INVESTIGATION OF THERMAL TRANSPORT IN AN ACCELERATED TURBULENT FLOW

A Dissertation

Submitted to the Graduate School
of the University of Notre Dame
in Partial Fulfillment of the Requirements
for the Degree of

Doctor of Philosophy

by
Arman Mirhashemi

Scott C. Morris, Director

Graduate Program in Aerospace and Mechanical Engineering
Notre Dame, Indiana
August 2018
INVESTIGATION OF THERMAL TRANSPORT IN AN ACCELERATED TURBULENT FLOW

Abstract

by

Arman Mirhashemi

This research investigated the turbulent transport of heat and momentum in an accelerated and highly turbulent flow with large density gradients. Both experimental and numerical data were acquired to study this phenomenon. Experimental measurements were obtained at the inlet and exit of an annular nozzle with two different inlet total temperature distributions. The two cases included a nominally uniform total temperature distribution and a non-uniform total temperature distribution where the total temperature at the walls was lower than the center of the span by approximately 10%. In order to evaluate various turbulence models and quantify the turbulent Prandtl number, numerical solutions were obtained in a computational domain similar to the experimental geometry. A theoretical model for turbulent Prandtl number was developed based on the intermediate mixing length concept. This model correctly predicted the turbulent Prandtl number required for the computational solution to closely match the experimental data for the nozzle flow rig. This model was then also validated against independent experimental measurements of turbulent Prandtl number in jet flow, flat plate boundary layer, and turbulent pipe flow.
This dissertation is dedicated to my family, and to all who fight ignorance with knowledge. My parents, Mohammadreza and Forough, who never stopped supporting me, even from thousands of miles away. Lastly, my brother Damon, who has shared the wonders of music and life with me.
CONTENTS

FIGURES ................................................................. vi

TABLES ................................................................. viii

SYMBOLS ............................................................... ix

ACKNOWLEDGMENTS ...................................................... xi

CHAPTER 1: INTRODUCTION ........................................... 1

1.1 Background and Motivation ....................................... 1

1.2 Analogies between Heat and Momentum Transfer Mechanisms in Turbulent Flows ................................................................. 6

1.2.1 Reynolds Analogy ................................................. 6

1.2.2 Strong Form of Reynolds Analogy ............................ 9

1.2.3 Prandtl-Taylor Analogy ....................................... 10

1.2.4 Von-Karman Analogy .......................................... 12

1.2.5 Chilton-Colburn Analogy ..................................... 13

1.3 Turbulent Prandtl Number Models ............................. 14

1.4 Research Objectives .............................................. 18

CHAPTER 2: EXPERIMENTAL FACILITY .......................... 20

2.1 Annular Nozzle Flow Facility ..................................... 20

2.2 Facility Instrumentation .......................................... 24

2.2.1 Total Pressure Measurements ................................. 25

2.2.2 Total Temperature Measurements ........................... 26

2.2.3 Standard Day Correction ..................................... 28

CHAPTER 3: HOTWIRE MEASUREMENTS .......................... 30

3.1 Hotwire Probe Manufacturing .................................... 35

3.2 Calibration Method ................................................. 38

3.3 Validation .......................................................... 42

CHAPTER 4: EXPERIMENTAL RESULTS .......................... 43

4.1 Nozzle Inlet ........................................................ 44

4.2 Nozzle Exit ........................................................ 47

iv
<table>
<thead>
<tr>
<th>FIGURES</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1 Schematic of Eddy Fluctuations in Turbulent Flow</td>
<td>4</td>
</tr>
<tr>
<td>1.2 Plane element of area showing fluctuating velocity components and the flux $J + j$ in $y+$ direction.</td>
<td>7</td>
</tr>
<tr>
<td>2.1 Schematic of Annular Nozzle Flow Facility</td>
<td>21</td>
</tr>
<tr>
<td>2.2 Schematic of the Annular Contraction in Annular Nozzle Flow Facility</td>
<td>22</td>
</tr>
<tr>
<td>2.3 Inlet conditions of the considered cases, (a) Circumferential averaged total temperature, (b) Circumferential averaged total pressure.</td>
<td>24</td>
</tr>
<tr>
<td>2.4 Schematic of 90 degrees section of the turbulence grid</td>
<td>24</td>
</tr>
<tr>
<td>2.5 Total Pressure Kiel Probe</td>
<td>26</td>
</tr>
<tr>
<td>2.6 Total Temperature Kiel Probe</td>
<td>26</td>
</tr>
<tr>
<td>2.7 Total Temperature Recovery Factor Calibration</td>
<td>27</td>
</tr>
<tr>
<td>3.1 Schematic of Hotwire Probe</td>
<td>35</td>
</tr>
<tr>
<td>3.2 Images of a Single Hotwire Prong Illustrating the Two-Step process for Attaching the sensor wire to the Prong, (a) Sensor Wire is Welded on the top surface of the Prong. (b) Spot Weld is Covered with a Thin Layer of Silver Paste</td>
<td>38</td>
</tr>
<tr>
<td>3.3 Hotwire Calibrations, (a) Hotwire Calibrations at Different Temperatures. (b) Corrected Hotwire Calibration Plots for a single temperature.</td>
<td>41</td>
</tr>
<tr>
<td>3.4 Comparison between velocity from hotwire measurement and total pressure measurement, (a) Low Mach number comparison (b) High Mach number comparison.</td>
<td>42</td>
</tr>
<tr>
<td>4.1 Uniform total temperature distribution case, (a) Normalized inlet total pressure, (b) Normalized inlet total temperature.</td>
<td>45</td>
</tr>
<tr>
<td>4.2 Inlet turbulence intensity distribution in uniform case</td>
<td>46</td>
</tr>
<tr>
<td>4.3 Non-uniform total temperature distribution case, (a) Normalized inlet total pressure, (b) Normalized inlet total temperature.</td>
<td>47</td>
</tr>
<tr>
<td>4.4 Inlet turbulence intensity distribution in non-uniform case</td>
<td>48</td>
</tr>
<tr>
<td>4.5 Uniform total temperature distribution case, (a) Normalized exit total pressure, (b) Normalized exit total temperature.</td>
<td>50</td>
</tr>
</tbody>
</table>
4.6 Exit turbulence intensity distribution in uniform case
4.7 Non-uniform total temperature distribution case, (a) Normalized inlet total pressure, (b) Normalized inlet total temperature.
4.8 Exit turbulence intensity distribution in non-uniform case
5.1 Inlet turbulence intensity distribution in uniform case
5.2 Computational inlet boundary conditions of non-uniform cases, (a) Total pressure. (b) Total temperature. (c) Turbulence kinetic energy (d) Turbulent dissipation (e) Specific turbulent dissipation
5.3 Grids generated for the computational study, (a) Nozzle grid. (b) Nozzle wall grid
5.4 Comparison between the computational results across the different grids, (a) Total pressure. (b) Total temperature. (c) Mean axial velocity. (d) Turbulence intensity
6.1 Comparison between experimental measurements and computational analysis based on realizable $k-\varepsilon$ model, (a) circumferentially averaged total pressure. (b) circumferentially averaged total temperature.
6.2 Comparison between experimental measurements and computational analysis based on SA model, (a) Circumferentially averaged total pressure. (b) circumferentially averaged total temperature
6.3 Comparison between experimental measurements and computational analysis based on Kato-Launder $k-\omega$ model, (a) Circumferentially averaged total pressure. (b) circumferentially averaged total temperature
6.4 Comparison between experimental measurements and computational analysis based on the Reynolds Stress Model, (a) Circumferentially averaged total pressure. (b) circumferentially averaged total temperature
6.5 Comparison between experimental measurements and computational analysis based on SST $k-\omega$ model, (a) Circumferentially averaged total pressure. (b) circumferentially averaged total temperature.
6.6 L2-norm between the experimental measurements and numerical solutions for various turbulence models, (a) $S_{II}$ for circumferentially averaged total pressure. (b) $S_{\Omega}$ for circumferentially averaged total temperature
6.7 Comparison between experimental measurements and computational analysis with varying constant turbulent Prandtl number based on SST $k-\omega$ model, (a) Circumferentially averaged total pressure. (b) circumferentially averaged total temperature
### TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>Specifications of adhesive materials used in probe fabrication</td>
<td>35</td>
</tr>
<tr>
<td>4.1</td>
<td>Measured experimental flow properties</td>
<td>42</td>
</tr>
<tr>
<td>5.1</td>
<td>Turbulence models investigated in the numerical analysis</td>
<td>51</td>
</tr>
<tr>
<td>5.2</td>
<td>Number of cells in the computational grids.</td>
<td>56</td>
</tr>
</tbody>
</table>
SYMBOLS

