260ºF). The mass flow rates investigated included 4.12 lbm/min (Run 62) and 10.7 lbm/min (Run 76). Figure 67 shows that skin temperatures increased with increasing piccolo mass flow rates, as was the case for the high piccolo inlet temperature previously presented in Section 3.6.3. Gains in LE skin temperature were observed over all three regions of the wing LE skin. The temperature gains at the forward region and downstream end of the lower surface were both approximately 39ºF higher for
the 10.7 lbm/min case compared to the 4.12 lbm/min case. The percent differences between the experimental and analytical skin temperatures for all three wing regions are presented in Table 9 and were in the range of 0.54% to 6.17%. Figure 72 shows that the decrease in total temperature with spanwise distance along the piccolo tube centerline was lower for Run 76 (10.7 lbm/min) due to the significant increase in the mass flow rate. Figure 72 also shows that the computational results over-predicted the piccolo temperature for Run 76 at spanwise distances further than 40 inches from the floor. This is attributed to heat loss through the end of the tube that was not accounted for with the isothermal boundary condition used in the FSM.

In addition, the piccolo hole jet flow in the PSM was supersonic in Run 76. As Fig. 75 (f) shows, the jet initially expanded, then decelerated suddenly through a shock wave, and finally reaccelerated until the stagnation region where it decelerated again through another shock wave [66]. Run 76 was the only case investigated that presented a supersonic jet flow.

3.6.5 Effect of Piccolo Configuration (Runs 26,107)

Leading edge skin temperatures were computed for two piccolo tube configurations: the baseline P45-0-M45 (Run 26) and the modified P70-P10-M40 (Run 107). Figure 68 shows that for similar internal hot air and external cold air conditions, the P70-P10-M40 configuration produced gains in upper surface temperatures without compromising peak temperatures by increasing the amount of hot air directed towards the upper surface. According to the computed temperatures listed in Table 9, the gain in skin temperatures over the wing upper region was 14.5ºF. However, the P70-P10-M40 piccolo tube configuration produced a small reduction of approximately 5.6ºF in wing skin temperatures over the lower region compared to the P45-0-M45 design. The percent difference between the experimental and computed temperature distributions ranged from 0.14% to 3.57% with the best correlation found near the forward region of the wing. Note that the computed gain in the skin temperatures over the wing forward region due to the use of the P70-P10-M40 piccolo was 2.3ºF as compared with 7.9ºF recorded in the experiments. The reason for the higher peak temperature recorded in the experiment was due to a spanwise misalignment of the P70-P10-M40 piccolo jets with respect to the thermocouples embedded in the skin. While the computed temperatures correspond to a spanwise station between the single- and double-jet stations, the experimental results presented correspond to a spanwise station close to the
thermocouples at station B but also slightly shifted closer to the single-jet station as evidenced by the sharp temperature peak not seen in any other case.

For Run 107, the spanwise total temperature distribution was not predicted with similar accuracy in the regions closer to the ceiling as for other cases. This is also due to heat transfer unaccounted for with the isothermal boundary condition at the end of the piccolo tube.

Figure 64. Effect of external flow conditions on LE skin temperature, PSM (Runs 26, 58).

Figure 65. Effect of hot air temperature on LE skin temperature, PSM (Runs 26, 62).

Figure 66. Effect of piccolo mass flow at high hot air temperature on LE skin temperature, PSM (Runs 26, 42, 66).

Figure 67. Effect of piccolo mass flow at low hot air temperature on LE skin temperature, PSM (Runs 62, 76).
Figure 68. Effect of piccolo configuration on LE skin temperature, PSM (Runs 26, 107).

Figure 69. Comparison of total temperature at the piccolo centerline, FSM (Runs 26, 58).

Figure 70. Comparison of total temperature at the piccolo centerline, FSM (Runs 26, 62).
Figure 71. Comparison of the total temperature at the piccolo centerline, FSM (Runs 26, 42, 66).

Figure 72. Comparison of the total temperature at the piccolo centerline, FSM (Runs 62, 76).

Figure 73. Comparison of the total temperature at the piccolo centerline, FSM (Runs 26, 107).
Figure 74. Comparison of temperature contours on the thermocouple face, PSM.
Figure 75. Comparison of Mach number contours for a hot-air jet, PSM.
CHAPTER 4
CONCLUSIONS AND RECOMMENDATIONS

4.1 Conclusions

A computational methodology was developed for the analysis of a hot air ice protection system with a commercial CFD code. Studies were performed for a 72-in span, 60-in chord two-dimensional business jet wing equipped with a bleed air ice protection system and installed in an icing tunnel test section operating in dry air flow conditions. Parametric investigations examined the effects of external flow conditions, hot air temperature, hot air mass flow rate, and piccolo configuration on the leading edge skin temperatures. The results were compared with available experimental data from testing of a similar setup in the NASA Glenn Icing Research Tunnel.

