Computational Modeling of Total Temperature Probes

Alex J. Schneider

Thesis submitted to the Faculty of the Virginia Polytechnic Institute and State University in partial fulfillment of the requirements for the degree of

Master of Science
in
Aerospace Engineering

Joseph A. Schetz, Co-Chair
K. Todd Lowe, Co-Chair
Christopher J. Roy

January 30, 2015
Blacksburg, Virginia

Keywords: Computational Fluid Dynamics, Heat Transfer, High Temperature Instrumentation
Copyright 2015, Alex J. Schneider
Computational Modeling of Total Temperature Probes

Alex J. Schneider

ABSTRACT

A study is presented to explore the suitability of CFD as a tool in the design and analysis of total temperature probes. Simulations were completed using 2D axisymmetric and 3D geometry of stagnation total temperature probes using ANSYS Fluent. The geometric effects explored include comparisons of shielded and unshielded probes, the effect of leading edge curvature on near-field flow, and the influence of freestream Mach number and pressure on probe performance. Data were compared to experimental results from the literature, with freestream conditions of $M = 0.3 - 0.9$, $p_t = 0.2 - 1$ atm, $T_t = 300 - 1111.1$ K.

It is shown that 2D axisymmetric geometry is ill-suited for analyses of unshielded probes with bare-wire thermocouples due to their dependence upon accurate geometric characterization of bare-wire thermocouples. It is also shown that shielded probes face additional challenges when modeled using 2D axisymmetric geometry, including vent area sizing inconsistencies.

Analysis of shielded probes using both 2D axisymmetric and 3D geometry were able to produce aerodynamic recovery correction values similar to the experimental results from the literature. 2D axisymmetric geometry is shown to be sensitive to changes in freestream Mach number and pressure based upon the sizing of vent geometry, described in this report. Aerodynamic recovery correction values generated by 3D geometry do not show this sensitivity and very nearly match the results from the literature.

A second study was completed of a cooled, shielded total temperature probe which was designed, manufactured, and tested at Virginia Tech to characterize conduction error. The probe was designed utilizing conventional total temperature design guidelines and modified with feedback from CFD analysis. This test case was used to validate the role of CFD in the design of total temperature probes and the fidelity of the solutions generated when compared to experimental results. A high level of agreement between CFD predictions and experimental results is shown, while simplified, low-order model results under predicted probe recovery.
Acknowledgments

I would like to extend a deep debt of gratitude to my co-advisors Dr. Joseph Schetz and Dr. Todd Lowe. After completing my undergrad degree in unspectacular fashion, they were willing to take a chance on me and allow me to work with them on what I consider to be a very rewarding project. Through constant feedback and high expectations, they have helped me to grow as an engineer and become accustomed to thinking on my feet in stressful situations. They have helped me grow both professionally and personally, and I look up to both of them as my mentors. Additionally, I would like to thank my final committee member Dr. Chris Roy for always being able to help me solve difficult Fluent issues when they (often) arose and doing so in a very friendly manner.

My family has been my support group through this whole process, often providing pep talks or words of encouragement any time they were needed. They also inspired my original drive to excel in education and pursue engineering. I can not ever repay them in enough ways for this.

My friends have also been a necessity to my mental health through my college career. Often it was something as simple as getting together and having a therapeutic group meal that has helped me reach this point in my education. Its been a pleasure to work with every one of them and I don’t believe I would have made it through this without any of the Mangi and their ability to keep me grounded.

I would like to thank our sponsor Pratt & Whitney for allowing us to pursue this project (Center of Excellence Task #84) and to develop my skills as an application-oriented engineer. I would specifically like to thank Dr. Charles Haldeman, Andrew Consiglio, and Justin Urban for taking the time to speak with us each and every week. It was this constant sponsor feedback which made this project so interesting and skill-building.
Contents

List of Figures vi

List of Tables ix

1 Introduction 1

1.1 Historical Background and Related Works 1

1.2 Description of Low-Order Modeling 4

1.2.1 Moffat Models 4

1.2.2 Thermal Resistance Model 6

1.2.3 Cell-based Numerical Model 7

1.3 Motivation and Plan of the Current Research 8

1.4 Selection of Glawe, Simmons, and Stickney Study Case 9

1.4.1 Glawe, Simmons, and Stickney Background and Probe Description 10

1.4.2 Glawe, Simmons, and Stickney Experimental Results 12

2 Computational Models and Methods 16

2.1 Governing Equations 16

2.1.1 Generalized 16

2.1.2 ANSYS Fluent Physics Model 17

2.2 Simulated Geometry and Flow Domain 18

2.2.1 Geometry Representation 18

2.2.2 2D Structured Meshing 21

2.2.3 3D Unstructured Meshing 26

2.3 Simulation Setup 30

3 Results 32
3.1 Mean Flow Features ............................................. 32
3.2 2D Axisymmetric Vent Size Determination ...................... 35
3.3 Effects of Leading Edge Curvature ................................. 40
3.4 Conjugate Heat Transfer ........................................ 40
3.5 Unshielded Probe Results ........................................ 44
3.6 Comparisons with Experiment .................................... 46
3.7 Effects of Freestream Mach Number and Pressure .............. 48

4 VT Validation Probe ............................................. 51
  4.1 Probe Design .................................................. 51
  4.2 Computational Models ......................................... 53
  4.3 Results ...................................................... 57

5 Conclusions ..................................................... 64
  5.1 Future Work .................................................. 67

Bibliography ..................................................... 68
List of Figures

1.1 Cross section of an early total temperature probe geometry specified by Franz. 1
1.2 Cross section of a total temperature probe designed for supersonic regimes by Goldstein et al. at NACA. 3
1.3 Cross section of straight-tube diffuser total temperature probe designed by Glawe et al. 3
1.4 Simple diagram of thermal resistance modeling. 7
1.5 Drawings of the total temperature probe geometries specified by Glawe et al. 11
1.6 Test setup used by Glawe et al. 12
1.7 Pressure-normalized aerodynamic recovery correction, $\Delta_0$, from Glawe et al. vs. Mach number. 14
1.8 Pressure-normalized aerodynamic recovery correction, $\Delta_0$, from Glawe et al. vs. Reynolds number. 14
1.9 Aerodynamic recovery correction ratio $\frac{\Delta}{\Delta_0}$ vs. Freestream total pressure from Glawe et al. 15
2.1 3D representation of probe geometry specified by Glawe et al. 19
2.2 2D axisymmetric representation of shielded probe geometry specified by Glawe et al. 20
2.3 2D axisymmetric representation of unshielded probe geometry specified by Glawe et al. 20
2.4 Leading edge modifications used with the 2D axisymmetric model of Glawe et al. 21
2.5 Example of a 2D axisymmetric structured mesh generated using the Glawe et al. geometry. 22
2.6 2D axisymmetric values of first-cell $y^+$ for near-wall boundary layer treatment. 23
2.7 Velocity distribution over probe using coarse 2D axisymmetric mesh. 25
2.8 Comparison of 2D axisymmetric mesh to 3D mesh. 27
2.9 An example of a 3D unstructured mesh generated using the Glawe et al. geometry (3D Mesh 1). 29
<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.10</td>
<td>3D values of first-cell $y^+$ for near-wall boundary layer treatment.</td>
</tr>
<tr>
<td>2.11</td>
<td>The Fluent boundary conditions applied to the 2D axisymmetric model.</td>
</tr>
<tr>
<td>3.1</td>
<td>Influence of the vent hole in downstream velocity and pressure distribution for 3D model.</td>
</tr>
<tr>
<td>3.2</td>
<td>3D streamlines of velocity characterizing the flow over and through the probe.</td>
</tr>
<tr>
<td>3.3</td>
<td>Velocity vectors demonstrating vortices generated by vent flow.</td>
</tr>
<tr>
<td>3.4</td>
<td>Comparison of velocity contours in planes through the vent for 2D axisymmetric and 3D geometry for equivalent mass flow rate cases (Iteration 3).</td>
</tr>
<tr>
<td>3.5</td>
<td>2D axisymmetric streamlines of velocity characterizing the flow over and through the probe.</td>
</tr>
<tr>
<td>3.6</td>
<td>Regions of interest in predicted flow contours for the 2D axisymmetric model of the shielded probe with $A_{ratio} = 1.293$.</td>
</tr>
<tr>
<td>3.7</td>
<td>Predicted 2D axisymmetric separation region profiles for all leading edge geometries.</td>
</tr>
<tr>
<td>3.8</td>
<td>Predicted temperature distribution of the 2D axisymmetric model at conditions $M = 0.6, p_t = 1$ atm for different total temperatures.</td>
</tr>
<tr>
<td>3.9</td>
<td>Contours of heat flux from solid to fluid on the thermocouple.</td>
</tr>
<tr>
<td>3.10</td>
<td>2D axisymmetric streamlines of velocity characterizing the flow over the unshielded probe.</td>
</tr>
<tr>
<td>3.11</td>
<td>Regions of interest in predicted flow contours for the 2D axisymmetric model of the unshielded probe.</td>
</tr>
<tr>
<td>3.12</td>
<td>Contours of heat flux from solid to fluid on the thermocouple of the unshielded probe.</td>
</tr>
<tr>
<td>3.13</td>
<td>The pressure-normalized aerodynamic recovery correction $\Delta_0$ for unshielded and shielded probes over a range of Mach numbers.</td>
</tr>
<tr>
<td>3.14</td>
<td>The pressure-normalized aerodynamic recovery correction $\Delta_0$ for unshielded and shielded probes over a range of Reynolds values based upon thermocouple wire diameter.</td>
</tr>
<tr>
<td>3.15</td>
<td>The aerodynamic recovery correction, $\Delta_0$ variation over a range of Mach numbers for a shielded probe.</td>
</tr>
<tr>
<td>3.16</td>
<td>The aerodynamic recovery correction, $\Delta_0$ variation over a range of Reynolds values based upon thermocouple wire diameter for a shielded probe.</td>
</tr>
<tr>
<td>Section</td>
<td>Description</td>
</tr>
<tr>
<td>---------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>4.1</td>
<td>Manufacturing drawing of probe designed by Englerth.</td>
</tr>
<tr>
<td>4.2</td>
<td>Computational model of validation case.</td>
</tr>
<tr>
<td>4.3</td>
<td>Computational mesh used for validation case.</td>
</tr>
<tr>
<td>4.4</td>
<td>Applied boundary conditions of validation case.</td>
</tr>
<tr>
<td>4.5</td>
<td>Velocity contours of the flow field of the validation probe.</td>
</tr>
<tr>
<td>4.6</td>
<td>Temperature contours of the flow field of the validation probe.</td>
</tr>
<tr>
<td>4.7</td>
<td>Temperature distribution within the solids of the validation probe.</td>
</tr>
<tr>
<td>4.8</td>
<td>Heat flux from the solids over the thermocouple of the validation case.</td>
</tr>
<tr>
<td>4.9</td>
<td>Combined corrected recovery results of experimental work, TRM, and CFD/CHT of the validation case of Englerth.</td>
</tr>
</tbody>
</table>
## List of Tables

1.1 Geometric parameters of the probe studied by Glawe *et al.* .......................... 12

2.1 2D Axisymmetric Mesh Element Counts .................................................. 23
2.2 2D Grid Independence Study Grid Sizes .................................................. 24
2.3 2D Grid Independence Study Results ....................................................... 25
2.4 3D Grid Independence Study Grid Sizes .................................................. 27
2.5 3D Grid Independence Study Results ....................................................... 28
2.6 3D Mesh Element Counts ............................................................................ 28
2.7 Fluent Simulation Inputs .............................................................................. 30

3.1 Comparison of 2D axisymmetric and 3D vent areas and associated mass flow rates ......................................................................................................................... 38

4.1 Geometric parameters of the probe designed by Englerth ............................ 52
4.2 Validation Case Mesh Element Counts ....................................................... 56
Nomenclature

\( A_C \)  
Surface area available for convection

\( A_R \)  
Surface area available for radiation

\( A_{\text{vent}} \)  
Total vent area of the model

\( A_{\text{inlet}} \)  
Total inlet area of the model

\( A_{\text{slot, 2D axisymmetric}} \)  
Vent area of slot on 2D axisymmetric model

\( A_{\text{holes, 3D}} \)  
Vent area of combined holes on 3D model

\( C_p \)  
Specific heat capacity of material

\( D_{\text{wire}} \)  
Diameter of thermocouple wire

\( E_V \)  
Velocity error

\( E_C \)  
Conduction error

\( E_R \)  
Radiation error

\( E_T \)  
Transient error

\( e \)  
Internal energy

\( e_a \)  
Relative error (\%)

\( \vec{f} \)  
External force vector applied to the fluid element

\( f_x, f_y, f_z \)  
Force components with respect to \( x, y, z \)

\( GCI \)  
Grid Convergence Index (Roache)

\( h \)  
Grid spacing

\( h_C \)  
Convective heat transfer coefficient

\( K_R \)  
Radiation view factor

\( k \)  
Thermal conductivity of material

\( k_s \)  
Thermal conductivity of solid material

\( L_{\text{immersion}} \)  
Length of exposed thermocouple wire and wire

\( M \)  
Mach number

\( m_{\text{inlet, 2D axisymmetric}} \)  
Inlet mass flow rate on 2D axisymmetric model