\[ \rho \quad \text{Density} \]
\[ P \quad \text{Pressure} \]
\[ \bar{u} \quad \text{Time Averaged Velocity} \]
\[ u' \quad \text{Velocity Fluctuations} \]
\[ \tau \quad \text{Shear Stress} \]
\[ \nu \quad \text{Viscosity} \]
\[ \epsilon_h \quad \text{Eddy Diffusivity of Heat} \]
\[ \epsilon_m \quad \text{Eddy Diffusivity of Momentum} \]
\[ \delta_{ij} \quad \text{Chronicle Delta} \]
\[ k \quad \text{Turbulent Kinetic Energy} \]
\[ \bar{T} \quad \text{Time Averaged Temperature} \]
\[ T' \quad \text{Temperature Fluctuations} \]
\[ \alpha \quad \text{Thermal Diffusivity} \]
\[ Pr \quad \text{Molecular Prandtl Number} \]
\[ Pr_t \quad \text{Turbulent Prandtl Number} \]
\[ Sc \quad \text{Schmidt Number} \]
\[ q \quad \text{Heat Transfer} \]
\[ l_h \quad \text{Thermal Mixing Length} \]
\[ c_p \quad \text{Specific Heat, Constant Pressure} \]
\[ St \quad \text{Stanton Number} \]
\[ Re \quad \text{Reynolds Number} \]
\[ Pe \quad \text{Peclet Number} \]
$C_f$  Friction Coefficient

$C_d$  Drag Coefficient

$h$  Convective Heat Transfer Coefficient

$\eta$  Normalized Radial Location

$\Theta_t$  Normalized Total Temperature

$\Pi_t$  Normalized Total Pressure
ACKNOWLEDGMENTS

I would like to express my appreciation and outmost gratitude to my advisor Prof. Scott Morris for his guidance and support throughout my time at Notre Dame. He showed me how to think like a scientist, and how to be my own critic in the endeavor to strive for excellence.

I would also like to thank my dissertation committee members, Drs. Mihir Sen, Kenneth Christensen, and Aleksander Jemcov for their advice and encouragement throughout my studies, and for generously spending their time helping me to improve this work.

I owe a great deal of gratitude to my current and former colleagues Andrew Oliva, Adam Hickman, Nick Jaffa, and Tamuto Takakura for always being available to discuss ideas. My special thanks goes to Joshua Szczudlak, which this work was not possible without his help. May we get another chance to work together and watch random TV shows while working long hours. Technical abilities of Orlando Mendez provided an invaluable asset for this work and I would like to thank him for it.

A portion of this work was a collaboration with General Electric. I have had the great opportunity of working with and benefiting from the inputs of our collaborators Sara Rostami and Greg Sluyter.

Lastly, I would like to thank my friends, Ali Rahnama, Elia Shahbazi, Maryam Naghashan for their immeasurable support. I specially appreciate Zeinab for all the support, friendship, and patience she has shown towards me. Elaheh was a rock of a friend through some of the hardest times in my life, thank you for your friendship.
CHAPTER 1

INTRODUCTION

1.1 Background and Motivation

Most fluid flows occurring in nature and engineering problems are turbulent. The wide range of length scales in turbulent flows contributes to the mixing and increased rates of mass, momentum and energy transport. This study focused on understanding and predicting turbulent transfer mechanisms of thermal energy and momentum in a highly turbulent accelerated flow with large density gradients. This was achieved by designing experiments and numerical simulations to investigate turbulent flow through a nozzle with inlet turbulence intensity values as high as 20%. Results from this work can be utilized in engineering applications with similar flow properties such as designing turbine nozzle vanes in gas turbine engines.