Two computational models were employed for the simulations: a full-span (72-in) partial-chord (forward 10% of the wing chord) wing model (FSM) and a partial-span (4.88-in) full-chord (60-in) model (PSM). Meshing and computational efforts were reduced by introducing geometric simplifications to the icing tunnel wing models. The FSM had square piccolo holes and a three-in-line piccolo jet hole pattern. The PSM had a shortened span and did not simulate the flow inside the piccolo tube.

The FSM was used to replicate the total temperature and total pressure experimental measurements along the piccolo tube centerline. Boundary conditions for the PSM were implemented with hot air data computed with the FSM and experimental data obtained from icing tunnel tests. The PSM was used to investigate the effect of hot air system parameters on wing leading edge skin temperatures and bleed air system interior flow properties.

Preliminary turbulence model, grid resolution, and spatial discretization studies were conducted prior to the anti-icing system simulations. A viscous flow analysis, including a review of the hot air impinging jet flow and the evaluation of eddy-viscosity turbulence models available in FLUENT, was conducted to select the most suitable formulation to simulate both the external wall-bounded flow and the internal jet impinging flow accurately and with physically feasible results. The turbulence model study explored the complex phenomena in impinging jet flow. The study focused on a single jet impinging on a flat surface and compared five different eddy-viscosity models (i.e. S-A, standard $\kappa - \varepsilon$, ...)
Realizable $\kappa-\varepsilon$, standard $\kappa-\omega$, and SST $\kappa-\omega$) at three different levels of turbulence intensity (1%, 3%, and 5%). The results showed that the SST $\kappa-\omega$ turbulence model provided the most constant and acceptable results.

The grid resolution and spatial discretization studies focused on a 3D single-jet geometry. Three different grid resolution levels and two discretization schemes were considered. The study concluded that a grid resolution based on 48 equally-spaced nodes around the jet holes circumference was sufficient to predict the recovery temperatures in a jet impingement scenario. In addition, the first order discretization scheme was selected as it produced satisfactory results with lower computational effort. Grid resolution requirements were different for the FSM and the PSM: the FSM was considerably coarser than the PSM. Five nodes were distributed evenly across the width of each square piccolo jet hole in the FSM compared to 18 in the PSM.

Another study evaluated two sets of boundary conditions at the piccolo jet holes in the PSM simulations. Numerical results obtained from the FSM showed good correlation with experimental data for hot air total pressure and total temperature along the piccolo centerline. The most significant reduction in total temperature along the span was for the case with the lowest bleed mass flow rate. Total air pressure remained constant over the entire piccolo length for all cases. The study also showed that the piccolo flow properties varied along the radial direction from the piccolo tube axis to the center of each piccolo jet hole. Hot air flow properties also varied across the jet hole plane. Data extracted from the FSM piccolo jet holes included centerline and averaged data. Results suggested that both approaches predicted the correct experimental trend, but using averaged boundary values provided more accurate wing skin temperature distributions in all regions of the wing leading edge.

Parametric studies were performed with the full-span and partial-span models after successfully identifying a turbulence model and a methodology for the selection of boundary conditions. The results showed that the FSM predicted with a fair degree of accuracy the spanwise distribution of total pressure and temperature along the piccolo centerline in all cases. Numerical results obtained with the PSM showed good agreement with experimental data for wing leading edge temperatures. For the cases investigated, the percent difference between the experimental and computed temperature distributions over the forward, upper, and lower surface regions of the leading edge ranged from 0.14% to 13.7%.
4.2 Recommendations for Future Work

The results from the numerical simulations have demonstrated the usefulness of CFD for the design and certification of anti-icing systems. The following recommendations point at a continuing improvement of the simulations:

- A computational methodology for dry air simulations was developed and validated against a limited set of experimental data. It is necessary to increase the number of parametric studies, particularly focusing on cases with cold hold conditions and additional wing angles of attack.

- Wet external flow analysis remains a step in the development of a general computational methodology. Fully evaporative or running wet simulations with water impingement require a time-accurate simulation of a multiphase flow containing a mixture of air flow and water droplets. The simulation must include water film dynamics and consider the effect of ice accretion on the external flow energy balance.

- The present work used the CFD code FLUENT for the anti-icing simulations. The CFD code CFX was use exclusively for external flow computations. A continuing effort at WSU should explore the use of other commercial CFD packages such as CFX, Star-CD, and CFD++.

- The computational methodology must be revised periodically to remain up-to-date with new advances in CFD including new or improved eddy-viscosity, second moment closure (SMC) turbulence models, or hybrid models with eddy-viscosity formulations and anisotropic second moment closure. These models could have the potential to overcome the deficiencies of the Boussinesq assumption by including the calculation of streamline curvature.