\( m_{\text{inlet, 3D}} \)  
Inlet mass flow rate on 3D model

\( \hat{p} \)  
Observed order of accuracy

\( p_s \)  
Flow static pressure

\( p_t \)  
Flow total (stagnation) pressure

\( R \)  
Overall recovery

\( R_{\text{corrected}} \)  
Mach-corrected overall recovery

\( R_e \)  
Reynolds number

\( r \)  
Grid refinement factor

\( T_{04} \)  
Turbine inlet total (stagnation) temperature

\( T_{05} \)  
Turbine exit total (stagnation) temperature

\( T_b \)  
Probe base temperature

\( T_j \)  
Indicated junction temperature
$T_s$  Flow static temperature

$T_t$  Flow total (stagnation) temperature

$T_w$  Temperature of duct surrounding probe

$u, v, w$  Velocity components with respect to x, y, z

$\vec{V}$  Velocity vector of the fluid element

$y^+$  Near-wall mesh height parameter

**Greek**

$\alpha$  Recovery factor

$\Delta$  Aerodynamic recovery correction

$\Delta_0$  Pressure-normalized aerodynamic recovery correction

$\Delta A_i$  2D mesh element area

$\Delta V_i$  3D mesh element volume

$\epsilon$  Material emissivity

$\varepsilon$  Absolute error

$\Theta$  Conduction driver

$\mu$  Fluid dynamic viscosity

$\nu$  Fluid kinematic viscosity

$\rho$  Fluid static density

$\rho_t$  Fluid total (stagnation) density

$\sigma$  Stefan-Boltzmann constant

$\tau_{1,2}$  Shear stress applied to the (1) face in the (2) direction (e.g. $\tau_{xy}, \tau_{yx} \cdots$)

$\tau_w$  Wall shear
## Technical Glossary

<table>
<thead>
<tr>
<th>Term</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Diffuser Shield</strong></td>
<td>Tube placed around exposed thermocouple bead to stagnate flow. Often referred to as the ‘shield’.</td>
</tr>
<tr>
<td><strong>Stagnation Tube</strong></td>
<td>Alternate name for diffuser shield.</td>
</tr>
<tr>
<td><strong>Potting</strong></td>
<td>Thermally-insulative material which encases thermocouple wires.</td>
</tr>
<tr>
<td><strong>Sheath</strong></td>
<td>Tube surrounding thermocouple wire and potting; used to thermally insulate and protect thermocouple wire</td>
</tr>
<tr>
<td><strong>Inlet</strong></td>
<td>The upstream portion of the diffuser shield located where freestream flow begins to enter the shield.</td>
</tr>
<tr>
<td><strong>Vent</strong></td>
<td>The downstream location where flow is designed to exit the diffuser shield.</td>
</tr>
<tr>
<td><strong>Vent holes</strong></td>
<td>Vent feature for 3D geometry. Holes drilled circumferentially in diffuser shield.</td>
</tr>
<tr>
<td><strong>Vent slot</strong></td>
<td>Vent feature for 2D axisymmetric geometry. Results from constant-width cut being revolved around probe axis.</td>
</tr>
<tr>
<td><strong>Recovery factor</strong></td>
<td>A nondimensional metric used to characterize the performance of total temperature probes, relating the indicated junction temperature to the true flow total temperature.</td>
</tr>
</tbody>
</table>
1 Introduction

1.1 Historical Background and Related Works

Accurate reporting of total temperature within turbine engines is vital for determining the efficiency at which the engine is operating. The amount of power delivered by the turbine to the upstream components in the engine can be determined using the difference in total temperature values before ($T_{04}$) and after ($T_{05}$) the turbine [1]. If these values are reported in real-time during engine operation, it is possible to control the engine more closely based upon knowledge of overall engine efficiency.

In an attempt to increase overall engine efficiency and performance, modern jet engines operate at much higher maximum temperatures than those which were characterized in early reports. These new operating regimes are well beyond the melting point of the materials used in these components, requiring significant amounts of cooling in all components. Due to this, the life of engine components can be highly limited in cases of overheating, requiring more precise temperature characterization and control [2]. This has driven the need for analysis of total temperature probes using new high-temperature materials.

The use of thermometry in turbomachinery began with the advent of the supercharger and the need to accurately measure internal flow temperatures to determine the efficiency of the system. Due to the high velocities experienced within these devices, measuring static temperature ($T_s$) proved ineffective due to the effects of the rapid velocity loss near the temperature-sensing element of early probes. It was determined by designers that the total temperature ($T_t$) of the flow would be a more repeatable measurement in such a complex flow environment [3]. This led to the addition of diffusers around temperature-sensing elements to predictably stagnate the flow, providing more repeatable $T_t$ measurements. One of the first pioneers in this field was A. Franz [4], who designed some of the first vented diffuser-style probes, later termed 'Franz Probes', shown in Figure 1.1.

![Figure 1.1: Cross section of an early total temperature probe geometry specified by Franz.](image)

These new probes tackled many of the challenges faced by non-diffuser designs, providing
increased thermocouple durability due to decreased aerodynamic loading and relatively low radiation heat transfer losses when compared to unshielded designs. As flow enters the probe, there is a loss of heat due to conduction and radiation within the shielding [5]. To minimize this heat loss within the probe, vents were present in the probe to allow flow to exit the probe, bringing new, hot flow into the probe. This provided the benefit of better temperature response times, a minimal loss of accuracy, and increased repeatability when compared to a non-vented designs [4] [6]. This was achieved by finding a balance between entrance area to exit area of the probe which allowed new, hot flow to enter and nearly stagnate, while removing cooled flow from the probe. One drawback of this design was its relative yaw and pitch sensitivity, likely due to the tailored internal contour of the diffusers, allowing only minimal yaw disturbances before internal boundary layer separation occurred, as noted by Hottel et al. [3]. Another drawback of the design was the complex manufacturing needed for the tailored internal contour of the diffuser. One example study completed using a Franz-style probe was that of Lindsey [7], which provided calibration curves of Franz-style probes for use in superchargers for Pratt & Whitney engines.

The performance of these early probes was often characterized by the recovery factor, $\alpha$ defined as

$$\alpha = \frac{T_j - T_s}{T_t - T_s}$$

(1.1)

where $T_j$ is the indicated thermocouple junction temperature, $T_t$ is the total (stagnation) temperature of the flow, and $T_s$ is the static temperature of the flow. The ideal value of recovery factor is 1, indicating no aerodynamic losses. Values of recovery factor for early Franz-style probes, as demonstrated by Lindsey [7], range from $\alpha = .85 - .99$ for probes of varying sizes over their operational regime.

These diffuser-style probes were further refined due to the increasing need of superchargers for piston-driven aircraft engines and the advent of the turbine engine. This led to the extensive study and testing of these probes in the following decades. Initially, these new probes were given complex internal features and angular external profiles. Many of these probes were designed for high velocity regimes that did not directly impact temperature probes for use in turbine applications, but furthered the analysis and design procedures for these probes. Examples of probes designed during this period can be found in Winkler [8], Gordon [9], and Goldstein et al. [5], shown in Figure 1.2.

After this period, most designs were often simplified to straight tube diffusers, due to their ease of manufacture and relatively small change in instrument accuracy [3]. Some studies of straight-tube diffusers were completed by Pratt & Whitney [3], Bontrager [10], Willbanks [11], Glawe et al. [12] (shown in Figure 1.3) with many others contributing. This thesis will focus upon the work of Glawe et al. [12].

Many of the reports during this period often considered a single aspect of design characterization of these probes. The main focus of Glawe et al. [12] was to characterize the aerodynamic and radiation corrections and time constants associated with probes using
bare-wire thermocouple sensors of varying design. Bontrager [10] focuses on probe size and vent ratio and their effect on recovery, as well as characterizing radiation error within these probes and methods to mitigate these effects. Willbanks [11] performed a standardized recovery characterization over a variety of Mach number and pressure ranges with emphasis on low Reynolds flow.

The error exhibited by total temperature probes can be characterized by three primary error sources: velocity error, conduction error, and radiation error. Other sources of error exist, such as transient errors, but they may be considered negligible or are not applicable to steady state analysis [13].

Velocity error is the name given to the error due to the flow not being fully stagnated at the thermocouple junction. If the velocity over the thermocouple is near zero, the flow around the thermocouple loses heat to the surrounding probe walls due to conduction and radiation, which causes the thermocouple to report a lower temperature than the stagnation temperature. To counteract this effect, the flow within the probe is given a non-zero velocity so that the convective heat transfer into the thermocouple dominates the conduction and radiation away from the thermocouple [14]. This necessary increase in flow velocity is what
leads to the velocity error in the recovery. This need to balance the modes of heat transfer is an important design parameter for total temperature probes and has been studied in various reports [5] [10].

Conduction error is defined as the error due to conduction along the thermocouple to its solid surroundings. This is driven by a difference in temperature between the thermocouple and the base to which it is mounted in [15]. The conduction down the thermocouple wires can be affected by increasing the immersion depth of the thermocouple, increasing the distance from the bead to the mount, as well as decreasing the thermocouple wire diameter [13].

Radiation error is defined as the error due to radiative heat transfer either into or out of the thermocouple. This is caused by a difference in temperature between surfaces which 'view' each other. The magnitude of the radiative heat transfer is based upon fourth-power temperature differences \( (T_A^4 - T_B^4) \) and is therefore very sensitive to the environment surrounding the thermocouple [14]. Unshielded probes are especially susceptible to this form of error due to their direct view of the surrounding environment. The radiation error of a probe can be decreased by adding one or more radiation shields around the probe in an attempt to have a large surface near the thermocouple bead with a small temperature difference from the bead [16]. Additionally the emissivity of the surfaces surrounding the can be minimized to reduce radiative heat transfer, but this is considered impractical for environments associated with turbomachinery.

After the extensive experimental studies that were performed in these works, much of the data were used to generate empirical models. Much of the design work after this period utilized these empirical models rather than new experimental design validation of probe performance. Reports, such as that of Moffat [13], were created with this empirical-based design in mind.

1.2 Description of Low-Order Modeling

1.2.1 Moffat Models

Moffat [13] characterizes the error seen in total temperature probes through a series of equations based upon the physical phenomena associated with the error. The velocity error, \( E_V \) is written as

\[
E_V = T_i - T_j = (1 - \alpha) \frac{\gamma - 1}{\gamma + \frac{1}{2} M^2} \frac{1}{M^2} T_i
\]

with strong dependence on the probe recovery factor, \( \alpha \), as well as the flow Mach number. The derivation of this equation utilizes the isentropic flow relations associated with temperature variation with Mach number. The error due to velocity can be minimized by lowering the flow speed over the thermocouple within the probe. This error type will be
the primary focus of this thesis.

Additionally, Moffat compiled results from other studies to provide a numerical estimation of the recovery factor associated with bare-wire thermocouples. The results of this compilation are

\[
\begin{align*}
\text{wires normal to flow : } & \alpha = 0.68 \pm 0.07 \\
\text{wires parallel to flow : } & \alpha = 0.86 \pm 0.09
\end{align*}
\]

which allows a probe designer to make quick preliminary calculation of probe performance for a particular design. Another design parameter given by Moffat for probes with a diffuser shield is an internal velocity to external velocity ratio of 1/8. This can be varied with the ratio of inlet area to vent area, with the value quoted by Moffat being a typical value. Combining these simplified models, it is possible to characterize the velocity error for both unshielded and shielded total temperature probes.

The conduction error is defined by Moffat to be

\[
E_C = T_t - T_j = \frac{T_t - T_b}{\cosh\left( L\sqrt{\frac{A}{D_{\text{wire}}}}\frac{h_C}{k_s}\right)} \tag{1.3}
\]

where the primary factors contributing to conduction are the driving temperature \((T_t - T_b)\), where \(T_b\) is the base temperature of the probe, thermocouple immersion \((\frac{L}{D_{\text{wire}}})\), and convective heat transfer coefficient \((h_C)\). This equation is derived using the 1-D heat conduction equation for a thin fin. This geometric simplification is logical because the shield and thermocouple have large overall lengths compared to their respective thickness and diameter. The high dependence upon driver temperature \((T_t - T_b)\) is sensible because conduction would be minimized if the mount temperature were very similar to the total (and therefore junction) temperature. For a probe designer, the simplest ways to reduce the error due to conduction are by altering the thermocouple \(\frac{L}{D_{\text{wire}}}\), using a mount which is near the flow temperature, or changing the internal flow velocity over the probe, which increases the velocity error [16]. Additionally, the conduction error may be reduced by minimizing the diameter of the thermocouple wire used \((D_{\text{wire}})\) or lowering its thermal conductivity \((k_s)\) by use of a different material.

Moffat characterizes the radiation error as

\[
E_R = T_t - T_j = \frac{K_R\sigma \epsilon A_R \left(T_j^4 - T_w^4\right)}{h_C A_C} \tag{1.4}
\]

with the primary drivers being the material emissivity \((\sigma\epsilon)\), the driving wall temperature \((T_j^4 - T_w^4)\), convective cooling coefficient \((h_C)\), and geometric parameters such as the view factor \((K_R)\) and associated areas \((A_R, A_C)\). The simplest way for the designer to minimize the radiation error is by decreasing the material emissivity, minimizing the driving wall
temperature by placing the probe in an environment of similar temperature, or increasing the internal flow velocity over the probe which, as stated above, increases the velocity error [16]. The transient error of a total temperature probe is the error due to the time response of the probe to changing conditions, defined as

\[ E_T = T_i - T_j = \tau \frac{dT_j}{dt} \]

where \[ \tau = \frac{\rho C_p D_{\text{wire}}}{4h_C} \] (1.5)

The transient error is only important for probes which are expected to provide time-accurate results. Experiments or simulations which are driven to steady-state conditions before data is taken will have a negligible transient error. The transient error is formulated to account for the heat storage rate of the material in the probe and the speed at which heat in the system is added or dissipated. This is done through the use of convective heat transfer \((h_C)\) and material properties such as the specific heat capacity \((C_p)\), density \((\rho)\), and wire diameter \((D_{\text{wire}})\). This can be minimized with material selection, increasing the velocity over the thermocouple, or by allowing the probe to come to a steady-state solution before taking data.