A simple way to approach the turbulence problem is by decomposing the velocity $U_i$ into its mean $\overline{u}_i$ and the fluctuation $u'_i$. The time averaged momentum equations can be written as

$$\rho \overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = \rho \bar{f}_i + \frac{\partial}{\partial x_j} \left[ -\rho \delta_{ij} + \mu \left( \frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right) - \rho u'_i u'_j \right]. \tag{1.1}$$

Equation 1.1 is referred to as Reynolds Averaged Navier Stokes equation (RANS). The time averaged equation, governing heat transfer in turbulent flow of a constant-density fluid can be derived from the thermal diffusion equation assuming no heat source terms as
\[ \frac{u_i}{\overline{u}_i} \frac{\partial \overline{T}}{\partial x_i} = \alpha \left( \frac{\partial^2 \overline{T}}{\partial x_i^2} \right) - \frac{\partial}{\partial x_i} \left( \overline{u'_i T'} \right), \]  

(1.2)

where \( \overline{T} \) is the time averaged temperature and \( \alpha \) is thermal diffusivity.

For a general, three dimensional flow, there are five independent equations governing the mean velocity field. These five equations contain more than five unknowns. In addition to velocity and pressure, the Reynolds stresses \( \rho \overline{u'_i u'_j} \) are not known. Therefore this set of equations is not closed and this is referred to as closure problem in turbulence. Reynolds stress terms can be computed using different models and therefore close the system of equations. Boussinesq [9], introduced the concept of eddy viscosity of momentum to relate the turbulence stresses to the mean flow for the purpose of closing the system of equations as,

\[- \overline{u'_i u'_j} \equiv 2\epsilon_m S_{ij} - \frac{2}{3} K \delta_{ij}, \]  

(1.3)

where \( S_{ij} \) is the mean rate of strain tensor, \( \epsilon_m \) is the turbulence eddy viscosity and \( K \) is the turbulence kinetic energy. Using the eddy diffusivity concept similar to the momentum equation, the unknown term \( \overline{u'_i T'} \), in energy equation can be defined as

\[- \overline{u'_i T'} \equiv \epsilon_h \frac{\partial T}{\partial x_j}, \]  

(1.4)

where \( \epsilon_h \) is the eddy diffusivity of heat. It is considered to be a primary variable of the flow field that needs to be modeled.

Prandtl [51] introduced the concept of turbulent Prandtl number to define a relationship between the eddy diffusivity of heat and eddy viscosity as

\[ Pr_t \equiv \frac{\epsilon_m}{\epsilon_h}. \]  

(1.5)

Turbulent Prandtl number relates the turbulent diffusion of thermal energy to that
of momentum. $Pr_t < 1$ means that the turbulent thermal diffusion is more effective than the turbulent momentum diffusion and vise-versa for $Pr_t > 1$. Turbulent Prandtl number of 1 states that both transport mechanisms are the same which is known as Reynolds analogy and will be discussed in more details in this chapter. $Pr_t$ has wide usage in computational fluid dynamics (CFD) simulations. In CFD, the eddy viscosity is usually modeled as part of a turbulence model and, the eddy diffusivity of heat is calculated based on the eddy viscosity and a user-defined turbulent Prandtl number. The energy equation (1.2) states that the convection of thermal energy in a steady state condition is governed by thermal conduction and turbulent transport. In turbulent flows, it has been observed that the turbulent flux is greater than the thermal conduction often by several orders of magnitude.

A two dimensional turbulent flow as in Figure 1.1 is considered in order to better understand the mechanism of heat transfer in turbulent flow. The curved line shows a general profile of variable $C(y)$ starting from $y = 0$ to $y = a$. $C$ can be interpreted as temperature, stream-wise velocity, or concentration. A differential volume of fluid as shown by gray squared element in Figure 1.1 moves from level $y_1$ to level $y_2$ with a velocity $v_{y_1}$ due to the turbulent motion. It carries with itself the temperature or velocity appropriate to level $y_1$. Another differential volume of fluid moves from level $y_2$ to $y_1$ carrying the corresponding temperature or velocity which is higher than the one of level $y_1$. These processes would be equal on average due to the steady state condition of the flow. A similarity between momentum and heat transfer mechanisms can be implied from the above explanations.

Substituting equation (1.4) in energy equation (1.2) and rearranging the terms, the energy equation can be written as

$$\frac{w_{ij}}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \frac{\partial T}{\partial x_i} + \epsilon_{hk} \frac{\partial T}{\partial x_j} \right].$$

(1.6)

If we consider the special case of $C(y)$ being the the temperature profile in the
2 – D case shown in Figure 1.1, the molecular and turbulent thermal diffusion terms on the right hand side of equation 1.6 are related to the heat flux per unit area per unit time ($q''$) as

$$
\frac{q''}{\rho c_p} = -(\alpha + \epsilon_h) \frac{dT}{dy}, \quad (1.7)
$$

or in high turbulence ($\epsilon_h \gg \alpha$),

$$
\frac{q''}{\rho c_p} \approx -\epsilon_h \frac{dT}{dy}, \quad (1.8)
$$

The equation 1.7 can be simplified for this case as,

$$
\frac{q''}{\alpha \rho c_p} = - \left[ 1 + (\epsilon_h/\alpha) \right] \frac{dT}{dy}. \quad (1.9)
$$

Rearranging the terms in the equation above gives

$$
\frac{q''}{\alpha \rho c_p} = - \left[ 1 + (\epsilon_h/\epsilon_m) (\nu/\alpha) (\epsilon_m/\nu) \right] \frac{dT}{dy}. \quad (1.10)
$$

Since, $Pr = \nu/\alpha$ and $Pr_t = \epsilon_m/\epsilon_h$. 

Figure 1.1. Schematic of Eddy Fluctuations in Turbulent Flow
\[
- \frac{q''}{\alpha \rho c_p} = \left\{ 1 + \left[ Pr^{-1}_f Pr \frac{\epsilon_m}{\nu} \right] \right\} \frac{dT}{dy}.
\] (1.11)

The energy equation is written in the form above to be comparable to the corresponding momentum flux (\(\tau\)) equation, which can be derived in the same manner as the energy equation, given by

\[
\frac{\tau}{\rho \nu} = \left[ 1 + \frac{\epsilon_m}{\nu} \right] \frac{d\bar{u}}{dy}.
\] (1.12)

The details of the derivation for the momentum equation is given by [55]. The similarity between the heat transfer and momentum transfer mechanisms can be identified by comparing equations 1.11 and 1.12. If \(Pr^{-1}_f Pr = 1\), both fluxes are related to the corresponding mean property gradient in a similar manner. Thus, the mechanisms of turbulent heat and momentum transfer are similar as well. This approach is best known as Reynolds analogy [55].