- Finally, the simulation of a high grid resolution full-span model must be eventually pursued. Ideally, the model should reproduce the full length of the tunnel, the full chord of the wing, the return duct formed by the front spar and the inner-liner skin inside the wing, and the exhaust of the return duct. A high resolution FSM could eliminate the need for a partial-span model and simplify the number of boundary conditions and thermal profiles required in the simulation of a bleed air ice protection system. However, the amount of computational resources required to mesh and to process such model pose considerable challenges.
LIST OF REFERENCES


APPENDICES
1.0 Geometry

The geometry was divided into 3 sections: wing, anti-icing system, and tunnel.

1.1 Airfoil

A Modern Business Jet airfoil sized to a 60-in chord length was used. Figure 1 shows the airfoil cross section. It had an under-cambered region on its pressure side near the trailing edge and a maximum thickness-to-chord ratio \((t/c_{\text{max}})\) of 0.12.

![Figure 1. HAARP Airfoil](image)

1.2 Wing Geometry

The CFD analysis was based on a 60-inch chord wing section with a 4.88-inch wingspan (Figure 2). It had a 0.1-inch thick skin, which was divided into an inner and outer skin in the frontal section (See Figure 3). Two thermal insulating sections were located at 14%-chord.
Figure 2. Wing Section (4.88-in span)

Figure 3. Wing Frontal Section: Outer and inner Skin
1.3 Anti-Icing System Geometry

The Anti-Icing System was located in the frontal section of the wing. It consisted of a 1.25-inch diameter piccolo tube with several 0.052-inch diameter piccolo jet holes in diamond configurations along the span. Equally-oriented holes were 2.44-inch apart. As Fig. 4 shows, cone regions were projected from the holes into the wing skin to facilitate the mesh generation and to increase to target grid resolution in those areas. Also, an inner liner was located behind the piccolo hole extending parallel to the inner skin until the 9.6% chord location where there were outlet faces on both upper and lower sides (see Fig. 5).

![Figure 4. Cone Projections on Wing Skin](image)
1.3.1 Piccolo-Hole Configurations

Two piccolo-jet hole configurations were modeled: -40°/10°+70° and -45°/0°+45°. The meshing process was similar for both cases. Table 1 lists the nozzle-to-plate distances for each piccolo jet hole. The average nozzle-to-plate distance was approximately 3.62.

Table 1. Piccolo jet holes nozzle-to-plate distances.

<table>
<thead>
<tr>
<th>Hole Angle</th>
<th>+45°</th>
<th>0°</th>
<th>-45°</th>
<th>+70°</th>
<th>+10°</th>
<th>-40°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length (zn)</td>
<td>0.1297</td>
<td>0.1479</td>
<td>0.2435</td>
<td>0.2443</td>
<td>0.1292</td>
<td>0.2355</td>
</tr>
<tr>
<td>zn/d</td>
<td>2.49</td>
<td>2.84</td>
<td>4.68</td>
<td>4.70</td>
<td>2.46</td>
<td>4.53</td>
</tr>
</tbody>
</table>

1.4 Tunnel Geometry

The tunnel geometry was modeled after the NASA Glenn Research Center Icing Research Tunnel (IRT). In the PSM, the test section width was set to 9-ft to form a rectangular cross section enclosing the wing spanwise segment (i.e. 4.88-inch tall). The tunnel inlet and outlet faces were located 125.4-ft from the wing leading edge. The tunnel geometry was rotated -3° with respect to the z-axis to create a wing angle of attack of +3°.
2.0 Mesh

2.1 Mesh Design

Grid generation took advantage of Hexahedral/Wedge elements that GAMBIT generates with a COOPER meshing scheme. The diffuser section was meshed with tetrahedral elements using a TGRID scheme. The solution in the diffuser was expected to be very sensitive to the grid resolution around the jet regions. Therefore, a fine tetrahedral mesh was used to increase the grid resolution in these areas using information obtained from preliminary grid resolution studies. Fixed size functions were employed to ensure a constant growth rate from the high density jet regions to the rest of the diffuser section and to reduce the number of elements in areas of the diffuser were the flow gradients may be lenient.

The wing skin was meshed with wedge elements and hexahedral elements wherever possible. The tunnel regions surrounding the wing closely were meshed with tetrahedral elements and meshed size functions. Table 2 shows the size functions used, their parameters, sources, and attachments. Figure 6 shows the regions were the size functions were applied.