### 1.2.2 Thermal Resistance Model

To model the conduction error within total temperature probes, a simplified method of thermal resistance modeling can be employed. This method creates a circuit of 'thermal resistors' which act in either series or parallel depending upon the physical arrangement of conductive materials within the probe. Nodes between 'resistors' are placed at material interfaces, and solid materials are treated with a thermal resistance based upon the material conductivity and the length of the material it is applied to, as shown in Figure 1.4. As heat flows through the material in an attempt to establish thermal equilibrium, the resistances of the materials interacts with the other resistors, which allows thermal equilibrium to be calculated. Boundary conditions are placed in the model using either surface temperatures or convective heat transfer coefficients placed on the material surface. A system of equations can be built using these resistances and arranged such that they model the physical geometry of the conductive media in the probe solids. Further information about this model and its application to total temperature probes can be found in Englerth [17].
This model is useful for characterizing the conduction error experienced within total temperature probes, as it can model complex thermal circuits such as those seen in sheathed thermocouples. These thermocouples must be modeled as parallel resistors to capture the streamwise conduction through the thermocouple required to characterize the conduction error within the probe. The physical arrangement of total temperature probes is well suited to thermal resistance modeling due to the use of multiple materials with solid-solid contact points often used in these probes. These thermal barriers are easily modeled using the circuit structure employed in this type of modeling. Weaknesses of this model are that external boundary conditions are required which must be known before the model can be executed. This does not allow the interaction of fluid and solid heat transfer, which would be necessary in an accurate thermal equilibrium calculation based upon fluid convection. Additionally, this model is useful when the driving temperature of the probe is high, allowing for maximum heat conduction through the model, but when the driving temperature is small, conduction plays a minimal role in the error of the probe, rendering this model much less useful.

Some results utilizing this model and their comparison to CFD and experimental results will be presented in this report. The primary test cases for this report (Glawe et al. [12]) contains minimal conduction error and is poorly suited for use with this model. The secondary test case (VT validation probe [17]) was analyzed using the thermal resistance model due to its focus on conduction error and high temperature driver.

### 1.2.3 Cell-based Numerical Model

Rhodes et al. [18] created a model of a total temperature probe using a cell-based numerical method. This method utilizes a very coarse 2D axisymmetric grid (approximately 30 cells) generated through the interior of the total temperature probe. Each cell in the model is
treated as a finite control volume which obeys the laws of conservation of mass and energy. An internal probe mass flow rate is prescribed based upon the external flow conditions, and the local cell-based flow rate is determined based upon the cell area as to obey conservation of mass. The energy flow rate of each cell in the model is calculated by modeling the conduction, convection, and radiation between the cells. This calculation of energy flow occurs in both solid and fluid cells with the interactions between them being characterized by the fluid flow rate over the surface of the solid element. A system of equations is generated by combining the governing equations for each cell in the model, and the system is solved iteratively to achieve a steady-state solution.

This model has the advantage of characterizing the conduction, convection, and radiation components of the probes, but is limited by the assumption of boundary conditions associated with the flow through the probes and the assumption of thermal boundary conditions associated with the conduction to the probe mount. External flow characteristics are ignored using this model, including the heat transfer to the shielding, radiation to external tunnel walls, and the heat transfer associated with the flow features around the vent holes.

1.3 Motivation and Plan of the Current Research

The need for this study has been driven by the following primary factors:

- The importance of accurate total temperature measurements in the determination of engine performance and durability of modern turbomachines
- The expansion of operating regimes of current total temperature probes
- Utilizing design tools which did not exist when previous studies were performed
- Enhancement of knowledge associated with characterizing the performance of these probes

When many of the reports characterizing the total temperature probes in current use were completed, computational modeling was not yet a viable tool in the design portfolio of engineers. In the succeeding decades, computational modeling has become increasingly viable as computing power increases and physical models continue to increase in complexity, capturing more advanced physics than earlier. Many of the complex physical interactions seen in total temperature probes can now be modeled using features such as compressible flow models, conjugate heat transfer (CHT) between fluids and solids, and radiation models that employ ray tracing applied to specific model geometry and surface treatments. In many areas of engineering, computational modeling now serves as an important tool to characterize performance early in the design process. This use potentially limits the size of experimental test matrices of new products and allows for regimes outside of the experimental
envelope to be studied. Total temperature probe design lends itself well to computational modeling, as experimental testing in extreme conditions is time-consuming and expensive, and many of the complex flow and heat transfer phenomena experienced by these probes are now incorporated into computational models. While full computational modeling is not yet matured enough to replace aerodynamic testing, it can be used to characterize many of the design features in these probes before fabrication and expensive testing occur. CFD/CHT also permits the investigation of detailed flow features found inside of physically small probes being investigated for use in modern engines, aiding in the understanding of complex internal flow and their effects on design.

Many of the studies performed on these probes were completed over 40 years ago, so much of the intrinsic knowledge associated with the design of these probes has not been updated or incremented using modern techniques and tools. This report will attempt to recreate some of the observations noted in these previous reports while also updating processes which utilize modern methods and add insight into the overall design analysis of total temperature probes. The low-order methods currently utilized in the design of total temperature probes often utilize methods based upon empirical results or are limited in the scope of their accuracy based upon knowledge of boundary conditions, as shown above. Development of a computational model which characterizes the underlying physics of flow and heat transfer in these probes can help to expand the usefulness of the low-order models. Complex fluid-solids interactions are not accounted for in the low-order models, including the effects of external flow on the internal heat transfer, which is often not negligible. This study can be used to extend the physics of these models and provide more accurate methods for their use.

1.4 Selection of Glawe, Simmons, and Stickney Study Case

An experimental case was chosen which could be recreated and compared with CFD/CHT as the primary test case. The desired experiment needed to use geometry which contained a straight-tube diffuser, similar to modern probes in use. Additionally, the experiment needed to be well documented with detailed probe construction and experimental conditions tested. These experiments needed to explore regimes relevant to the probes used in jet engines, principally high temperatures at high subsonic Mach numbers.

It was found that documentation on experiments which fit these criteria were often lacking in detail. While multiple reports had well-documented, applicable geometry, they did not choose specific sources of error which to characterize. Conversely, while some reports did explore the desired error characterization, they were poorly documented and could not be reliably recreated for use in CFD/CHT.

The experimental work of Glawe et al. [12] was chosen as it meets all of the criteria above.
One of the probes tested in the study was a shielded straight-tube diffuser style probe with detailed construction geometry included in the report. Experiments were performed over Mach number and pressure ranges to characterize the aerodynamic recovery of the probe. Experiments were performed at high temperatures to characterize the radiation correction needed for the probe.

The characterization of the different error modes was completed independently, with error sources being uncoupled in the test matrix. The velocity error and the radiation error of the probe were characterized in separate experiments, while having negligible conduction and transient error due to design. This allows the user to independently study these phenomena with minimal interaction between them, allowing for each to be characterized individually. This allowed CFD/CHT to be used to its fullest potential, as these physics model can be disabled when not being utilized, allowing for a true 'single error source' characterization.

The level of detail given in the report was sufficient to allow recreation of both the geometry used and the test conditions to be simulated. The sections following are a summary of the experiments performed by Glawe et al.

### 1.4.1 Glawe, Simmons, and Stickney Background and Probe Description

Glawe et al. [12] performed experiments at NACA Lewis in the 1950's using a collection of probes designed to measure the total temperature of a flow. Each probe had unique design features that were to be characterized and compared to probes of other designs. The goal of this study was not only to compare multiple probe designs but also to characterize the aerodynamic recovery and radiation corrections of these probes over their operational range. The probes of interest in this report, designated Probes 5 & 6 and shown in Figure 1.5, are an unshielded, bare-wire thermocouple probe and its vented, shielded counterpart.
Figure 1.5: Drawings of the total temperature probe geometries specified by Glawe et al. [12].

The Glawe et al. unshielded probe (Probe 5) consisted of an Inconel tube sheathed, magnesium oxide-insulated bare-wire chromel-alumel (Type K) thermocouple, which was bent at a 90° angle to ensure that the thermocouple was parallel to the flow being measured. The shielded probe was constructed by using the unshielded probe and augmenting it with an Inconel diffuser shield with vent holes drilled into the shielding, which was welded to the thermocouple sheath. The vent on the shielding consisted of 8 holes drilled symmetrically around the outer diameter of the shield, located as shown in Figure 1.5b, with a vent ratio of $\frac{A_{\text{vent}}}{A_{\text{inlet}}} = 0.364$. The bead immersion for the probe (the $\frac{3}{16}$ in. exposure length indicated in Figures 1.5a and 1.5b) is defined as $\frac{L_{\text{immersion}}}{D_{\text{wire}}} = 5.859$. Additional geometric properties for both probes are shown in Figure 1.5 and Table 1.1. The shielded probe (Probe 6) will be the primary focus of this study, with geometric variations upon the base design being modeled and simulated.

The tests in Glawe et al. were performed in a test section which was located downstream of a 4 in. diameter combustor nozzle, as shown in Figure 1.6. Studies to characterize the aerodynamic recovery of test probes were performed for Mach numbers between 0.3 and 0.9, total pressures between 0.1 and 2 atm, and at total temperature of 300 K. Additional tests were performed to characterize radiation correction and time constants for test probes, but they will not be considered in this study.
Figure 1.6: Test setup used by Glawe et al. [12]. Glawe, G.E., Simmons, F.S., and Stickney, T.M. Radiation and recovery corrections and time constants of several chromel-alumel thermocouple probes in high-temperature, high-velocity gas streams. Report TN-3766 National Advisory Committee for Aeronautics 1956. Used under fair use, 2015.

<table>
<thead>
<tr>
<th>Strut</th>
<th>Outer Diameter</th>
<th>3/16 in.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Tubing Material</td>
<td>Inconel</td>
</tr>
<tr>
<td></td>
<td>Potting Material</td>
<td>Magnesium Oxide</td>
</tr>
<tr>
<td>Shield</td>
<td>OD</td>
<td>1/4 in.</td>
</tr>
<tr>
<td></td>
<td>Material</td>
<td>Inconel</td>
</tr>
<tr>
<td>Vent Holes</td>
<td>Number</td>
<td>8</td>
</tr>
<tr>
<td></td>
<td>Diameter</td>
<td>0.04 in.</td>
</tr>
<tr>
<td>Thermocouple</td>
<td>Type</td>
<td>Bare-wire Type K</td>
</tr>
<tr>
<td></td>
<td>Material</td>
<td>Chromel-Alumel</td>
</tr>
<tr>
<td></td>
<td>Size</td>
<td>20-Gage (0.032 in.)</td>
</tr>
<tr>
<td></td>
<td>Shape</td>
<td>Wedge-shaped bead</td>
</tr>
</tbody>
</table>

Table 1.1: Geometric parameters of the probe studied by Glawe et al.

1.4.2 Glawe, Simmons, and Stickney Experimental Results

In this study, comparisons will be made with the data contained in Glawe et al. [12]. These results are given using the nondimensional aerodynamic recovery correction,

$$\Delta = \frac{T_t - T_j}{T_t} \quad \text{or} \quad \Delta = (1 - \alpha) \left(1 - \frac{T_s}{T_t}\right)$$ (1.6)
where $T_t$ is the gas total temperature, $T_s$ is the gas static temperature, and $T_j$ is the indicated junction temperature. Ideally, the value of $\Delta$ would be 0, resulting in a probe with no aerodynamic losses. $\Delta_0$ is the form of the aerodynamic recovery correction, $\Delta$, which has been normalized with $p_t = 1$ atm. The pressure-normalized aerodynamic recovery correction, $\Delta_0$ in Glawe et al. [12] is recreated in Figure 1.7. This figure shows the median value of $\Delta_0$ reported in Glawe et al. as a line, with uncertainty bands from the report encompassing the experimental data in shaded boxes. Figure 1.8 shows the change in $\Delta_0$ with Reynolds number based upon thermocouple wire diameter using freestream conditions, shown as

$$Re = \frac{\rho_t u D_{\text{wire}}}{\mu} \quad (1.7)$$

where fluid properties are evaluated at the stagnation temperature [13]. With knowledge of the fluid properties at stagnation, and the freestream Mach number, it is possible to convert the data of Glawe et al. to a form utilizing Reynolds number, which is more independent of Mach number effects, as shown in Figure 1.8. It is shown in Figure 1.9 that when $p_t = 1$ atm, $\Delta_0 = 1$. Using a combination of Figures (1.7, 1.8) and 1.9, a $\Delta_0$ value can be found for any probe under test, providing a comparable performance value independent of freestream pressure and geometry.