The above describes the general physics of the heat and momentum transfer by turbulent fluctuations. The purpose of this research is to investigate the mechanism of heat transfer in a highly accelerated turbulent flow with large density gradients. The fundamental concept to investigate is the analogy between momentum and heat transfer physical mechanisms. Better understanding of this analogy facilitates more realistic modeling of the turbulent diffusion of thermal energy based on the turbulent diffusion of momentum using the turbulent Prandtl number. The following sections present a more in depth look at the analogies between the heat and momentum transfer mechanisms as well as various turbulent Prandtl number models.
1.2 Analogies between Heat and Momentum Transfer Mechanisms in Turbulent Flows

This section investigates the existing analogies between the heat and momentum transfer mechanisms in turbulent flow. These analogies explain the similarity between the two aforementioned mechanism and how one can be related to the other.

1.2.1 Reynolds Analogy

In a two dimensional statistically steady turbulent flow where the mean velocity is in the $x$ direction, a plane element of unit area remaining stationary within the flow can be considered as shown in Figure 1.2. The fluctuating value of the property to be discussed is denoted by $S + s$ (amount per unit mass of fluid), and the varying flux of this property through the elemental area is $J + j$ (amount per unit area per unit time). The fluctuating component of velocity in the direction $y$ normal to the plane is $V + v$. The fluid density denoted by $\rho$ is assumed to be uniform, and the molecular diffusivity of the property $S$ is denoted by $K$ (amount per unit area per unit time). In terms of these quantities, the instantaneous flux of $S$ is

$$J + j = \rho K \frac{\partial (S + s)}{\partial y} + \rho (S + s) (V + v).$$

(1.13)

The time mean value of the flux of $S$ is then found to be

$$J = -\rho K \frac{\partial S}{\partial y} + \rho SV + \rho Sv.$$

(1.14)

This result shows that the net flux $J$, is the result of:

I Molecular diffusion due to the mean concentration gradient.

II Bulk convection associated with the mean concentration and the mean velocity normal to the unit plane.

III The average transfer affected by turbulent mixing, which will depend on the
degree of concentration between $s$ and $v$, and on their intensities.

The last statement may be formally converted to a diffusion term by modeling an eddy diffusivity

$$\overline{sv} = -\nu_s \frac{\partial S}{\partial y}. \quad (1.15)$$

Furthermore, the effect of turbulent mixing can be represented by an effective lateral convection velocity $V_s$ or by an effective mass flux called Reynolds flux $G_s$, 

$$\rho \overline{sv} = \rho V_s S = G_s \triangle S, \quad (1.16)$$

where $\triangle S$ is the change in $S$ across some finite interval $\triangle y$. Certain parameters commonly used to specify momentum, heat and mass transfers, friction coefficients and Stanton number can be interpreted as scaled forms of the Reynolds flux $G_s$. Thus,

$$\frac{G_s}{\rho U_0} = \text{dimensionless transfer coefficient}, \quad (1.17)$$
where $\rho$ is the uniform density of the fluid, and $U_0$ is a characteristic mean velocity.

For heat transfer, $\Delta S = \Delta H$, where $H$ is enthalpy, the Reynolds enthalpy flux is given by

$$q_h = G_h \Delta H,$$

(1.18)

in terms of Stanton number, the equation above can be rewritten as

$$St = \frac{q_h}{\rho U_0 \Delta H} = \frac{G_h}{\rho U_0}.$$  (1.19)

Also the Reynolds momentum flux is given by

$$C_f = \frac{\tau}{\frac{1}{2} \rho U_0^2} = 2 \frac{G_m}{\rho U_0},$$  (1.20)

where, $C_f$ is friction coefficient and $\tau_w$ is shear stress at the wall. Reynolds theorized that in a wall bounded flow only skin friction is present, therefore this analogy cannot be applied where liquids or drag forces are present.

The relationships 1.19 and 1.20 lead to a much-used analogy between momentum and heat transfer. Combined, they give

$$2 \frac{G_m}{C_f} = \frac{G_h}{St} = \rho U_0,$$  (1.21)

or

$$\frac{\tau}{U_0} = \frac{q}{\Delta H} = G_m.$$  (1.22)

This series of results are commonly referred to as Reynolds analogy. Provided that the mean distribution of $U$ and $H$ are similar as

$$\frac{1}{\Delta H} \frac{\partial H}{\partial y} = \frac{1}{U_0} \frac{U}{\partial y},$$  (1.23)
Substituting the above in equation 1.15 gives

\[ \frac{\epsilon_h}{\epsilon_m} = \frac{G_h}{G_m}, \]  

(1.24)

which indicates that the two diffusivities (heat and momentum) are equal when the two Reynolds fluxes are equal. This implies that \( Pr_t \equiv \frac{\epsilon_h}{\epsilon_m} = 1 \). This simple way of relating diffusive and stress-generating capacities is a direct result of the Reynolds analogy.

There are two main vital criticisms to this analogy. First, in most situations, the relationship between the eddy diffusivities of heat and momentum changes through the domain. For example in a wall bounded flow a constant turbulent Prandtl number is not valid. Also, in most cases of practical interest, diffusion through an essentially non-turbulent region (mostly in wall layer) plays an important part in the overall turbulent transfer. Second, when the molecular Prandtl number is much different than unity (for instance in liquid metals), Reynolds analogy would be dramatically wrong.