<table>
<thead>
<tr>
<th>Size Function</th>
<th>Type</th>
<th>Sources</th>
<th>Attachment</th>
<th>Start Size (in)</th>
<th>Growth Rate (in)</th>
<th>Max Size (in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Front LE</td>
<td>Meshed</td>
<td>Outer skin faces</td>
<td>Volume region forward of the wing leading edge</td>
<td>-</td>
<td>1.1</td>
<td>0.50</td>
</tr>
<tr>
<td>Back TE</td>
<td>Meshed</td>
<td>Outer Skin faces</td>
<td>Volume region behind the wing trailing edge</td>
<td>-</td>
<td>1.2</td>
<td>0.50</td>
</tr>
<tr>
<td>Inside (1 – 5)</td>
<td>Fixed</td>
<td>High density regions</td>
<td>Volumes inside the diffuser bay</td>
<td>0.003</td>
<td>1.1</td>
<td>0.05</td>
</tr>
<tr>
<td>Low wing 1/2</td>
<td>Meshed</td>
<td>Outer skin faces</td>
<td>Volume region on the wing lower surface</td>
<td>-</td>
<td>1.1</td>
<td>0.50</td>
</tr>
<tr>
<td>Back 1</td>
<td>Meshed</td>
<td>Edges of volume “Back TE”</td>
<td>Faces in volume behind of “Back TE”</td>
<td>-</td>
<td>1.2</td>
<td>0.50</td>
</tr>
<tr>
<td>Front 1</td>
<td>Meshed</td>
<td>Edges of volume “Front TE”</td>
<td>Faces in volume forward of “Front LE”</td>
<td>-</td>
<td>1.2</td>
<td>0.50</td>
</tr>
<tr>
<td>Wing US 1/2/3</td>
<td>Meshed</td>
<td>Outer Skin faces</td>
<td>Volume region on wing upper surface</td>
<td>-</td>
<td>1.1</td>
<td>0.50</td>
</tr>
<tr>
<td>Top/Bottom</td>
<td>Meshed</td>
<td>Top volume faces</td>
<td>Top volume</td>
<td>-</td>
<td>1.3</td>
<td>3.75</td>
</tr>
<tr>
<td>Back 2</td>
<td>Meshed</td>
<td>Rear volume edge</td>
<td>Rear Volume of the tunnel geometry</td>
<td>-</td>
<td>1.3</td>
<td>30</td>
</tr>
<tr>
<td>Front 2</td>
<td>Meshed</td>
<td>Front volume edge</td>
<td>Front volume of the tunnel geometry</td>
<td>-</td>
<td>1.3</td>
<td>30</td>
</tr>
</tbody>
</table>
2.2 Boundary Layers

The grid contained seven different boundary layers attached to several faces and edges. Inside the diffuser region, there were three boundary layers: one attached to the piccolo tube (including the piccolo holes), another one attached to the inner skin, and another one to the inner liner. The grid took full
advantage of the boundary layer algorithms included in GAMBIT to maximize resolution in important areas, particularly near the expected impingement location of the hot air jets. Table 3 lists the main characteristics of the boundary layers used.

Table 3. Boundary Layers

<table>
<thead>
<tr>
<th>Boundary Layer Name</th>
<th>Algorithm</th>
<th>First Row (a) (in)</th>
<th>Growth Factor (b/a)</th>
<th>Rows</th>
<th>Last Percent (c/w)</th>
<th>Attachment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Piccolo Tube</td>
<td>Uniform</td>
<td>1.0x10^-5</td>
<td>1.36</td>
<td>20</td>
<td>-</td>
<td>Faces</td>
</tr>
<tr>
<td>Inner Skin</td>
<td>Aspect Ratio (last)</td>
<td>1.0x10^-5</td>
<td>-</td>
<td>20</td>
<td>25</td>
<td>Faces</td>
</tr>
<tr>
<td>Inner Liner</td>
<td>Aspect Ratio (last)</td>
<td>1.0x10^-5</td>
<td>-</td>
<td>20</td>
<td>25</td>
<td>Faces</td>
</tr>
<tr>
<td>Gap</td>
<td>Uniform</td>
<td>1.0x10^-5</td>
<td>1.4</td>
<td>20</td>
<td>-</td>
<td>Faces</td>
</tr>
<tr>
<td>Wing Front</td>
<td>Aspect Ratio (last)</td>
<td>1.0x10^-5</td>
<td>-</td>
<td>23</td>
<td>50</td>
<td>Faces</td>
</tr>
<tr>
<td>Wing Rear</td>
<td>Aspect Ratio (last)</td>
<td>1.0x10^-5</td>
<td>-</td>
<td>23</td>
<td>50</td>
<td>Edges</td>
</tr>
<tr>
<td>Tunnel</td>
<td>Aspect Ratio (last)</td>
<td>1.0x10^-5</td>
<td>-</td>
<td>30</td>
<td>50</td>
<td>Edges</td>
</tr>
</tbody>
</table>

2.3 Resulting Mesh

Figures 7 – 12 show different perspectives of the resulting mesh.