It should be noted that for both probes, as Mach number increases, $\Delta_0$ also increases. This relationship is more evident for the unshielded probe due to the direct interaction between the freestream flow and the thermocouple without a diffuser shield to stagnate the flow. The converse of this is true for the pressure correction factor, $\frac{\Delta}{\Delta_0}$, with the shielded probe receiving a greater influence from the freestream pressure.
Figure 1.7: Pressure-normalized aerodynamic recovery correction, $\Delta_0$, from Glawe et al. vs. Mach number. Error bands included in Glawe et al. data. $p_t = 1$ atm, $T_t = 300$ K.

Figure 1.8: Pressure-normalized aerodynamic recovery correction, $\Delta_0$, from Glawe et al. vs. Reynolds number. Reynolds number based upon thermocouple wire diameter. Error bands included in Glawe et al. data. $p_t = 1$ atm, $T_t = 300$ K.
Figure 1.9: Aerodynamic recovery correction ratio $\frac{\Delta}{\Delta_0}$ vs. Freestream total pressure from Glawe et al.
2 Computational Models and Methods

2.1 Governing Equations

The use of computational modeling in this study requires the use of mathematical models and assumptions which attempt to capture physical phenomena and represent them in the computational space.

2.1.1 Generalized

This section is a review of information from Anderson [19]. The fundamental equations behind the computational work in the study are modeled using the Navier-Stokes equations, which are primarily used in the study of viscous flows. The Navier-Stokes equations consist of multiple individual equations, each capturing unique characteristics of the flow. Mass conservation of the computational system by satisfying the continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0$$

shown here in conservation form. To model the momentum conservation of the system, the following equations must be satisfied:

$$\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u \vec{V}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho \vec{f}_x$$  \hspace{1cm} (2.2a)

$$\frac{\partial (\rho v)}{\partial t} + \nabla \cdot (\rho v \vec{V}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho \vec{f}_y$$  \hspace{1cm} (2.2b)

$$\frac{\partial (\rho w)}{\partial t} + \nabla \cdot (\rho w \vec{V}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho \vec{f}_z$$  \hspace{1cm} (2.2c)

and are shown in conservation form, with an equation being written for each Cartesian coordinate direction. To model the conservation of energy in the system, the following equation must be satisfied:

$$\frac{\partial}{\partial t} \left[ \rho \left( e + \frac{V^2}{2} \right) \right] + \nabla \cdot \left[ \rho \left( e + \frac{V^2}{2} \right) \vec{V} \right] = \rho \dot{q} + \frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( k \frac{\partial T}{\partial z} \right)$$

$$+ \frac{\partial}{\partial x} \left( \tau_{xx} \right) + \frac{\partial}{\partial y} \left( \tau_{xy} \right) + \frac{\partial}{\partial z} \left( \tau_{xz} \right) + \frac{\partial}{\partial x} \left( \tau_{yx} \right) + \frac{\partial}{\partial y} \left( \tau_{yy} \right) + \frac{\partial}{\partial z} \left( \tau_{yz} \right) + \frac{\partial}{\partial x} \left( \tau_{zx} \right) + \frac{\partial}{\partial y} \left( \tau_{zy} \right) + \frac{\partial}{\partial z} \left( \tau_{zz} \right) + \rho \vec{f} \cdot \vec{V}$$  \hspace{1cm} (2.3)
which includes terms accounting for changes in bulk kinetic energy as well as internal kinetic or thermal energies.

Additional equations are used to model turbulence within the system. Turbulence models are designed to model Reynolds stresses within the flow. Due to the importance of both laminar viscous and Reynolds stresses in these models, there is no generalized form which works for all flow situations for the studies undertaken here. The turbulence model used in this study was the $k - \omega$ SST model [20]. This model was chosen due to its applicability to both the low and high Reynolds flows, both of which are extensively seen in these probes. The $k - \omega$ SST model behaves as a damped model in the near-wall region, capturing boundary layer behavior correctly. Boundary layer characterization is especially important in the design of total temperature probes due its heavy influence on wall heat transfer. The $k - \omega$ SST model is a hybrid model which becomes the $k - \epsilon$ model further away from the wall. This transformation is important as it causes the model to become less dependent on turbulence boundary conditions of the freestream.

To increase the physical accuracy of the above equations, additional equations can be added to this system, allowing it to compute more complex physical phenomena [21]. Many of these equations can model complex interactions such as Structural Fluid-Solid Interaction (FSI), Conjugate Heat Transfer (CHT), Multi-Phase/Reacting Flows, or Electromagnetic Transport. This ability is termed as 'multiphysics', combining multiple physical models and allowing their results to interact with the results of other models. For the purposes of this thesis, Conjugate Heat Transfer (CHT) is the primary multiphysical model studied and will be discussed later in this report.

### 2.1.2 ANSYS Fluent Physics Model

This study utilized the commercial computational fluid dynamics software, ANSYS Fluent®, chosen due to the wide array of flow, heat transfer, and multiphysics models which are included, as well as its availability to designers in the engineering community. The software ANSYS® Workbench™ included geometry import tools, mesh generation tools, simulation tools, and postprocessing tools. Simulations were completed using this package to aid in package interoperability, when possible.

The flow solver employed during this study was the Density-Based Navier-Stokes (DBNS) solver. The density-based solver was chosen over the pressure-based solver due to its ability to model compressible phenomena, which would be seen at the higher Mach number regimes explored in this study. The Second-Order Upwind spatial discretization was used to provide an accurate solution to the flow problem while keeping simulation iteration times manageable.

To implement Conjugate Heat Transfer (CHT) within the model, the geometry and mesh were designed such that fluid and solid elements were adjacent and were able to be modeled as ‘shadow walls’ in Fluent. ‘Shadow wall’ is the terminology given to a non-physical,
infinitesimally thin wall of infinite thermal conductivity located at the boundary between the fluid and solid domains. This imaginary wall is generated to model the heat transfer between the fluid and solid domains in the probe, allowing the solver to process direct heat transfer between elements of dissimilar material types.

2.2 Simulated Geometry and Flow Domain

2.2.1 Geometry Representation

To simulate the probes studied by Glawe et al., the geometry of the experiments needed to be regenerated using modern CAD tools. This process resulted in multiple models of probes for the study of various geometric changes.

A 3D geometry was created for the shielded probe, shown in Figure 2.1. The flowfield shown around the probe was defined as the flow coming from the 4 in. diameter of the nozzle used in the Glawe et al. experiment shown in Figure 1.6, resulting in a 2 in. distance from center to the edge of the flow domain. Flow outside of this direct jet region was ignored due to its small impact on the near-field flow characteristics around the probe as was considered to be part of the ’farfield’ flow. The upstream and downstream distances of the model were created according to guidelines of Silton [22], who found that for slender bodies in subsonic flow, upstream and downstream boundaries may be placed 3-4 body lengths away from the body of interest. In an effort to minimize computational expenses the model was given multiple planes of symmetry resulting in a 45° section of the geometry. This selection of cut size was limited by the number of vent holes (8) located about the circumference of the diffuser shield. One geometric simplification required by the symmetry was that the individual wires of the thermocouple could no longer be modeled, and they were therefore simplified to a homogeneous rod of equal cross-sectional area to the thermocouple wire found in the original Glawe et al. probe. The rounded 90° support used during testing was simplified to a streamwise-oriented straight support to allow symmetry. The finalized CAD model of the 3D geometric representations are shown in Figure 2.1.

A corresponding, simplified 2D axisymmetric model of the shielded probe, shown in Figure 2.2 was created. This type of geometry was chosen due to its low computational expense with relatively high assumed physical accuracy for early design work, which will be discussed later in this study. The goal of this geometry was to accurately model the flow near the thermocouple tip with a simplified representation of the vent flow. Also, geometric changes could be made to the model with relative ease and could be simulated and analyzed with little time cost to the user. This model was heavily influenced by the assumptions needed for axisymmetry. These assumptions included the simplification of the vent holes as an axisymmetric slot around the circumference of the shield, as well as the simplification of the thermocouple to a homogeneous rod of equal cross-sectional area to the Glawe et al.
Figure 2.1: 3D representation of probe geometry specified by Glawe et al.. The model is cut to a 45° section to minimize computational cost.

thermocouple. The initial slot size of the 2D axisymmetric geometry was chosen using equivalent flow areas to the 3D model, described by $A_{\text{slot, 2D axisymmetric}} = A_{\text{holes, 3D}}$. This slot size was modified for later studies, as outlined below. Additionally, the rounded 90° support used during testing was simplified to a streamwise-oriented straight support to allow axisymmetry.

Modifications were made to the 2D axisymmetric model including changing the vent slot size. The results of this modification will be outlined later in this report.

Further geometric modifications of the 2D axisymmetric model were used in other simulations in this study. Simulations were completed comparing the shielded probe to the unshielded probe, resulting in a 2D axisymmetric model of the unshielded probe, shown in Figure 2.3. It should be noted that the geometry of the thermocouple in this model is kept as the same homogeneous rod as the shielded probe.

Modifications were made to the leading edge of the shielding in an attempt to study the effects of leading edge curvature on boundary layer separation. This study was initially performed to determine the leading edge geometry which would not contain point singularities like the idealized square edge, but would remain true to the square edge flow pattern. Figure 2.4 shows the modifications performed to the leading edge for these simulations. From this study, the leading edge geometry with rounded corners (Figure 2.4c) was chosen for all future
Figure 2.2: 2D axisymmetric representation of shielded probe geometry specified by Glawe et al.

Figure 2.3: 2D axisymmetric representation of unshielded probe geometry specified by Glawe et al.

studies, as it provided the least amount of modification from the square leading edge probe while removing the point singularity located at the square corner.
2.2.2 2D Structured Meshing

Meshing of the 2D axisymmetric geometry was performed using ANSYS® ICEM CFD™ meshing software. This was completed by importing the 2D axisymmetric geometry into ICEM CFD, generating the mesh, and exporting the mesh to be suitable for ANSYS Fluent. Meshes for the 2D axisymmetric geometry were comprised of structured quadrilateral elements, generated using a bottom-up blocked mesh technique. As part of the blocked mesh procedure, the complex geometry of the probe was broken down into multiple mesh 'blocks'. The faces of these blocks were then mapped to the faces of the probe geometry. When a quadrilateral mesh was generated inside a rectangular block and mapped to a non-rectangular surface, the mesh followed the curvature of the model geometry.

Particular attention was paid to the mesh sizing at the fluid-solid interaction where the flow boundary layer was located. This region is commonly referred to as the 'inflation layer' during meshing processes, because the surface mesh on the solid is typically extruded, or 'inflated', from the surface, creating a layer of uniformly-sized cells in this region of interest. The behavior of the flow in the near-wall region has a large effect on flow parameters which are important to this study, including the location of flow separation and the amount of fluid-solid heat transfer which occurs. The size of the mesh in this region should be very fine and oriented in the stream-wise direction to properly capture the boundary layer phenomena. The quality of the mesh in this region was measured by monitoring the $y^+$ of the mesh in this near-wall region. The $y^+$ is a measure of the grid size in relation to the wall shear (and subsequently the fluid velocity) over the surface:

$$y^+ = \frac{y \sqrt{\frac{2\mu}{\rho}}}{\nu}$$

(2.4)
where \( y \) is the height from the surface, \( \tau_w \) is the wall shear, \( \rho \) is the fluid density, and \( \nu \) is the fluid kinematic viscosity. For the \( k-\omega \) SST model to function properly in the near-wall region, the height of the first cell should correspond to \( y^+ < 5 \) so that it exists in the viscous sublayer, as discussed by Menter \textit{et al.} [23].

The mesh used throughout the rest of this report, known as Mesh 2 in the grid independence study outlined below, is shown in Figure 2.5. Element counts for this mesh are presented in Table 2.1. The average execution time of the 2D axisymmetric model with this mesh sizing was approximately 5 hours on a server utilizing 12 computing cores for a case with freestream conditions \( M = 0.6, \ p_t = 1 \ atm, \ T_t = 300 \ K. \)

Figure 2.6 shows that the \( y^+ \) values over the surfaces of the probe are within the viscous sublayer, with the exception being the rounded leading edge of the shield. This is due to the increased friction velocity over the rounded surface.

\[ \text{(a) Flow Domain} \]

\[ \text{(b) Probe including solid materials (detailed view of Figure 2.5a)} \]

Figure 2.5: Example of a 2D axisymmetric structured mesh generated using the Glawe \textit{et al.} geometry.
Figure 2.6: 2D axisymmetric values of first-cell $y^+$ for near-wall boundary layer treatment. Note: Post-processing software revolves results for surface visualization. Model is 2D axisymmetric geometry.

Table 2.1: 2D Axisymmetric Mesh Element Counts

<table>
<thead>
<tr>
<th>Domain</th>
<th># of Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Housing</td>
<td>6687</td>
</tr>
<tr>
<td>Potting</td>
<td>2745</td>
</tr>
<tr>
<td>Thermocouple</td>
<td>2448</td>
</tr>
<tr>
<td>Fluid</td>
<td>91781</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td><strong>103661</strong></td>
</tr>
</tbody>
</table>

*Processor-hour/element: $5.79 \times 10^{-4}$*

2D Grid Independence Study

A grid independence study was performed to show that the mesh chosen for study was of an adequate fineness to capture the relevant physics. Simulations were performed on meshes of three varying sizing with an approximate refinement factor, $r$, of 2. The freestream conditions used for this test are the reference conditions utilized throughout this report: $M = 0.6$, $p_r = 1$ atm, $T_r = 300$ K. The method of defining discretization error outlined by Celik *et al.* [24], outlined below, was used to determine if the solution was grid independent.
The mesh spacing, $h$, was defined as

$$h = \left[ \frac{1}{N} \sum_{i=1}^{N} (\Delta A_i) \right]^{1/2} \quad \text{for 2D} \tag{2.5}$$

$$h = \left[ \frac{1}{N} \sum_{i=1}^{N} (\Delta V_i) \right]^{1/3} \quad \text{for 3D} \tag{2.6}$$

with $h$ being the average sizing over the domain, $N$ being the number of elements, and $\Delta A_i$, $\Delta V_i$ being the element sizes for 2D and 3D models, respectively.