1.2.2 Strong Form of Reynolds Analogy

The requirement for validity of Reynolds analogy from section 1.2.1 can be written as

\[ \frac{1}{\Delta H} \frac{\partial H}{\partial y} = \frac{1}{U_0} \frac{\partial U}{\partial y}. \]  

(1.25)

For a parallel mean flow \( U(y) \), the two fluxes are given by

\[ \rho (\nu + \epsilon_m) \frac{dU}{dy} = G_m U_0, \]  

(1.26)

and
\[- \rho (\alpha + \epsilon_h) \frac{dH}{dy} = G_h \Delta H. \tag{1.27}\]

With the assumption that the mean property profiles are similar, it can be found (see [55]) that

\[
\frac{G_m}{G_h} = \frac{\nu + \epsilon_m}{\alpha + \epsilon_h} \equiv Pr_e, \tag{1.28}\]

where \(Pr_e\) is the effective Prandtl number. It takes the molecular diffusivities into account as well. The strong Reynolds analogy can be written as

\[
G_m = Pr_e G_h. \tag{1.29}\]

Assuming \(Pr_e = 1\), the above equation simplifies to Reynolds analogy

\[
G_m = G_h. \tag{1.30}\]

1.2.3 Prandtl-Taylor Analogy

The Reynolds analogy assumes that the mechanism of heat and momentum transfer and the analogy among them does not vary based on the location in the flow. For example, based on the Reynolds analogy, the similarity between heat and momentum transfer is the same close to the walls or away from them in the core flow. Therefore, the value of the turbulent Prandtl number stays constant throughout the flow regardless of its distance from solid walls. Prandtl and Taylor in separate attempts presented a more elaborate analogy, which allowed a pseudo laminar layer for two turbulent regimes. The thermal diffusion sublayer extends to a distance \(d_h\) from the solid wall. The heat transfer is effectively controlled by molecular diffusion closer to the wall and by turbulence diffusion as the distance from wall increases. If the thermal diffusion sublayer \(d_h\) coincides with the distance \(d_l\) of laminar sublayer from the
solid wall, the heat transfer in this thin layer (by conduction alone) is easily obtained in the form of [19]

$$ q'' = -\alpha \rho c_p \frac{dT}{dy} \approx -\alpha \rho c_p \frac{(T_{di} - T_0)}{d_t}. \quad (1.31) $$

Also based on the definition of viscosity, for this layer the shear stress is

$$ \tau_0 = \frac{\mu U_{di}}{d_t}. \quad (1.32) $$

Replacing the above into equation 1.31

$$ \frac{q''}{\tau_0} = -\left( \frac{\alpha \rho c_p}{\mu} \right) \left( \frac{T_{di} - T_0}{U_{di}} \right) = - \left( \frac{c_p}{Pr} \right) \left( \frac{T_{di} - T_0}{U_{di}} \right). \quad (1.33) $$

In turbulence zone, assuming that the Reynolds analogy can be applied from the core flow up to the edge of the laminar sublayer (assuming that \( d_h = d_t \))

$$ \frac{q''}{\tau_0} = -c_p Pr_t^{-1} \left( \frac{T_m - T_{di}}{U_m - U_{di}} \right). \quad (1.34) $$

By eliminating \( T_{di} \) between equations 1.33 and 1.34 putting \( T_m - T_0 = \Delta T \) and rearranging the terms

$$ \frac{-q''}{c_p \Delta T} \left[ 1 + \left( \frac{U_{di}}{U_m} \right) \left( Pr_t^{-1} Pr - 1 \right) \right] = Pr_t^{-1} \frac{\tau_0}{U_m}. \quad (1.35) $$

It can also be shown that [19]

$$ \frac{U_{di}}{U_m} = 5 \left( \frac{C_f}{2} \right)^{0.5}. \quad (1.36) $$

Therefore the equation 1.35 becomes

$$ \frac{-q''}{c_p \Delta T} Pr_t \left[ 1 + 5 \left( \frac{C_f}{2} \right)^{0.5} (Pr_t^{-1} Pr - 1) \right] = \frac{\tau_0}{U_m}. \quad (1.37) $$
Thus, the Reynolds fluxes are

$$Pr_t \left[1 + 5 \left(\frac{C_f}{2}\right)^{0.5} (Pr_t^{-1}Pr - 1)\right] = G_h = G_m. \quad (1.38)$$

This equation, allowing for a viscous region near the wall, is more accurate than the simple Reynolds analogy, to which it reduces in the special case of $Pr_t^{-1}Pr = 1$. Prandtl specifically focused on the Stanton number denoted by $St$, to investigate the heat transfer in pipes. It is an alternative heat transfer coefficient which gives a measure of the ratio of the heat transfer coefficient $h$ to the flow of heat along the pipe due to the velocity and heat capacity of the fluid. Stanton number can be written based on the Prandtl-Taylor analogy as

$$St = Pr_t \frac{C_f}{2} \left[1 + 5 \left(\frac{C_f}{2}\right)^{0.5} (Pr_t^{-1}Pr - 1)\right]^{-1}. \quad (1.39)$$

1.2.4 Von-Karman Analogy

Von Karman extended Prandtl’s work by taking the effect of a buffer zone between the laminar sublayer and the turbulent core into account. The result (see [13] for the details of derivation), is given by

$$St = \frac{C_f}{2} \left[1 + 5 \left(\frac{C_f}{2}\right)^{0.5} \left\{(Pr_tPr - 1) + \ln \left[\frac{5Pr + 1}{6}\right]\right\}\right]^{-1}. \quad (1.40)$$

Von Karman’s analogy, reduces to Reynolds analogy for gases with $Pr$ and $Sc$(Schmidt) numbers equal to one. Schmidt number ($Sc$) is a dimensionless number defined as the ratio of momentum diffusivity and mass diffusivity,

$$Sc = \frac{\nu}{D}, \quad (1.41)$$

where $D$ is the mass diffusivity.
1.2.5 Chilton-Colburn Analogy

While Prandtl and Von Karman adapted the Reynolds analogy by considering the transfer equations in the boundary layer, Chilton and Colburn\[13\] sought modifications to the Reynolds analogy using experimental data. They suggested a simple modification for situations where $Pr$ and $Sc$ numbers are different from unity. This analogy defines an empirical factor

$$j_h = \left( \frac{C_f}{2} \right) = \frac{h}{\rho c_p U} Pr^{\frac{2}{3}}. \quad (1.42)$$

If Prandtl number in the above equation is set to unity, it reduces to Reynolds analogy. The Stanton number can be utilized to simplify the equation given above. It is defined by

$$St = \frac{Nu}{RePr} = \frac{h}{\rho c_p U}. \quad (1.43)$$

By replacing the above in the equation $^{1.42}$ the Chilton-Colburn analogy can be written as

$$j_h = \frac{C_f}{2} = StPr^{\frac{2}{3}}. \quad (1.44)$$

Therefore the Stanton number can be redefined as

$$St = \left( \frac{C_f}{2} \right)^{Pr^{\frac{2}{3}}}. \quad (1.45)$$

If the friction factor is known, the Stanton number can be calculated. The Nusselt number can be calculated accordingly. The $j$-factor for mass transfer can be written based on the Chilton-Colburn analogy as

$$j_D = \frac{k_c}{U_\infty} Sc^{2/3}. \quad (1.46)$$
This analogy can be summarized as

$$\frac{C_f}{2} = j_D = j_H. \quad (1.47)$$

It can be inferred from this equation that the Chilton-Colburn analogy reduces to Reynolds analogy in special case of $Pr = 1$.