Figure 7. Side View of Partial Span Model Mesh, P45-0-M45
Figure 8. Isometric View of Partial Span Model skin mesh, P45-0-M45

Figure 9. Isometric View of Partial Span Model – Leading Edge Outer Skin Mesh, P45-0-M45

Figure 10. Isometric View of Partial Span Model Interior Mesh and Symmetry Face, P45-0-M45

Figure 11. Isometric View of Partial Span Model Interior Mesh, P45-0-M45.
2.4 Continuum Types

The four mediums listed below were specified in GAMBIT before exporting the grid. The "Cold Air" region was further sub-divided in FLUENT to create the laminar region around the wing leading edge.

Table 3. Continuum Types

<table>
<thead>
<tr>
<th>Continuum</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wing Skin</td>
<td>Solid</td>
<td>Region enclosed by the inner and outer skins.</td>
</tr>
<tr>
<td>Insulator</td>
<td>Solid</td>
<td>Region in the skin of the wing, but with a different material.</td>
</tr>
<tr>
<td>Cold Air</td>
<td>Fluid</td>
<td>Region surrounding wing and enclosed by the tunnel.</td>
</tr>
<tr>
<td>Hot Air</td>
<td>Fluid</td>
<td>Internal flow region bounded by the inner skin, the inner liner, and the piccolo tube.</td>
</tr>
</tbody>
</table>

2.5 Boundary Conditions

Table 4. Boundary Conditions

<table>
<thead>
<tr>
<th>Boundary Condition</th>
<th>Type</th>
<th>Boundary Condition</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tunnel Inlet</td>
<td>Pressure Inlet</td>
<td>Right Tunnel Wall</td>
<td>Wall</td>
</tr>
<tr>
<td>Tunnel Outlet</td>
<td>Pressure Outlet</td>
<td>Left Tunnel Wall</td>
<td>Wall</td>
</tr>
<tr>
<td>Piccolo Holes (Full)</td>
<td>Pressure Inlet</td>
<td>Inner Skin Surface</td>
<td>Wall</td>
</tr>
<tr>
<td>Piccolo Holes (Half)</td>
<td>Pressure Inlet</td>
<td>Outer Skin Surface</td>
<td>Wall</td>
</tr>
<tr>
<td>Hot Air Outlet (Upper Surface)</td>
<td>Pressure Outlet</td>
<td>Inner Liner Surface</td>
<td>Wall</td>
</tr>
<tr>
<td>Hot Air Outlet (Lower Surface)</td>
<td>Pressure Outlet</td>
<td>Piccolo Tube Surface</td>
<td>Wall</td>
</tr>
<tr>
<td>Symmetry Planes</td>
<td>Symmetry</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
APPENDIX B

CFX TRANSITION MODEL FORMULATION

The local correlation-based transition model developed by Menter and Langtry [72, 82] relies on two transport equations for transition onset momentum-thickness Reynolds number and Intermittency. The first equation relates empirical correlations to the onset criteria, whereas the second equation triggers transition by interacting with the turbulence model equations.

The transition onset momentum thickness Reynolds number ($Re_{\theta_1}$) is calculated in the freestream using an empirical correlation and is then diffused into the boundary layer. The empirical correlation (Eqns. 1 and 2) as presented in [82] accounted for turbulence intensity ($Tu$), pressure gradients ($\lambda_\theta$), and flow acceleration in the streamwise direction ($K$). The correlation is presented here for referential purposes as it was modified later in [72] to improve the prediction of natural transition. The new correlation is proprietary.

$$Re_{\theta_1} = 803.73[Tu + 0.6067]^{1.827} F(\lambda_\theta, K)$$

$$F(\lambda_\theta, K) = \begin{cases} 1 - \left[-10.32\lambda_\theta - 89.47\lambda_\theta^2 - 265.51\lambda_\theta^3\right] e^{-\left(Tu/3.0\right)} , & \lambda_\theta \leq 0 \\ 1 + \left[0.0962[K10^6] + 0.48[K10^6]^2 + 0.0141[K10^6]^3\right] \times (1 - e^{-\left[Tu/1.5\right]}) + \left[0.556[1 - e^{-23.9\lambda_\theta}]\right] e^{-\left[Tu/1.5\right]} , & \lambda_\theta > 0 \end{cases}$$

The pressure gradient parameter (known as Thwaites' pressure gradient coefficient), the flow acceleration parameter, and the acceleration in the streamwise direction term are given by Equations 3, 4, and 5, respectively.