<table>
<thead>
<tr>
<th>Mesh Name</th>
<th># Elements</th>
<th>Mesh Spacing, $h$ (m)</th>
<th>$T_j$ (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>424123</td>
<td>.00011329</td>
<td>298.958</td>
</tr>
<tr>
<td>2</td>
<td>103661</td>
<td>.00022916</td>
<td>298.926</td>
</tr>
<tr>
<td>3</td>
<td>24121</td>
<td>.00047505</td>
<td>298.805</td>
</tr>
</tbody>
</table>

Table 2.2: 2D Grid Independence Study Grid Sizes

Using the mesh spacings, shown in Table 2.2, the refinement factor of the grid independence study is defined as

$$r = \frac{h_{\text{coarse}}}{h_{\text{fine}}} \quad \text{(2.7)}$$

the results being shown in Table 2.3. The approximated relative error, $e_a$, and absolute error, $\varepsilon$, were then calculated for each mesh sizing step using formulas of the form

$$e_{a,1-2} = \left| \frac{T_{j,1} - T_{j,2}}{T_{j,1}} \right| \quad \text{(2.8)}$$

$$\varepsilon_{1-2} = T_{j,2} - T_{j,1} \quad \text{(2.9)}$$

where $T_j$ is the junction temperature result for the meshing sizing indicated in the subscript. The observed order of accuracy, $\hat{p}$, of the solution can be calculated using the formula

$$\hat{p} = \frac{1}{\ln(r_{1-2})} \left| \ln \left( \frac{\varepsilon_{2-3}}{\varepsilon_{1-2}} \right) + \ln \left( \frac{r_{1-2}^\hat{p} - s}{r_{2-3}^\hat{p} - s} \right) \right| \quad \text{where } s = 1 \cdot \text{sign} \left( \frac{\varepsilon_{2-3}}{\varepsilon_{1-2}} \right) \quad \text{(2.10)}$$

Note that due to the dependence of $\hat{p}$ on itself, an fixed-point iterative method was used to solve for this value. A second-order accurate discretization scheme was used for this study, therefore this value should approach 2 under ideal conditions. Lastly, the Grid Convergence Index, $GCI$, can be solved using the equation of the form

$$GCI_{1-2} = \frac{1.25e_{a,1-2}}{r_{1-2}^\hat{p} - 1} \quad \text{(2.11)}$$
Table 2.3: 2D Grid Independence Study Results

As the GCI approaches 0, the solution is shown to be grid independent. The results of the grid independence study are shown in Table 2.3. It can be seen that the GCI values for all grid sizes are very small, indicating the solution can be considered mesh independent.

Due to the solution being relatively independent of the meshes considered, the sizing used for further studies can then be based upon ease of mesh generation, calculation time required, smoothness of solution, and stability of solution. The flow solution produced using Mesh 3 contained areas where the element boundaries had large gradients across them and could be clearly seen in the solution, as shown in Figure 2.7. This caused flow features, including the large separation region located external to the shield to not have smooth gradients along them. Mesh 1, while having the highest resolution solution of the meshes simulated, began to have solution stability issues which were not seen in other meshes. Due to the small cell sizes, the acceptable CFL value which maintained solution stability for this mesh was smaller.
than that of the other meshes, resulting in a longer calculation time for this mesh even after scaling by element count. For these reasons, the mesh indicated as Mesh 2 was chosen for further study.

2.2.3 3D Unstructured Meshing

Meshing of the 3D geometry was performed using ANSYS® Meshing, found within the ANSYS® Workbench™ suite of computational analysis tools. This tool was chosen due to its integration in the ANSYS Workbench and high level of interoperability with ANSYS Fluent. This ensured that mesh import to Fluent worked consistently, even with large changes in mesh size and complexity.

The meshes generated for the 3D geometry consisted of unstructured tetrahedral elements with extruded triangular prisms in the inflation layer close to the solids. Unstructured meshing was chosen for the 3D models rather than structured meshing for several reasons. The use of unstructured mesh generation techniques decreased the lead time necessary for mesh generation and simulation setup. The complex internal geometry of the 3D probe proved to be time-consuming to generate the meshing framework for structured elements of complex interior regions. Additionally, due to the 45° cut of the model, triangular elements would be required near the rotational axis, necessitating a hybrid mesh if structured elements were used in other parts of the mesh. Generating an unstructured mesh using an automated solver proved more time-efficient than generating these complex meshes manually.

The disadvantage of using the unstructured tetrahedral mesh is that it require more elements than quadrilateral meshes to achieve similar resolution. This is due to the limitation on skewness of the tetrahedral elements, so that they provide correct solutions with minimal instability [25]. This lower resolution can be seen in the results of the 3D solutions, but is not believed to affect the thermal results, as the affected areas are a sufficient distance from the solids to affect heat transfer solutions.

3D Mesh Development and Grid Independence Study

Due to the substantial computation time to complete a full grid independence study of 3D geometry, sizings for initial 3D meshes were taken from 2D meshing and a less rigorous grid independence study was completed. This study involved generating a mesh which was based upon 2D meshing sizing in the inflation layer and then generating a mesh which was more coarse to compare to. Figure 2.8 shows the 2D mesh and corresponding 3D mesh to compare the inflation layer sizing that was matched. It is shown in this figure that the inflation layers of the meshes were similarly sized to serve as an analog to the grid-independent 2D mesh. The meshes shown in Table 2.4 correspond to the meshes used for the 3D grid study, with Mesh 1 being the mesh based upon 2D mesh sizing.
A grid independence study was completed for the 3D model using the same method as outlined for the 2D model using Equations 2.6 - 2.11. The meshes tested are shown in Table 2.4 and the results of the study are shown in Table 2.5.

<table>
<thead>
<tr>
<th>Mesh Name</th>
<th># Elements</th>
<th>Mesh Spacing, h (m)</th>
<th>Tj (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4007117</td>
<td>.0002729</td>
<td>299.288</td>
</tr>
<tr>
<td>2</td>
<td>561254</td>
<td>.0005255</td>
<td>299.280</td>
</tr>
</tbody>
</table>

Table 2.4: 3D Grid Independence Study Grid Sizes

It is shown that coarsening the mesh by a refinement factor of 2 results in a difference of the junction temperature which is very small.

Mesh 1 is shown in Figure 2.9 and the mesh element counts are presented in Table 2.6. The
average execution time of a 3D model with the above mesh sizing is approximately 50 hours on a machine utilizing 48 compute cores.

Close attention was once again paid to the $y^+$ values of elements near the wall to ensure proper boundary layer capture. Figure 2.6 shows that the $y^+$ values over the surfaces of the probe are within the viscous sublayer.

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$r_{1-2}$</td>
<td>1.9255</td>
</tr>
<tr>
<td>$\varepsilon_{1-2}$</td>
<td>0.008</td>
</tr>
<tr>
<td>$e_{a,1-2}$</td>
<td>.002%</td>
</tr>
</tbody>
</table>

Table 2.5: 3D Grid Independence Study Results

<table>
<thead>
<tr>
<th>Domain</th>
<th># of Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Housing</td>
<td>35446</td>
</tr>
<tr>
<td>Potting</td>
<td>25740</td>
</tr>
<tr>
<td>Thermocouple</td>
<td>21538</td>
</tr>
<tr>
<td>Fluid</td>
<td>3834049</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td><strong>4007117</strong></td>
</tr>
</tbody>
</table>

Processor-hour/element: $6 \times 10^{-4}$

Table 2.6: 3D Mesh Element Counts
Figure 2.9: An example of a 3D unstructured mesh generated using the Glawe et al. geometry (3D Mesh 1).

Figure 2.10: 3D values of first-cell $y^+$ for near-wall boundary layer treatment.
2.3 Simulation Setup

After the meshing of the model was completed, it was imported to ANSYS Fluent and the parameters for the simulation were established. Table 2.7 contains some of the input parameters used in this study for all simulation setup cases. This includes solver and physics model selection, as discussed previously.

<table>
<thead>
<tr>
<th>Models</th>
<th>Solver</th>
<th>Density-based</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Energy</td>
<td>On</td>
</tr>
<tr>
<td></td>
<td>Turbulence</td>
<td>$k - \omega$ SST</td>
</tr>
<tr>
<td>Materials</td>
<td>Air Density</td>
<td>Ideal Gas</td>
</tr>
<tr>
<td></td>
<td>Air Viscosity</td>
<td>Sutherland’s Equation</td>
</tr>
<tr>
<td></td>
<td>Solid Materials</td>
<td>Material-specific $k, C_p$</td>
</tr>
<tr>
<td>Boundary Conditions</td>
<td>Pressure-farfield</td>
<td>Case-dependent $M, p_s, T_s$</td>
</tr>
</tbody>
</table>

Table 2.7: Fluent Simulation Inputs

The boundary conditions for the generalized simulation are shown in Figure 2.11. A 'pressure-farfield' boundary condition allows the user to specify Mach number, static pressure, static temperature, and turbulence conditions for the flow at the model boundary at the freestream flow locations. This was used to ensure that the simulation was directly comparable to the freestream conditions from Glawe et al. [12], which were presented in terms of $M$, $p_t$, $T_t$. The remaining boundary is the rotational axis for the axisymmetric modeling, which is defined by an 'axis' boundary condition.

![Figure 2.11: The Fluent boundary conditions applied to the 2D axisymmetric model.](image)

The boundary conditions applied to the 3D model were the same as the 2D axisymmetric boundary conditions on the corresponding surfaces of the 3D model. At the 45° cut planes, a 'symmetry' boundary conditions was applied. The 'axis' boundary condition used for the 2D axisymmetric model is not applicable for the 3D model and is accounted for by the intersection of the symmetry surfaces located on either side of the cut model.
After setup, simulations were executed using a low value for Courant number (typically 5) until the flow solution stabilizes or began to regularly oscillate. If the case was not stabilized using a low value for the Courant number, this caused the solution to diverge when flow calculations were being performed. When a steady point was reached in the flow solution, the Courant value was raised (ranging from 50-500 depending on the stability of the mesh), and the iterative residual values were significantly reduced. Raising of the Courant number increased the local time stepping of the elements, making it large enough to allow the timescale associated with the conduction through solids to be captured efficiently. The thermal equilibrium of the model was then monitored through the use of ‘volume monitors’ located in the thermocouple of the model. When the temperature at all of these points reached a steady state value, the model was said to be ‘thermally converged’.
3 Results

Studies were completed which characterized the performance of total temperature probes with varying geometric constraints. Some of the effects studied were:

- The process of mass-flow-matching simplified 2D axisymmetric geometry to 3D geometry
- Evaluating the effectiveness of the conjugate heat transfer model used in this study
- The effect of the inclusion of diffuser shields on the aerodynamic recovery
- The effect of the curvature of the shield leading edge geometry on the flow caused by the probe
- The effects of freestream Mach number and pressure on aerodynamic recovery of shielded probes
- Comparison of predictions with experimental values of Glawe et al.

3.1 Mean Flow Features

Figures 3.1a and 3.1b show a typical predicted velocity contour of the shielded probe used in this study. As flow reaches the probe, some of the flow enters the probe, but additional flow goes around the exterior of the probe. The high velocity of this flow, coupled with the bluntness of the shield forms a separation region over the exterior of the probe. This region has a large influence downstream including the probe vent, whose orientation is determined by the height of the separation region above the probe exterior.

There is an additional separation region located inside the probe caused by the adverse pressure gradient resulting from the rapidly changing flow direction at the rear interior wall of the probe as well as the low pressure region in the vent. The internal flow is of great importance due to its direct impact on the heat transfer into the thermocouple. It is imperative that this separation region be consistent when using model simplifications so that the thermocouple is in comparable heat transfer regions.

For the 3D geometry shown in Figure 2.1, it was found that downstream of the vent hole the flow is non-uniform around the exterior of the probe. Locations directly downstream of the vent hole, such as in Figure 3.1a, have a large region of slow, separated flow which has not reattached to the surface of the probe. For locations which are in planes between the
vent holes, shown in Figure 3.1b, the flow quickly reattaches to the surface of the probe. This rapid reattachment of the flow is likely due to the contra-rotating vortices generated by the vent hole, as shown in Figure 3.3. These 3D features rotate such that locations at the periphery of the vent have a downward velocity, forcing the flow to reattach sooner than flow affected by the vent hole directly. This behavior is characterized in Figure 3.2, showing the large separation regions, and the out-of-place vortices around the vent jet. The uneven velocity distribution in the tangential direction causes uneven heat transfer to occur on the exterior of the probe, which is not captured using 2D axisymmetric geometry.