1.3 Turbulent Prandtl Number Models

Many researches have studied the Turbulent Prandtl number using both experimental and analytical approaches. These studies mainly focused on investigating the turbulent Prandtl number close to solid boundaries. The first to propose a more detailed analytical modification to the Reynolds analogy was Jenkins[26]. He considered the motion of a spherical element of fluid across a turbulent flow. The element would lose thermal energy by simple conduction and its surface temperature would vary linearly with the time of motion during its movement. He derived an equation for $\epsilon_h$ by using the formula for the average temperature of a sphere, and combined it with an expression for $\epsilon_m$ to propose an analytical turbulent Prandtl number model given by

$$Pr_t = \frac{1}{Pr} \frac{1 - (90/\pi^2) \sum_{n=1}^{\infty} n^{-4} \exp \left( -n^2 \pi^2 l_k/a^2 v' \right)}{1 - (90/\pi^2) \sum_{n=1}^{\infty} n^{-4} \exp \left( -n^2 \pi^2 l_k/a^2 v' \right)}, \quad (1.48)$$

where $a$ is the radius of the sphere equal to the mixing length $l$, and $v'$ is the lateral fluctuation in the velocity. In this model, none of the unknown parameters can be defined or adjusted to match experiments. The turbulent Prandtl number values predicted by this model were much higher than the experimental measured values. Sleicher and Tribes[62] simply modified the model proposed by Jenkins as

$$Pr_t(Pr) = \frac{Pr_t(air,\text{experimental})}{Pr_t(air,Jenkins)} \times Pr_t(Pr,Jenkins). \quad (1.49)$$
Predictions based on the above equation highly depend on the experimental data sets implemented in it thus, lacking any universality.

Rohsenow and Cohen[57] modified The model proposed by Jenkins in a more fundamental way by arguing that the eddy diffusivity of momentum was negligible comparing to kinematic viscosity. Hence, they proposed a model that only depended on the molecular Prandtl number. It can be written as

\[
Pr_t = 27.8Pr \left[ 1 - \left( \frac{90}{\pi^2} \right) \sum_{n=1}^{\infty} n^{-4} \exp \left(-Cn^2\pi^2/Pr\right) \right].
\] (1.50)

with \( C = 0.0024 \).

In another approach, Deissler[20] made the same assumption as Jenkins. A sphere moving normal to the mean flow, and taking into account the heat transfer from the moving element. The model is somewhat less explicit than Jenkins’ but, like it, takes the sphere radius to be proportional to the mixing length, and assumes molecular diffusion. Deissler’s model defers fundamentally from Jenkins in taking the transfer to and from the sphere to be controlled by diffusive processes external to the moving element, rather than within it. It can be written as

\[
Pr_t^{-1} = bPe \left[ 1 - \exp \left(-1/bPe\right) \right],
\] (1.51)

where \( Pe \) is mean flow Peclet number given by

\[
Pe = \frac{LU}{\alpha}.
\] (1.52)

Here \( L \) is a characteristic length and \( U \) is flow velocity, \( b \) is a constant in equation 1.51 which can be defined using the experimental data. Deissler calculated the value as \( b = 0.000153 \) by utilizing experimental data specific to his case of study. It is worth noting that Deissler made the assumption that the mean flow Peclet number is proportional to the Peclet number of the moving element.
Lykoudis and Touloukian\cite{11} modified the model proposed by Deissler, by choosing the transfer time in such a way that the length scale of the motion did not influence the small scale transfers. Their model is given by

\[
Pr_t^{-1} = \frac{6}{\pi^2} \left( \sum_{n=1}^{\infty} n^{-2} \exp \left( -C n^2 / Pr \right) \right),
\]

(1.53)

where \( C = 0.01 \). Aoki\cite{3} also modified Deissler’s analysis by taking into account for small scale turbulence mixing by introducing a transfer law resulting into

\[
Pr_t^{-1} = K Re^{0.45} Pr^{0.2} \left[ 1 - \exp \left( \frac{-1}{K Re^{0.45} Pr^{0.2}} \right) \right].
\]

(1.54)

Where \( K \) is a constant based on the type of the flow. In case of the pipe flow \( K = 0.014 \).

An additional argument to consider a more sophisticated picture of turbulent activity can be achieved by allowing a variability in the distance and directions traveled by the moving fluid elements, and not explicitly assuming instantaneous mixing with the terminal surroundings. Azer and Chao\cite{4} gave a model for turbulent pipe flow as,

\[
Pr_t = \frac{1 + 57f(y/R) / (Re^{0.46} Pr^{0.58})}{1 + 135f(y/R) / Re^{0.45}}.
\]

(1.55)

The variation across the pipe is given by

\[
f(y/R) = \exp \left[ - \left( y/R \right)^{1/2} \right].
\]

(1.56)

Tyldesley and Silver\cite{71}, considered a more general fluid element. They introduced distortion factors into the drag and transfer equations to derive their proposed model as
\[ Pr_t = 2/3 + 2 \left( \psi / \psi' \right) / (9Pr) \]  \hspace{1cm} (1.57)

Where \( \psi \) and \( \psi' \) are the distortion factors. For a sphere \( \psi / \psi' = 1 \).