$$\lambda_\theta = \left(\theta^2 / \nu\right) \frac{dU}{ds}$$
\[ K = \left( \frac{\nu}{U^2} \right) dU / ds \]  

\[ \frac{dU}{ds} = \left( \frac{u}{U} \right) \frac{dU}{dx} + \left( \frac{v}{U} \right) \frac{dU}{dy} + \left( \frac{w}{U} \right) \frac{dU}{dz} \]  

The empirical correlation and the pressure and flow acceleration parameters are limited in [82] for numerical robustness to the following ranges:

\[-0.1 \leq \lambda_0 \leq 0.1 \]
\[-3 \times 10^{-6} \leq K \leq 3 \times 10^{-6} \]
\[ \text{Re}_0 \geq 20 \]

Once \( \text{Re}_0 \) is computed in the freestream, a transport equation (Eqn. 6) is used to diffuse the freestream information into the boundary layer regions.

\[ \frac{\partial (\rho \text{Re}_0)}{\partial t} + \frac{\partial (\rho U_j \text{Re}_0)}{\partial x_j} = P_{\text{Re}} + \frac{\partial}{\partial x_j} \left[ \sigma_{x_i} (\mu + \mu) \frac{\partial \text{Re}_0}{\partial x_j} \right] \]  

\[ P_{\text{Re}} = c_{\text{Re}} \frac{\rho}{t} \left( \text{Re}_0 - \text{Re}_{\text{Re}} \right) (1.0 - F_{\text{Re}}) \]  

\[ t = \frac{500 \mu}{\rho U^2} \]  

\[ F_{\text{Re}} = \min \left\{ \max \left[ F_{\text{wake}}, e^{-4y/\delta}, 1.0, \left( \frac{1.0 - 1/c_{z_2}}{\gamma - 1/c_{z_2}} \right)^2 \right], 1.0 \right\} \]  

\[ \delta = \frac{50 \Omega y}{U} \delta_{\text{BL}}; \quad \delta_{\text{BL}} = \frac{15}{2} \theta_{\text{BL}}; \quad \theta_{\text{BL}} = \frac{\text{Re}_{\text{BL}} \mu}{\rho U} \]  

\[ \text{Re}_{\text{BL}} = \frac{\rho \Omega y}{\mu} \]  

\[ F_{\text{wake}} = e^{-\left( \frac{\text{Re}_{\text{BL}}}{1000} \right)^2} \]
A critical transition onset momentum thickness Reynolds number is calculated using another proprietary empirical correlation \( R_{e,\theta} = f(Re_{\theta}) \) and is used in the computation of the intermittency parameter \( \gamma \).

\[
\frac{\partial (\rho \gamma)}{\partial t} + \frac{\partial (\rho U_j \gamma)}{\partial x_j} = P_{\gamma 1} - E_{\gamma 2} + P_{\gamma 2} - E_{\gamma 3} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_f} \right) \frac{\partial \gamma}{\partial x_j} \right]
\]

(13)

\[ P_{\gamma 1} = 2F_{\text{onset}} \rho S \beta \gamma \]

(14)

\[ E_{\gamma 1} = P_{\gamma 1} \gamma \]

(15)

\[ F_{\text{onset}} = \max(F_{\text{onset2}} - F_{\text{onset3}}, 0) \]

(16)

\[ F_{\text{onset2}} = \min[\max(F_{\text{onset1}}, F_{\text{onset1}}^4), 2.0] \]

(17)

\[ F_{\text{onset1}} = \frac{R_{e,\theta}}{2.193Re_{\theta}} \]

(18)

\[ R_{\gamma} = \frac{\rho U^2 S}{\mu} \]

(19)

\[ F_{\text{onset3}} = \max \left[ 1 - \left( \frac{R_{\gamma}}{2.5} \right)^3, 0 \right] \]

(20)

\[ R_{\gamma} = \frac{\delta K}{\mu U} \]

(21)

\[ P_{\gamma 2} = 2c_{\gamma 1} \rho \Omega \Omega F_{\text{turb}} \]

(22)

\[ E_{\gamma 2} = c_{\gamma 2} P_{\gamma 2} \gamma \]

(23)

\[ F_{\text{turb}} = e^{-(R_{\gamma} / 4)^4} \]

(24)

A final calculation is done to account for separation. The method for this computation was also updated in [72] from the original formulation presented in [82]:

101
The constants for the transition onset momentum thickness Reynolds number and the intermittency computations are listed below. $F_{\text{length}}$ in Equation 14 is obtained from a proprietary empirical correlation [72, 82].

$$c_{\gamma_1} = 0.03;\; c_{\gamma_2} = 50;\; c_{\gamma_3} = 0.5;\; \sigma_f = 1.0$$

$$c_{\theta_1} = 0.03;\; \sigma_{\theta_1} = 2.0$$

The transition model interacts exclusively with the SST $\kappa$-$\omega$ turbulence model in CFX. The final intermittency parameter directly multiplies the production and destruction of turbulent kinetic energy terms. An additional modification is performed to the blending function controlling the switch between the $\kappa$-$\omega$ and $\kappa$-$\varepsilon$ formulations in order to prevent the blending function from activating too early in a laminar boundary layer.