Figure 3.1c shows the pressure distribution within a shielded probe. The region of high pressure within the probe extends forward of the shield, forcing incoming flow to be directed around the shield. The flow going over the leading edge of the shield cannot negotiate the rapid change in direction and the flow separates over the exterior of the probe.
Figure 3.1: Influence of the vent hole in downstream velocity and pressure distribution for 3D model. $M = 0.6, p_t = 1$ atm, $T_t = 300$ K
Figure 3.2: 3D streamlines of velocity characterizing the flow over and through the probe. \( M = 0.6, p_t = 1 \text{ atm}, T_t = 300 \text{ K} \)

Figure 3.3: Velocity vectors demonstrating vortices generated by vent flow. Cutplane taken downstream of vent flow, view is in the streamwise direction facing downstream. \( M = 0.6, p_t = 1 \text{ atm}, T_t = 300 \text{ K} \)

### 3.2 2D Axisymmetric Vent Size Determination

The creation of 2D axisymmetric models of the shielded probe geometry allows for simulations to be completed in much less time than equivalent 3D models. This model simplification requires validation, because some geometric features are inherently 3D and can not be directly simplified to 2D axisymmetric geometry. In the case of total temperature probes, the vent holes used to regulate flow rate are inherently 3D features. The metric used
to qualify that 2D axisymmetric simplifications were appropriate to model the flow near the thermocouple tip was that the internal mass flow rate of the probes in both the 2D and 3D configuration must be equivalent.

A representative set of flow conditions was chosen for this validation, with $M = 0.6$, $p_t = 1$ atm, $T_t = 300$ K. Results for both geometries are shown in Figure 3.4. It is shown that both the 2D and 3D geometries qualitatively show the same flow features, including the separation region over both the shield and the thermocouple. The vent in the 3D model generates a larger downstream disturbance than that of the 2D model. This is to be expected due to the cut plane used to generate Figure 3.4b, which coincides with the vent hole of the model. If a cut plane between the vents were chosen, the downstream disturbance of the vent would be minimal. The 2D axisymmetric model generates a uniform downstream disturbance rather than localized disturbances located at the vent holes. The 2D model is still useful due to its accurate characterization of the separation region around the thermocouple and, therefore, the convective heat transfer to the thermocouple.

Measurement planes were created slightly within the inlet of both geometries and the mass flow rates through the probes were directly compared. The quantities which were compared in this study were the vent area ratio, denoted by

$$ A_{\text{ratio}} = \frac{A_{\text{slot, 2D axisymmetric}}}{A_{\text{holes, 3D}}} $$

(3.1)

and the mass flow ratio, denoted by

$$ \dot{m}_{\text{ratio}} = \frac{\dot{m}_{\text{inlet, 2D axisymmetric}}}{\dot{m}_{\text{inlet, 3D}}} $$

(3.2)

Initial 2D axisymmetric slots were sized by giving them equivalent area as the 3D vent hole configurations, denoted by the ratio $A_{\text{ratio}} = 1$. The mass flow rate through the probe for this initial sizing yielded the result $\dot{m}_{\text{ratio}} = 0.792$. 
Schneider 2015 Chapter 3 Results

Figure 3.4: Comparison of velocity contours in planes through the vent for 2D axisymmetric and 3D geometry for equivalent mass flow rate cases (Iteration 3). $M = 0.6, p_t = 1$ atm, $T_t = 300$ K

The correlation shown between vent area and mass flow rate was found to be unequal. Due to the inherent differences in flow patterns through the circular holes and the axisymmetric slot. The area of the \textit{vena contracta} associated with each of these geometric flow features would provide a more refined method of comparing the 2D and 3D flow geometries. Without detailed knowledge of the \textit{vena contracta} associated with each of these, it was determined that a comparable flow ratio could be found by iterating multiple slot sizes until the internal mass flow rates were matched. To speed convergence, values for slot size were chosen by dividing the current slot area by the mass flow ratio to achieve a better estimate for the next iteration. Table 3.1 shows the history of this process. The final result yielded the mass flow ratio $\dot{m}_{\text{ratio}} = 0.992$ and associated area ratio of $A_{\text{ratio}} = 1.293$. 
Table 3.1: Comparison of 2D axisymmetric and 3D vent areas and associated mass flow rates

<table>
<thead>
<tr>
<th>Iteration</th>
<th>$A_{\text{ratio}}$</th>
<th>$m_{\text{ratio}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.792</td>
</tr>
<tr>
<td>2</td>
<td>1.264</td>
<td>0.977</td>
</tr>
<tr>
<td>3</td>
<td>1.293</td>
<td>0.992</td>
</tr>
</tbody>
</table>

All 2D axisymmetric simulations presented below were obtained with $A_{\text{ratio}} = 1.293$ for the vent slot. Simulations completed for this geometry are shown in Figure 3.6.

Figure 3.5: 2D axisymmetric streamlines of velocity characterizing the flow over and through the probe. $M = 0.6, p_t = 1$ atm, $T_t = 300$ K
Figure 3.6: Regions of interest in predicted flow contours for the 2D axisymmetric model of the shielded probe with $A_{ratio} = 1.293$. $M = 0.6, p_t = 1 \text{ atm}, T_t = 300 \text{ K}$
3.3 Effects of Leading Edge Curvature

The next flow feature which was studied was the effect of the leading edge curvature of the shielding. This curvature was modified in the probe geometry and simulations were completed for a square leading edge, a model with .005 in.-radius filleted rounded corners on the leading edge, and a fully rounded leading edge utilizing a fillet of radius $\frac{1}{64}$ in.

2D axisymmetric flow simulations were performed at these conditions and results were post-processed to produce a velocity contour similar to Figure 3.6a, clearly showing a large separation region being caused by the leading edge of the probe shielding. This separation region was then characterized by generating at iso-line plot for a value which captures the velocity at the boundary of this separation region. For the case presented here, that value was chosen to be $\|\vec{V}\| = 250$ m/s. The iso-line plots were then processed to compare all leading edge geometries, as shown in Figure 3.7.

It is shown in Figure 3.7 that as the leading edge radius of curvature increases, the height of the separation region decreases, thereby affecting the behavior of the vent jet where they meet the separation region boundary.

Figure 3.7: Predicted 2D axisymmetric separation region profiles for all leading edge geometries. Based upon iso-lines at $\|\vec{V}\| = 250$ m/s. Freestream conditions: $M = 0.6, p_t = 1$ atm, $T_t = 300$ K.

3.4 Conjugate Heat Transfer

Figure 3.8 shows predicted 2D axisymmetric temperature contours for probes which had mesh located in the solid geometry, shown in Figure 2.5b, and conjugate heat transfer enabled between the fluid and solids. These models were placed in flows of different total temperatures, as noted in Figures 3.8b and 3.8a. It is shown in both of these models that the conjugate heat transfer in the models is working due to the near-wall temperature being
coupled to the internal temperature of the solids located at that point. It should be noted that in both cases, there is a small region next to the thermocouple with fully stagnated flow. This is due to the flow becoming separated over the thermocouple and following the primary flow path out of the exit slot. This stagnated flow aids in ensuring that the thermocouple is near the total temperature of the flow by surrounding its largest surface in high temperature flow. The correct surface temperature and convection coefficients would allow for better characterization of internal temperature distributions within the probe and allow for complex internal heat transfer simulations to be studied.

It should also be noted that the separation region located on the outside of the shield loses much of its energy to the shield due to its low speed, and it is not re-energized by the flow around it.

Note that in the lower temperature case shown in Figure 3.8a, the internal fluid temperature is very similar to the temperature of the solids surrounding it, suggesting nearly-stagnated flow. The freestream conditions for this simulation were the conditions which the axisymmetric slot size was based upon \((M = 0.6, p_t = 1 \text{ atm}, T_t = 300 \text{ K})\). The slot area was not resized for the higher temperature case shown in Figure 3.8b, so this is considered an off-design case for the slot sizing. Due to the increased temperature of the flow, the density of the fluid is much lower than that of the colder case. This decrease in fluid density causes the internal velocities of the hotter case to be approximately twice that of the colder case due to the change in the behavior of the vena contracta at the lower density. This results in fluid that is not at stagnation temperature and not close to the temperature of the solids.
Figure 3.8: Predicted temperature distribution of the 2D axisymmetric model at conditions $M = 0.6, p_t = 1$ atm for different total temperatures. Note: Temperature scales are dissimilar.

With the implementation of the conjugate heat transfer modeling, it becomes possible to observe phenomena in the interior of the probes which was previously not able to be characterized. Figure 3.9 shows the heat transfer from the fluid to solids elements of the model. It should be noted that close attention should be paid to the magnitude of the color scale of these plots, with values toward the blue end of the spectrum representing higher heat flux values. While the values are qualitatively similar between the 3D and 2D axisymmetric results shown, the differences between them are likely due to limitations of
the 2D axisymmetric modeling in accurately capturing the internal flow features of the probe. The predicted tip temperature values for the 3D and 2D axisymmetric models are 299.288 K and 298.926 K, respectively, which correspond to aerodynamic corrections of $\Delta_0 = 0.0024, 0.0036$. While the difference in the aerodynamic corrections is small, it shows that the difference of flow patterns over the thermocouple affect the indicated junction temperature. The locations of lower heat transfer (denoted in red) are areas which correspond to where the flow has separated over the thermocouple. The regions of higher heat transfer occur where the flow has reattached to the probe. Figure 3.9c demonstrates the behavior of the boundary layer through the use of streamlines of velocity to show the correlation between boundary layer attachment and heat flux.

Figure 3.9: Contours of heat flux from solid to fluid on the thermocouple. $M = 0.6, p_t = 1$ atm, $T_t = 300$ K. Note: 2D axisymmetric model revolved for better visualization
3.5 Unshielded Probe Results

2D axisymmetric models were generated which removed the shield from the probe to represent the unshielded probe tested by Glawe et al. The mesh and boundary condition definition for these models were based upon that of the shielded probe, providing results which are directly comparable to the shielded probe. Some results of these simulations are presented in Figures 3.10 - 3.12.

Figure 3.10 shows that without the shield to stagnate the flow, fluid traveling at the freestream Mach number directly impacts the thermocouple directly at the junction. This interaction with the flow causes a large separation to occur at the corner of the simulated thermocouple junction and remain separated due to the interaction with the blunt edge of the sheath and potting downstream of the thermocouple junction. Below this separation is a recirculation zone with stagnated fluid which is not rapidly replenished with heat from the freestream. The stagnated flow limits the amount of convective heat transfer to the solid of the thermocouple, as seen by the low value of heat transfer shown in Figure 3.12. The heat flux at the flow separation point, located at the corner of the thermocouple junction, is large due to the velocity of the flow passing over it which causes the thermocouple to be cooled at this point.

Due to the axisymmetric assumption employed in this study, the thermocouple is modeled as a solid rod rather than a bare-wire thermocouple as used by Glawe et al. The aerodynamic performance of this rod will be very different than that of cross-flow over the cylindrical wire used in a thermocouple. The shielded probe is less dependent upon the aerodynamic performance of the thermocouple as the flow is essentially stagnated before it reaches the thermocouple through the use of the diffuser shield. In the flow field shown in Figures 3.10 and 3.11, it is apparent that the outcome of these simulations is heavily dependent upon the geometry of the simulated thermocouple and the separation region around it.

![Figure 3.10: 2D axisymmetric streamlines of velocity characterizing the flow over the unshielded probe. $M = 0.6, p_t = 1$ atm, $T_t = 300$ K](image-url)
Figure 3.11: Regions of interest in predicted flow contours for the 2D axisymmetric model of the unshielded probe. $M = 0.6, p_t = 1 \text{ atm}, T_t = 300 \text{ K}$
Figure 3.12: Contours of heat flux from solid to fluid on the thermocouple of the unshielded probe. \( M = 0.6, p_t = 1 \text{ atm}, T_t = 300 \text{ K}. \) Note: Model revolved for better visualization.

### 3.6 Comparisons with Experiment

Axisymmetric simulations were performed to characterize the effects of shielding on the performance of the probe. The simulated conditions were \( M = 0.3 - 0.9, p_t = 1 \text{ atm}, T_t = 300 \text{ K}. \) During the post-processing of these models, a junction temperature was obtained by averaging the values of temperature in the solid region near the pointed end of the
thermocouple. These values were used to compute a recovery correction correction, $\Delta_0$ and compared to the data of Glawe et al. and the Moffat models in Figure 3.13.

Figure 3.14 shows the same data that has been converted to Reynolds number using Equation 1.7 and the stagnation fluid properties. The formulation of Reynolds number was utilized by Moffat [13] in the formulation of the low-order models which are presented below for comparison.

While the 2D axisymmetric CFD/CHT shielded result very nearly matches the Glawe et al. data, the unshielded probe results do not closely match the data. This is likely due to the unshielded probe’s high dependence upon the detailed aerodynamics of flow over the thermocouple.

The simplified Moffat models for the velocity error of the unshielded probes utilizing the recommended value of $\alpha = 0.86$ nearly matches the Glawe et al. data, but as described above, does not match the predicted CFD value, likely due to the large geometric differences between the simulated rod and the bare wires of Moffat. The shielded result of the use of the Moffat model with the $1/8$ internal velocity scaling parameter is also shown. This results in an very small magnitude to the velocity error due the $M^2$ term used in the Moffat model (Equation 1.2), which results in a $1/64$ scale factor being applied to the correction, rendering it physically unrealistic.