In order to evaluate turbulent Prandtl number from experimental measurements, one needs to obtain four quantities in the flow: the turbulent shear stress, the turbulent heat flux, the velocity gradient, and the temperature gradient as follows

\[ Pr_t \equiv \frac{\overline{u'v'} \frac{\partial T}{\partial y}}{\overline{T'v'} \frac{\partial u}{\partial y}} \]  \hspace{1cm} (1.58)

The difficulty of such measurements limits the experimental work in this field however, attempts for such analysis have been done. These measurements were mostly obtained in the wall boundary layers. Kays and Crawford\[31\] suggested the following formulation

\[ Pr_t = \frac{1}{0.5882 + 0.228 \left( \frac{\nu_m}{\nu} \right) - 0.0441 \left( \frac{\nu_m}{\nu} \right)^2 \left[ 1 - \exp \left( \frac{-5.165}{\left( \frac{\nu_m}{\nu} \right)} \right) \right]} \]  \hspace{1cm} (1.59)

Wassel and Catton\[73\] have proposed an expression which also depended on \( \frac{\nu_m}{\nu} \) as

\[ Pr_t = \frac{C_3 \left[ 1 - \exp \left( -C_4 \frac{\nu_m}{\nu} \right) \right]}{C_1 \left[ 1 - \exp \left( -C_2 \frac{\nu_m}{\nu} \right) \right]} \]  \hspace{1cm} (1.60)

Blom\[8\] presented an expression which depended on the wall velocity \( u^+ \). Having \( x = 0.4u^+ \), his proposed model is given by

\[ Pr_t = 1 - \frac{x^4}{e^x - 1 - x - \frac{x^2}{2} - \frac{x^3}{6}} \]  \hspace{1cm} (1.61)

Another simple model was introduced by Kays\[31\] which can be written as
Turbulent Prandtl number in this model, is a function of location in the boundary layer and holds a value of 0.9 outside the boundary layer. Thomas\textsuperscript{69} introduced a model which depended on the wall velocities as well. This model held a constant $Pr_t = 0.7$ outside the boundary layer and $Pr_t = 1$ for $y^+ < 5$. The turbulent Prandtl number proposed by Thomas is given by

$$Pr_t = Pr + \left(\frac{\nu/\epsilon_m}{\nu} \right) \exp \left[ - \left(0.5PrC_f \right)^2 y^+ \right] \frac{1}{1 + \left(\nu + \epsilon_m \right) \exp \left[ - \left(0.5C_f \right)^2 y^+ \right]}.$$  \hspace{1cm} (1.63)

Other models have been introduced in the literature for evaluating the turbulent Prandtl number which have been briefly reviewed in the course of this research.

1.4 Research Objectives

The main objective of this research was to quantify the turbulent Prandtl number in an accelerated nozzle flow with high inlet turbulence intensity and high density gradients. Turbulent Prandtl number defines the relationship between the mechanisms of turbulent transport of momentum and heat in this problem. This objective was met by designing and building an experimental facility capable of providing the required boundary conditions. In order to acquire the properties of fluid flow through the experimental setup, a novel hotwire calibration technique was developed. This method allowed obtaining hotwire data in varying ambient flow temperatures with a single calibration at a constant temperature. The experimental measurements were utilized as boundary conditions in a computational study with identical geometry as the experiment. Performance of several turbulence models and turbulent Prandtl number models were investigated to achieve a numerical solution which matched the experimental measurements. The value of turbulent Prandtl number in the experi-
ment was defined through these analysis thus, defining the analytical analogy between the turbulent transports of thermal energy and momentum.

Given the computational results, it was of interest to define a second objective. The second objective was to develop a general turbulent Prandtl number model where known inputs from the flow field can provide an estimate of the turbulent Prandtl number. Several models were developed and will be presented that provided accurate estimation of the turbulent Prandtl number compared to the values acquired in the first objective as well as values from various independent experimental and numerical measurements of $Pr_t$.

In summary, the main objectives of this research were:

- Quantifying the turbulent Prandtl number in a highly accelerated flow with high turbulence intensity values and density gradient.

- Develop a model capable of estimating turbulent Prandtl number in highly accelerated flow with high turbulence intensity values and density gradient as well as other flow types.
CHAPTER 2

EXPERIMENTAL FACILITY

The experiments for this study were conducted in the Hot Annular Nozzle Flow facility. This facility was designed to provide air flow with specific boundary conditions and properties needed for better understanding of the detailed flow physics around cooled and uncooled nozzle vanes in the first stage turbine nozzle of gas turbine engines. The following sections will highlight the important features of the research facility, as well as explain the experimental setup and methods used in this study.

2.1 Annular Nozzle Flow Facility

A schematic of the annular nozzle flow facility is shown in Figure 2.1. Air is drawn from the atmosphere through the inlet by two 250 hp variable speed compressors arranged in series. These provide a net total pressure ratio of 1.23 to the test section. Downstream of the test section, a portion of the air is exhausted to the atmosphere using two exit valves. Majority of the air is recycled by adjusting the recirculation valve and the exit valves. Recirculating the air allowed the test section inlet temperatures in excess of 650 K. The facility is capable of operating continuously at this temperature.
Figure 2.1. Schematic of Annular Nozzle Flow Facility

Various size test sections including cooled and uncooled turbine vanes can be incorporated in the hot annular nozzle flow facility. The test section used in this study was an annular contraction. Figure 2.2 shows the schematic of this test section. High temperature air entered the test section. Air at slightly higher than room temperature (320 K), was injected into the annulus using two 50 hp auxiliary blowers through multiple openings denoted in Figure 2.2 by OD/ID cooling flow path. This allowed the control of span-wise distribution of the inlet total temperature. Closing the OD/ID cooling flow paths provided a nominally flat temperature distribution through the test section. Figure 2.3 shows the circumferentially averaged inlet profiles examined in this study.
Figure 2.2. Schematic of the Annular Contraction in Annular Nozzle Flow Facility

Total temperature \( T_t \) was normalized by the maximum average total temperature in the midspan \( T_{t_{\text{max}}} \) as

\[
\Theta_t = \frac{T_t}{T_{t_{\text{max}}}}.
\]  
(2.1)

Total pressure \( P_t \) was normalized by the maximum measured average total pressure \( P_{t_{\text{max}}} \) as

\[
\Theta_t = \frac{P_t}{P_{t_{\text{max}}}}.
\]  
(2.2)

The radial location was normalized as

\[
\eta = \frac{r}{r_{OD} - r_{ID}},
\]  
(2.3)

where \( r_{OD} \) and \( r_{ID} \), were the radius of the outer wall and the inner wall, respectively. Thus, \( r \) was normalized by the radial span of the considered plane. Figure 2.3a shows
Figure 2.3. Inlet conditions of the considered cases, (a) Circumferential averaged total temperature, (b) Circumferential averaged total pressure.