$$P_k^{\text{New}} = \gamma_{\text{eff}} P_k^{\text{Orig}};\; D_k^{\text{New}} = \min[\max(\gamma_{\text{eff}}, 0.1)] D_k^{\text{Orig}}.\quad (28)$$

Some important characteristics of the transition model formulation are outlined below:

- The transport equations do not attempt to model the physics of the transition process. Instead, they provide a framework for the implementation of the experimental correlations.
- The model is designed to predict a large increase in $Re_{\theta_1}$ whenever a favorable pressure gradient is present or when turbulence intensity decays. Conversely, $Re_{\theta_1}$ decreases when an adverse pressure gradient is present or when the freestream turbulence intensity increases. This behavior is consistent with experimental observations as reported by Menter and Langtry [72, 82, 83].
• There is a lag between the changes in the freestream value of $\tilde{Re}_\theta$ and that inside the boundary layer. The lag allows the model to compute the onset of transition within the sub-viscous region of the boundary layer ($y^+ < 5$) by focusing on the past history of pressure gradient and turbulence intensity and not on the local value of $\tilde{Re}_\theta$ in the freestream and in the outer-region (log-law region) of the boundary layer. This implies that there is turbulence developing in the main flow before transition is triggered within the sub-viscous region of the boundary layer. Therefore, the behavior of $\tilde{Re}_\theta$ in the freestream has to be followed to determine the point where turbulence begins to form substantially and affects the convective heat transfer.

• In cases of laminar separation bubbles, the intermittency variable is allowed to exceed 1.0 in order to increase the production of turbulent kinetic energy and enhance the prediction of the reattachment location.
Simulations with a partial-span model (PSM) were conducted to evaluate the effects of external flow transition on the LE skin temperature distribution for the warm hold condition corresponding to experimental Run 42. The simulations were conducted with the pressure-based segregated solver, first-order spatial discretization, and the SST $\kappa - \omega$ turbulence model.

Three cases were evaluated:

- Fully turbulent: simulation without laminar zones in the external cold airflow around the wing leading edge. The SST $\kappa - \omega$ turbulence model is solved in the entire flowfield.

- Transition ($x/c=2.97\%$): simulation with laminar zones around the LE extending on the upper surface until $x/c=2.97\%$ according to the results from the transition modeling study in Chapter 3. The SST $\kappa - \omega$ turbulence model scalars are set to zero in this region.

- Transition ($x/c=7.40\%$): simulation with laminar zones around the LE extending on the upper surface until $x/c=7.40\%$, a location predicted by XFOIL with the $e^N$ method ($N=9$) under the assumption of low-freestream turbulence intensity.

The boundary conditions at the piccolo holes as well as other boundary surfaces of the PSM were defined as follows for Run 42:

- Piccolo Holes: Pressure inlet with $P_{Total} = 14.85$ psig and $T_{Total} = 365.86^\circ$F.

- Hot air outlets: Pressure outlet with a gauge static pressure of 0.47 psig at the upper and lower exits of the diffuser passages.

- Inner-liner surface: No-slip wall condition with a chordwise temperature profile based on experimental data.

- Piccolo tube: wall condition with a fixed temperature of 330$^\circ$F.

- Tunnel inlet: pressure inlet with a total pressure of 0.375 psig and a total temperature of 22.98$^\circ$F as recorded in the experiment.

- Tunnel outlet: pressure outlet with a zero gauge static pressure. The resulting tunnel speed from the pressure difference between the tunnel inlet and outlet was approximately 115 kts.
• Symmetry conditions at both span ends of the 4.88-in segment.
• No-slip adiabatic walls at the interior surface of the wing skin downstream of the Teflon® inserts.
• No-slip adiabatic walls at the two side walls of the tunnel.
• Wall conditions along portion of the wing LE skin that extended from the inner-liner exit to the insulating insert with a chordwise temperature profile specified from the experimental data.

Figure 1 shows the computed LE skin temperature for the three cases previously described. The results highlight the importance of implementing laminar zones around the wing leading edge. The temperature distribution for the fully-turbulent case was considerably lower than for the transitional cases due to excessive cooling in the simulation. On the other hand, the temperature distribution for the case with transition at $x/c=7.4\%$ shows insufficient cooling over the upper surface of the wing LE. Finally, the temperature distribution with transition at $x/c=2.97\%$ shows the improvement obtained after shifting the transition location further upstream following the results from the transition modeling study discussed in Chapter 3.
This appendix is a succinct description of the instrumentation in the icing tunnel model tested at the NASA Glenn Icing Research Center Tunnel (IRT) by WSU and that was reproduced computationally in the current numerical investigation. The description presented here was originally published in reports co-authored by the present author [19, 84] as well as other publications published by the WSU icing Research Group [19, 20, 84].