![Figure 3.13: The pressure-normalized aerodynamic recovery correction $\Delta_0$ for unshielded and shielded probes over a range of Mach numbers. 2D axisymmetric CFD/CHT predictions compared to data from Glawe et al. and low-order Moffat models.](image)
3.7 Effects of Freestream Mach Number and Pressure

A study was completed to explore the effects of freestream Mach number and pressure on the aerodynamic recovery correction and compare to the data of Glawe et al. This involved using both 2D axisymmetric and 3D models to determine if assumptions made during the 2D conversions were the cause of any limitations in the accuracy of the simulation. To calculate the \( \Delta_0 \) value for simulations completed at \( p_t \neq 1 \) atm involved using Equation 1.6 and Figure 1.9. The results of this study are shown in Figure 3.15. The corrected values taken at \( p_t = 0.2 \) atm would ideally agree with the values at \( p_t = 1 \) atm, but as shown in Figure 3.15, this is not the case. The discrepancy is likely due to the pressure-normalization factor provided by Glawe et al. in Figure 1.9 has only limited applicability in the 2D axisymmetric model. Due to the vent slot behaving differently than vent holes, the \textit{vena contracta} would be affected differently by freestream pressure. Figure 3.16 shows the data over a range of Reynolds values, using Equation 1.7 and the fluid stagnation properties to create a comparable value to that of Moffat [13].

It should be noted that the results using the 3D model vary in a nearly linear fashion, matching the trend of the Glawe et al. data. The 3D results also most nearly match the magnitude of the Glawe et al. data. The 2D axisymmetric data is more nonlinear than either the 3D or Glawe et al. data. This is likely due to the fact that the mass flow for the 2D axisymmetric model was matched at \( M = 0.6 \) conditions, not ensuring that the mass flow
rates are equal at higher Mach numbers. This shows an inherent flaw in the 2D axisymmetric modeling of total temperature probes, with models for each freestream Mach number regime needing to be carefully defined.

Figure 3.15: The aerodynamic recovery correction, $\Delta_0$ variation over a range of Mach numbers for a shielded probe. Predictions compared to data from Glawe et al. and low-order Moffat models.
Figure 3.16: The aerodynamic recovery correction, $\Delta_0$ variation over a range of Reynolds values based upon thermocouple wire diameter for a shielded probe. Predictions compared to data from Glawe et al. and low-order Moffat models.
4 VT Validation Probe

After the study of the Glawe et al. probe was completed, computational analysis was completed for a smaller diameter conventional-style total temperature probe which was embedded in a cooled strut. This work was completed in parallel to low-order modeling and experimental work performed by Englerth [17], such that the data could be compared at the conclusion of the project.

Due to the use of a cooled strut in this experiment, the primary error studied was that of conduction error. For the purposes of this study, the conduction driver, $\Theta$, is defined as

$$\Theta = \frac{T_t - T_b}{T_t}$$

where $T_t$ is the flow total temperature and $T_b$ is the base temperature at the intersection of the thermocouple and the shield. The parameter was varied through an adjustable cooling system attached to the strut, as developed by Englerth [17].

The conduction error in this case was studied experimentally using the Virginia Tech Hot Jet facility, as well as through the use of a thermal resistance model (TRM), detailed earlier in this report. CFD was completed to complement these data sets and compare the results of them comprehensively. In addition to being an extension of the previous work completed on the Glawe et al. probe, this work provided an opportunity to characterize the use of CFD for estimation of conduction errors dominated by large temperature drivers. The temperature ranges explored in this study were high enough that radiation was not truly negligible, but was not an important source of error in the experiment.

4.1 Probe Design

The probe simulated in this exercise was a conventional straight-tube diffuser probe to be mounted within a cooled strut. The cooled strut that the probe was mounted to has an airfoil-shaped outer profile to maintain aerodynamic efficiency and can be cooled using air or water as a cooling medium. This strut was had an additional layer of zirconia thermal barrier coating (TBC) applied to the exterior to minimize the heat transfer from the flow to the strut. This was done to thermally isolate the strut and aid in keeping it at an isothermal state.

The shielded probe shown in Figure 4.1 was designed to be used with this cooled strut, with an outer diameter equal to that of the mounting locations in the strut. Additional design parameters are shown in Table 4.1 and further information can be found in Englerth [17].
This probe was designed to follow current ‘standard’ design procedure associated with total temperature probes. These features included the inlet to vent area ratio, which was suggested to be approximately 20% by Bontrager [10]. This ratio was minimized to decrease velocity error, while ensuring that convective heat transfer was sufficient to decrease conduction and radiation error.

The thermocouple used in this case is a sheathed Type K thermocouple without an exposed junction. This sheath provides improvements in the overall durability of the sensor while providing additional challenges associated with the transient response and the effective thermal conductivity of the assembly. The use of sheathed thermocouples has become advantageous to designers with the high-temperatures and unsteady aerodynamic loading seen in the design of probes for turbine engines. It also proved easier to model than the bare-wire probe of Glawe et al. [12].

<table>
<thead>
<tr>
<th>Shield</th>
<th>OD</th>
<th>0.08 in.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material</td>
<td>Stainless Steel 303</td>
<td></td>
</tr>
<tr>
<td>Length</td>
<td>0.58 in.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Vent Holes</th>
<th>Number</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter</td>
<td>0.0135 in.</td>
<td></td>
</tr>
<tr>
<td>Distance from LE</td>
<td>0.193 in.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Thermocouple</th>
<th>Type</th>
<th>Sheathed Type K</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sheath Diameter</td>
<td>0.032 in.</td>
<td></td>
</tr>
<tr>
<td>Sheath Material</td>
<td>Inconel</td>
<td></td>
</tr>
<tr>
<td>Wire Diameter</td>
<td>0.01 in.</td>
<td></td>
</tr>
<tr>
<td>Wire Material</td>
<td>Chromel-Alumel</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Strut</th>
<th>Material</th>
<th>Inconel</th>
</tr>
</thead>
<tbody>
<tr>
<td>TBC Material</td>
<td>Zirconia</td>
<td></td>
</tr>
<tr>
<td>TBC Thickness</td>
<td>0.01 in.</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.1: Geometric parameters of the probe designed by Englerth [17]

Figure 4.1: Manufacturing drawing of probe designed by Englerth. All dimensions in inches.
4.2 Computational Models

Simulations of this probe were performed using 3D geometry, shown in Figure 4.2, which utilized a 180° symmetry plane on which the geometry was cut. The symmetry plane was placed so that the airfoil strut was divided along its pitch axis, as shown in Figure 4.2. The use of a 180° symmetry plane allowed for complex geometry, such as the internal features of the sheathed thermocouple, to be simulated. Other complex features, such as internal cooling channels of the strut were also modeled. These cooling channels allowed for the base temperature boundary condition to be placed at multiple locations, providing a more physically-realizable boundary conditions to use for thermal equilibrium calculations.

Figure 4.2: Computational model of validation case.

A mesh was generated for this case using some of the knowledge gained during the grid independence studies associated with the Glawe et al. probes. As shown previously, thermocouple junction temperature is only changed marginally with meshes of increasing complexity. Due to this, it was decided to use a mesh which was relatively coarse in the flow regions, with emphasis being placed on the region closest to the probe and the boundary layers at the surface. For this reason, it can be seen in Figure 4.3 that a refinement region was generated in the near-probe region which resembled the shape of the exterior separation zone generated by the blunt-edge shield of the probe. Additionally, the boundary layer meshing over the probe faces was were refined with to a sufficient level to work with the selected turbulence models. Table 4.2 contains the element counts of the mesh, given component-wise of the model.
Mesh adaptation was investigated for use in this model to refine the mesh when flow properties were rapidly changing in areas of interest. It was found that when adaptation was employed, the regions which were refined were the regions around the strut which contained the largest variable gradients in the flow domain. Due to this, the mesh in the regions of interest around the probe were not refined to the point of improvement before the freestream mesh was refined to a point of unnecessary refinement. For these reasons, a manual refinement region was chosen based on predicted flow behavior and this region was given a high degree of mesh refinement than the freestream flow.

Simulations were set up using the same computational models utilized as the Glawe et al. probe, as noted in Table 2.7. The boundary conditions that were applied to the model are shown in Figure 4.4. The base temperature, $T_b$, as determined from the temperature driver, $\Theta$, was applied to the interior surfaces of the cooled channels within the strut. A zirconia thermal barrier coating was simulated on the exterior of the strut by modifying the boundary condition at the fluid-solid interface to be a material of finite thickness (0.01 in.). The compute time of this model was approximately 15 hours, utilizing 48 compute cores.
Figure 4.3: Computational mesh used for validation case.
Table 4.2: Validation Case Mesh Element Counts

<table>
<thead>
<tr>
<th>Domain</th>
<th># of Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Strut</td>
<td>331448</td>
</tr>
<tr>
<td>Shield</td>
<td>33206</td>
</tr>
<tr>
<td>Potting</td>
<td>13890</td>
</tr>
<tr>
<td>Sheath</td>
<td>23885</td>
</tr>
<tr>
<td>Thermocouple</td>
<td>4898</td>
</tr>
<tr>
<td>Fluid</td>
<td>740169</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td><strong>1147495</strong></td>
</tr>
</tbody>
</table>

\(\text{Processor-hour/element: } 6.27 \times 10^{-4}\)

Figure 4.4: Applied boundary conditions of validation case.
4.3 Results

The test matrix of the experimental portion of this study was determined by the capabilities of the Hot Jet facility which was used. The cases studied experimentally were completed over the ranges of $M = 0.1 - 0.8$, $T_i = 422.039 - 727.594$ K, and $\Theta = 0.05 - 0.6$. To bound the test space, simulations were completed at the extremes of this test matrix, as well as intermediate points to aid in showing data trends.

Figures 4.5 and 4.6 show sample velocity and temperature contours of the fluid around the probe. The flow patterns of this probe have both similarities and differences from the Glawe et al. probe, displaying fundamental commonalities as well as geometry-specific features. The largest difference between the flowfields is the inclusion of the airfoil-shaped strut and its effect on the flow over the probe. As seen with the Glawe et al. probe, the external flow separates at the leading edge of the probe shield and passes over the vent, but unlike the Glawe et al. probe, the flow is pushed further away from the body by the interaction with the strut rather than reattaching downstream. This increased separation distance is likely due to the large pressure gradient induced upstream of the stagnation point of the thick airfoil. This mount is required to be of the thickness shown due to the internal cooling channels required to generate temperature drivers. The flow being forced over the thick airfoil then causes a large velocity gradient to occur over the location of maximum thickness, which results in uneven thermal loading over the surface of the strut.

Due to the location of the vent hole being further upstream than most conventional probes, there is a large region of stagnated flow inside the probe aft of the vent holes. The air is not replenished at an appreciable rate, causing conduction through the air to be the primary mode of heat transfer in this region. It can be seen in Figure 4.6 that this region is shown having relatively low temperatures due to its direct contact with the cooled strut.

![Figure 4.5: Velocity contours of the flow field of the validation probe. $M = 0.8$, $p_s = 0.932$ atm, $T_i = 727.59$ K, $\Theta = 0.5$.](image-url)
Figure 4.6: Temperature contours of the flow field of the validation probe. $M = 0.8$, $p_s = 0.932$ atm, $T_t = 727.59$ K, $\Theta = 0.5$.

Figure 4.7 shows the internal temperature distribution of the solids of the probe and the leading edge of the strut. The effects of the cooled strut can be seen by the large gradients in temperature seen through the streamwise direction of the model. The temperature in the probe varies linearly over the length of the probe, which is expected because the model is in thermal equilibrium, balancing the conduction down to the strut and the convective heat loading over the surfaces. It is seen in both of the cases shown that the temperature gradients are not constant for the shield and the thermocouple. This inequality causes similar streamwise locations within the shield to be different temperatures in the shield and thermocouple. This is likely due to the thermocouple being treated as a thermal composite material, consisting of multiple materials with varying thermal conductivities. The thermocouple as a whole is often treated as having an 'equivalent' thermal conductivity which is equal to the conductivities of the individual component materials if they were wired in parallel down the length of the thermocouple, according to thermal resistance modeling. The difference in thermal conductivities causes a change in the thermal equilibrium point which is reached for each of the solid materials, as this consists of a balance of thermal loadings and their energy transport rates.

It can also be seen in Figure 4.7 that the thermal loading is larger for the higher velocity case. This is due to the increased convective loading caused by the increased flow speed both inside and outside of the shield. The increased thermal loading can be seen as causing a larger increase in temperature inside of the strut, which is not seen in the lower speed case. This phenomena is also exaggerated by the modeling of the solid-solid material interfaces as having perfect thermal contact. This perfect thermal contact assumption implies that there is a high level of physical contact between the surfaces, as well as containing no imperfections in the physical interface. This is not a physically realizable assumption, but assuming perfect thermal contact with the knowledge that the answer is 'ideal' would result...
in a better understanding than applying imperfect thermal contact conditions without true knowledge of the actual values which are physically realizable and seen in experimentation.

Figure 4.7: Temperature distribution within the solids of the validation probe. $p_s = 0.932$ atm, $T_f = 727.59$ K, $\Theta = 0.5$.