The wing model was equipped with 76 surface pressure taps distributed chordwise at the mid-span station. Pressure distributions were obtained with the pressure leading edge prior to the start of the icing tests. Figure 1 shows an isometric view of the internal structure and Figs. 2 and 3 show the end of the piccolo tube was a threaded section that was used to adjust the piccolo.

Leading edge skin temperatures were measured at four spanwise stations shown in Fig. 4. The spanwise length of the thermal wing leading edge (Fig. 5) was 71.75 inches (1.823 m) from the tunnel floor to the ceiling. Leading edge skin temperatures and bleed air system thermal and pressure data were obtained with T-type thermocouples, resistance temperature detectors, heat flux gages, and pressure transducers.

Station 1 was located at 27.725 inches (0.704 m) above the IRT test section floor and contained 12 Micro-Foil® heat flux gages attached to the interior of the leading edge skin as shown in Fig. 6. Each heat flux sensor was equipped with a thermocouple that was used to monitor interior surface temperatures and correct the heat flux measurements. The heat flux sensors were 0.003 inches (0.08 mm) thick, 0.276 inches (7 mm) wide and 0.709 inches (18 mm) long, and quantifies heat transfer (loss or gain) through the surface where it is mounted. The sensor measures the temperature difference between opposite sides of a thin layer of separator material to obtain a direct measurement of the heat loss or gain through the separator material.

Station 2 (also referred to as station B) at 30.175 inches (0.766 m) and station 3 (also referred to as station A) at 42.425 inches (1.077 m) above the tunnel floor were instrumented with 32 T-type thermocouples embedded approximately halfway between the inner and outer leading edge skin surfaces as shown in Fig. 7.
Station 4 at 52.225 inches (1.327 m) above the tunnel floor was instrumented with 15 four-wire platinum RTD’s embedded inside the leading edge skin (Fig. 8). The RTD’s were 0.045 inches (1.14 mm) thick, 0.25 inches (6.35 mm) wide and 0.5 inches (12.7 mm) long.

Figures 9 and 10 show the instrumentation installed in the diffuser and the passages between the inner-liner and leading edge skins. Pressure taps installed on the inner-liner skin were used to monitor pressures in the diffuser and inner-liner passages and T-type thermocouples were used to measure hot air temperature.

Another set of T-type thermocouples and four heat flux gages were installed on the back of the inner-liner skin. These thermocouples were used to quantify the heat loss through the inner-liner skin. Inner-liner and diffuser pressure taps and thermocouples were placed at two spanwise stations corresponding to stations A and B in Fig. 4. Heat fluxes were also attached at the back of the inner-liner skin at station B.

In addition to the instrumentation described above, thermocouples and total pressure ports were installed inside the piccolo at four spanwise stations 6.92 inches, 30.18 inches, 42.43 inches and 65.69 inches with respect to the IRT floor. Four thermocouples were installed in the hot air return duct formed by the front spar and the inner-liner skin to monitor the temperature of the spanwise flow in the return duct. Two thermocouples and two pressure taps (a total and a static) were installed at the exhaust manifold to monitor exhaust flow properties. A flow meter, two thermocouples, and a Pitot-Static probe were used to monitor bleed air mass flow properties upstream of the piccolo inlet as shown in Fig. 11. The mass flow measurement system was calibrated in a bleed air laboratory prior to the IRT tests. An absolute pressure transducer was installed below the tunnel floor to monitor ambient pressure for the duration of the icing tests.
Figure 1. Details of Wing model interior structure [84].

Figure 2. CAD model of piccolo threaded end [84].

Figure 3. Piccolo threaded end in real model [84].
Figure 4. Spanwise locations of leading edge skin thermal instrumentation [84].
All dimensions in inches with respect of NASA IRT test section floor.
Figure 5. Wing thermal leading edge assembly [84].
Figure 6. Chordwise distribution of 12 heat flux sensors attached to the skin interior surface [84].

Figure 7. Installation of 32 T-type thermocouples inside the wing leading edge skin [84].

Figure 8. Chordwise distribution of 15 RTD’s embedded inside leading edge skin [84].

Figure 9. T-type thermocouples installed in diffuser and diffuser passages at each spanwise station [84].
Figure 10. Twelve pressure taps installed on inner-liner skin at each spanwise station [84].
(Two stations A & B; 24 Taps Total)

Figure 11. Bleed air supply line and instrumentation [84].