Figure 4.8 shows the heat flux from the fluid to the solids of the thermocouple. One of the uses of these plots is that they can be potential indicators of locations where heating could be uneven. If the thermocouple junction were to be located in one of these 'hot spots,' an inaccurate reading of the flow could be reported by the thermocouple. These 'hot spots' are easily visible for the $M = 0.8$ case where the vicinity around the vent hole is highly nonuniform in the heat flux distribution. This nonuniformity is very localized to the area in direct contact with the vent flow, but the locations near this spot could be potentially affected by its nonuniformity. Additionally, it is shown that the heat flux around the leading edge of the thermocouple is raised due to the higher level of convective heat transfer caused by the increased velocity over the tip. It is also shown that the rearmost portion of the thermocouple experiences a positive heat flux due to contact with the cooled strut and the lower temperature, stagnated flow in the rear of the probe, as shown in Figure 4.6.
Figure 4.8: Heat flux from the solids over the thermocouple of the validation case. $p_s = 0.932$ atm, $T_t = 727.59$ K, $\Theta = 0.5$.

To directly compare the data experimental work, thermal resistance modeling (TRM) and CFD, it was determined that the metric to be compared should be the simplified overall probe recovery, $R$, due to the fact that this is a combination of all error sources and is not specific to any one type of error. Overall recovery is defined as

$$R = \frac{T_j}{T_t} \quad (4.2)$$

where the ideal value is 1, indicating that junction temperature is equal to the flow total temperature.
Figure 4.9 shows the combined results as calculated from experimental work, TRM, and CFD for 2 total temperature values and 2 driver values. The data is displayed in terms of Reynolds number rather than Mach number to highlight the effects which are not solely dependent upon Mach number. The data shown was also given a Mach correction to remove Mach number effects utilizing the Moffat simplified models. This characterization is defined as

\[ R_{corrected} = R + (1 - R_{velocity}) \quad \text{where} \quad R_{velocity} = (1 - \alpha) \frac{\gamma - 1}{2} \frac{M^2}{1 + \frac{\gamma - 1}{2} M^2} \]  \hspace{1cm} (4.3)

where \( \alpha \) is the recovery factor value from Moffat for wires parallel to the flow (\( \alpha = 0.86 \)). This corrected overall recovery highlights the conduction error of the problem, which becomes the primary error source when this Mach correction has been applied.

It is shown in Figure 4.9 that there is good agreement between experimental and CFD results when the Mach correction factor has been applied. Due to facility limitations, experimental data was difficult to obtain for very low Mach numbers (and thus low Reynolds), which does not allow the user to see if the trend at low Mach number seen in CFD and TRM persists in experimental data. The CFD values at higher Reynolds number slightly overpredict the experimental results for both total temperature values shown, which is likely due to the lack of radiation modeling in the CFD. In the temperature regimes shown, radiation does not dominate, but could have an effect similar to that seen in the discrepancies of the experimental values to the CFD. The data utilizing the TRM model underpredicts values of recovery across the given Reynolds range. This is likely due to the limited physics associated with the model, including the limited convective heat transfer information given at the model boundaries. This modeling does appear to follow the perceived trend of the experimental and CFD data, with reduced recovery being seen at lower Reynolds values, as predicted by theory.
This study was used to extend the principles learned from studying the Glawe et al. probe to a case with different geometric parameters and data which it could be validated against. Some of the advantages of using CFD were highlighted during this study, including the ability to characterize complex phenomena occurring inside of small diameter probes, as well as the ability to predict issues with the geometric configuration of the probe before manufacturing and experimentation began.

CFD could be used further to improve the design of this probe, studying such items as optimization of the strut and its interaction with the probe flow, probe vent orientation with reference to the strut, or design of a multi-probe strut in which probe aerodynamic profiles do not interfere with each other. Each of these items could be accomplished while using CFD during the design process to further improve the probes. CFD could additionally be used to provide preliminary results of implementing these changes so that when a design is chosen, the performance can be predicted at specified points in the design matrix while experimentation is ramping up.

The use of the conjugate heat transfer modeling proved effective in the characterization of conduction error within these probes, not only showing the balance between the convection and conduction in the thermal equilibrium, but also allowing the heat flux into the model to be characterized, such as in Figure 4.8. This provided insight into design recommendations for future probes, such as the importance of the placement of the thermocouple junction in
relation to the vent holes and its effect upon the heat flux experienced at a point. The effects of freestream Mach number on the convective heat transfer and the steady-state thermal equilibrium can be seen in plots such as Figure 4.7.

It was shown that not only could CFD be coupled with experimentation and low-order models, but it is shown to complement them and add to the overall strength of a test program. While experimentation is useful to true physical modeling and being able to generate data which fills in the test matrix, CFD has a place as an early design tool and a predictor of performance where experimentation and low-order modeling are limited. No one piece of this test plan should be used without validation of the others, but as a complete package their strengths can be combined to form a start-to-finish design solution for total temperature probes.
5 Conclusions

It was shown in this study that total temperature probes can be modeled using CFD/CHT. These probes contain a mixture of complex flow phenomena, with results being sometimes non-intuitive. This section will review the behavior seen for these probes and the lessons learned about the complex flow fields surrounding them.

The large separation region over the exterior of shielded probes is the most prominent flow feature shown in these simulations. This is due to the inability of a streamline to remain attached around the blunt leading edge, and is strongly affected by the leading edge curvature of the shield. This separation region has large implications to the heat transfer experienced by the probe shield. The reduced velocity of the region under the separation increases the thermal conduction to the shield, resulting in stagnant, low-energy flow at the surface. This would lead to the shield located in this region to be slower in response time to any temperature changes in the freestream due to its low convective heat transfer rate. The vent jet also has an effect on the separation region, often causing an disturbance in the shape of the separation region at the point where the vent jet meets the separation, seen for both 2D axisymmetric and 3D geometry. This effect is amplified in 3D geometry, where the separation region behind the vent is heavily affected, while the regions not directly behind the jet are affected by flow features causing the flow to reattach.

The suitability of using 2D axisymmetric geometry for the analysis of total temperature probes was characterized. While the 2D axisymmetric geometry provided a platform for studies of large geometric changes in these probes, it was determined to have physical limitations and assumptions which are only valid at design conditions. When simplifying models from 3D geometry to 2D axisymmetric, the fundamental flow differences between 3D vent holes and 2D axisymmetric vent slot require internal flow conditions of the models to be matched. Using equal flow area for the vent features does not provide similar internal mass flow rates. This is likely due to the physical area of the geometric feature not being equal to the effective flow area driven by the vena contracta through the geometry. For the case of $M = 0.6$, $p_t = 1$ atm, $T_t = 300$ K, the effective area correction for equivalent mass flow through the probe was found to be $A_{\text{ratio}} = 1.293$. The internal mass flow rates through the probes are dependent upon freestream conditions as well as geometry. At off-design conditions (conditions where the 2D vent was not resized by comparisons with 3D calculations), the validity of the axisymmetric assumptions rapidly deteriorated due to changes in internal flow rate caused by changes in freestream Mach number, pressure, or temperature. The reasoning behind this is likely due to the fundamental difference in the modeling of the vena contracta of the 3D vent hole to the 2D axisymmetric slot. When the mass flow rates were matched at the specific boundary conditions, the area of the vena contracta for the 2D and 3D flow features were likely the same. When off-
design conditions are presented to the probe, the response of the *vena contracta* through the different geometries were fundamentally different. To maintain consistent internal mass flow rates, 2D axisymmetric geometry must be generated for each freestream conditions tested, making the process of using 2D axisymmetric geometry inefficient. While the execution time of a 2D axisymmetric model is an order of magnitude faster while utilizing $\frac{1}{4}$ of the computational power of an equivalent 3D model, the time spent generating individual geometries and meshes based upon equivalent mass flow rates can often negate the time savings of using 2D axisymmetric models.

2D axisymmetric geometry also does not allow the exact geometry of bare thermocouple wires to be modeled. This simplification alters the flowfield in close proximity to the thermocouple, changing the heat flux into the thermocouple and therefore altering the temperature distribution within the geometrically simplified model. This condition is highly evident in results for the unshielded probe, due to the bare-wire elements being exposed to flow at the freestream Mach number value, suffering largely from geometry-dependent velocity error. Due to the use of bare-wire thermocouples in the experiments of Glawe *et al.*, 2D axisymmetric wire geometry had an effect on probe performance, particularly for unshielded probes.

It was shown during the analysis of the VT validation probe that the use of sheathed thermocouples reduces the dependence of the solution on the exact thermocouple geometry. The sheaths used on these thermocouples can often be simplified to 2D axisymmetric shapes, which can be more readily modeled using geometry containing multiple planes of symmetry. The heat flux and temperature distribution on the exterior of this thermocouple sheath can be accurately characterized, which bare-wire model simplifications are unable to capture.

An additional limitation of 2D axisymmetric models includes its inability to model the tangential non-uniformity of the flowfield around the 3D model. Higher local velocities and heat transfer loading are shown directly downstream of the vents due to the vortices generated by the vent hole. This is not accounted for in 2D axisymmetric models, due to the assumed uniform vent slot. The non-uniformity of the external temperature distribution is made more evident in the 2D axisymmetric model by the lack of solid material bridging the upstream portion of the shield to the downstream portion of the shield. This thermal disconnect would not allow the true equilibrium temperature of the solids to be reached, especially in cases of extreme temperature loading on the upstream portion of the shield due to the large separation region over the exterior.

While the 2D axisymmetric model is limited in its physical applicability, it is useful for its ability to rapidly characterize geometric changes and their effect upon the flow field around the probe. Geometric modifications were made to the probe shield and simulations were completed to study the effects of the large geometric changes. The effects on the flowfield due to the curvature of the leading edge of the shield, as well as the removal of the shield were studied using 2D axisymmetric geometry. Due to the physical applicability of these studies being questionable, many of the results were more useful in a qualitative rather than
a quantitative fashion. One of the findings of these models was that the leading edge of the shield has an impact on the separation region located around the exterior of the probe, which in turn has an effect on the overall heat transfer into the solids of the probe. The difference in the flowfield of the shielded probe from the unshielded probe was characterized, making evident that the exact modeling of the thermocouple geometry was crucial for unshielded probes.

The 3D modeling completed in this study provided valuable insight to characterize the complex flows in real probe geometries. The results of the 3D models were very near those of historical results for the Glawe et al. study and contemporary experimental results for the VT validation probe study. The modeling lessons learned from the Glawe et al. study were applied to the VT validation probe study and tested to ensure that the CFD remained physically valid. In the Glawe et al. study, the mesh resolution near the probe was of sufficient quality to capture the pertinent phenomena, but the mesh resolution further away from the model was likely too fine to be computationally efficient. The farfield computational domain is important only in that it provides a ‘boundary condition’ to the near-probe flow. For the VT validation probe study, the farfield mesh density was coarsened to make the model more computationally efficient, while maintaining the near-probe mesh refinement used for the other case. It was shown that the mesh resolution in the farfield region could be decreased while still capturing the flow and heat transfer phenomena relevant to the study of total temperature probes. This mesh coarsening allowed the 3D modeling of these probes to be more time efficient, with faster turnaround time for results.

Through the completion of the experimentally-paired VT validation study, the effectiveness of CFD in the life-cycle of total temperature probe design was examined. The strength of CFD in the case of total temperature sensors is its ability to model geometric changes and their effects on the overall flowfield over and through the probe. This ability is best used in the early design process of a probe to model large-scale geometric changes. The pure numerical nature of CFD allows probes of varying sizes and shapes to be modeled with little discrimination. Probes of very small diameter, of which the interior can not be physically characterized with instrumentation, can be simulated. This allows the user to understand the complex flow features in and around the probe when governed by small length scales. This ability to characterize the behavior at small length scales allows for the simulation of newer, miniaturized probes which are being standardized for use in turbomachinery.

Turnaround time from preliminary design of a probe to CFD results can often be much shorter than the time to complete the probe design, have it manufactured, and the experimental apparatus modified to work with the probe. During the lead time that an experimental probe is being manufactured and calibrated, CFD simulations can be performed to understand the physics of the flowfield of the probe before experimentation begins. Experimentation would likely be more time-efficient than CFD when the probe has been manufactured and calibrated. CFD solutions which are generated to validate the experimental results of a probe while it is being manufactured are often useful. This allows the CFD user time to prepare the simulations and begin producing results and
building a database of validation cases for the experimentalist before experimentation of the manufactured probe begins.

A useful trait of CFD, which could be more fully implemented in future studies, is that it can be used to test extreme conditions which experimental facilities may not be capable of. While CFD physics can be limited in extreme cases, such as very high temperatures or Mach number values, it may be able to provide data points outside of the limitations of experimental facilities. This can be useful for bounding the design space of a sensor. If CFD predicts that a sensor will produce results well outside of the acceptable limits, these results can be used to shape the experimental test matrix to validate that this is physically accurate.

5.1 Future Work

One of the primary areas which are not covered in this study were the effects of radiation. In the high-temperature regimes which most total temperatures probes operate, radiation is one of the primary modes of heat transfer. The effects of radiation are important in understanding the differences between shielded and unshielded probes in hot environments. An additional effect of the shielding is its effect on mitigating radiation from exterior sources. Characterizing this effect would aid in the design of these probes for use in high temperature environments. This could be readily implemented in the current simulation framework due to its inclusion in ANSYS Fluent as a solver option.

Additional work which would further the results of this study would be to complete simulations using full 3D geometry of both shielded and unshielded probes. The exact geometry of the bare wire thermocouple was not modeled in this study, and it would have large effects on the aerodynamic recovery of both probes, with emphasis on the unshielded probe. This could be completed with a 180° geometry model to conserve computational energy around a symmetry plane. This study would also have implications in the study of radiation, as using the exact thermocouple surface area and view factors would be essential to accurately characterizing the radiation being both absorbed and emitted by the thermocouple.
Bibliography