The circumferentially averaged total temperature profiles for the uniform and non-uniform inlet total temperature distribution cases. The uniform case was achieved by closing the OD/ID cooling flow paths and recirculating the hot air through the experimental setup. The total temperature was mostly constant through the span of the inlet measurement plane. The temperature slightly decreased close to the OD wall due to the heat transfer from the test section walls to the ambient. Total $\Theta_t$ had a value of 1 for $0.36 \leq \eta \leq 0.64$ in the non-uniform distribution case. It decreased outside the aforementioned range of normalized radius. The total temperature close to both OD and ID was controlled to be 10% lower than the total temperature in the mid-span.

Circumferentially averaged total pressure profiles are shown in Figure 2.3a. Both profiles have a local minimum in the mid-span. The total pressure value increased moving away from the mid-span toward the edges of the span. The maximum value for both profiles occurred close to the OD and ID wall. Overall, the total pressure profiles were similar in both cases considered in this study. The details of these measurements are discussed further in the chapter. 

23
The air was then driven through a multi scale turbulence grid. The grid introduced non-uniformity in the velocity and the total pressure distributions while increasing the level of inlet turbulence. Multi-scale turbulence generation grids are passive means of turbulence generation with non-homogeneous mesh geometry. Turbulence is generated at multiple scales and high turbulence intensity values by these grids, which resemble turbulence in naturally occurring phenomena. Ranade [52] studied various grid geometries to choose the appropriate multi-scale grid for this facility.

A schematic of the cross-grid scaled for use in the annulus is shown in figure 2.4. The turbulence grid had two different sizes of radial and circumferential bars. The grid was located at 14 cm upstream of the first inlet measurement plane and was capable of producing 18% turbulence intensity on average.

![Figure 2.4. Schematic of 90 degrees section of the turbulence grid](image)

2.2 Facility Instrumentation

Four total pressure ($P_t$) and total temperature ($T_t$) rakes were installed upstream of the test section. Each rake consisted of nine pairs of total pressure and total temperature probes. The rakes were positioned 90° apart. The turbulence grid had a
periodicity of $10^\circ$ thus, the relative positions of the rakes to the local periodic sector of
the grid were identical. Comparing the measured values of the rakes provided a mea-
sure of the circumferential uniformity of the total pressure and the total temperature
distributions in the inlet.

A traverse mechanism was designed to provide radial and circumferential move-
ments for traversing total pressure, total temperature and hot wire probes at the
measurement planes. It also had the capability of rotating the probe along its axis
(yaw). The measurement planes locations are shown in Figure 2.2. The traverse
mechanism was capable of traversing a radial distance of 16 cm, and had a circum-
ferential range of 25 degrees.

2.2.1 Total Pressure Measurements

Total pressure Kiel probe used for measurements in annular nozzle flow facility
is shown in Figure 2.5. The probe body was made from type 304 stainless steel
tubing with diameter of 6.4 mm and a length of 45 cm in order to extend through the
traverse path and traverse mechanism. All the probes were made with a 1.27 cm sting
so that the measurement plane would be located in the upstream of the straight probe
body and the traverse plane. This was chosen to ensure that the measurements were
obtained far away enough from the traverse location so that the flow was not affected
by the traverse mechanism geometry or the possible small leakages at the traverse
location. It is shown that Kiel probes are generally insensitive to the Reynolds
number except at extremely low velocities. Therefore, Keil probes are suitable for
total pressure measurements in Mach numbers ranged from 0.08 to 0.5 at 644 K.

The Kiel probe was used to survey the inlet measurement plane in a domain that
ranged circumferentially from 260 to 280, and radially from 3% to 97% span of the
flow path. The same probe was used to obtain total pressure measurement at the
exit measurement plane in a domain that ranged circumferentially from 260 to 280,
and radially from 14% to 86% span of the flow path. The Kiel probes were shown to be ineffective close to the solid walls thus, no measurements were acquired outside the radial ranges defined above. Total pressure probe was connected to a Netscanner pressure transducer system. The measurements were sampled at 50 Hz for 7 seconds at each traverse point.

2.2.2 Total Temperature Measurements

Total temperature Kiel probe was designed and made in house to meet the specific conditions of annular nozzle flow facility as shown in Figure 2.6. The probe body was made from type 304 stainless steel to the same measurements as total pressure probe. The traverse domain and resolution were chosen similar to the total pressure Kiel probe. Total temperature probe was connected to National instruments high accuracy thermocouple module model 9214 and was sampled at 50 Hz for 7 seconds.

The measured junction temperature by the total temperature Kiel probe is related to the flow total temperature as
\[ T_j = T_s + r(T_t - T_s), \quad (2.4) \]

where \( r \) is the recovery factor, \( T_j \) is the junction temperature, \( T_t \) is the total temperature, and \( T_s \) is static temperature. Thus, the recovery factor is defined as,

\[ r = \frac{T_j - T_s}{T_t - T_s}. \quad (2.5) \]

The recovery factor versus free-stream Mach number is plotted in Figure 2.7 for the total temperature probe used in this facility. This was achieved by calibrating the total temperature probe in a calibration jet with known total temperature. The Mach number of the jet was varied over the range of interest. Total temperature values measured by probe and the calibration jet instrumentation were recorded to calculate the recovery factor.

![Figure 2.7. Total Temperature Recovery Factor Calibration](image)

Data obtained in the exit measurement plane was corrected using the probe recovery factor calibration and the downstream Mach number.
\[ \frac{T_j}{T_i} = r + \frac{T_s}{T_i} (1 - r) \] (2.6)

### 2.2.3 Standard Day Correction

Data acquired in the annular nozzle flow facility consisted of measurements obtained in the upstream and downstream of the nozzle by a variety of probes in different conditions. Comparisons between data sets not acquired simultaneously have the potential to introduce measurement biases due to the facility operating conditions. Small changes in these conditions can occur over the course of a single test as well as day-to-day operation. Such small changes in the operating point can be corrected for. This correction method is described below.

The standard day correction for total temperature is given by

\[ T_{t,cor} (r_i, \theta_i) = \left[ \frac{T_t (r_i, \theta_i)}{T_{rake,i}} \right] T_{STD}, \] (2.7)

where \( T_t (r_i, \theta_i) \) is the raw total temperature value, \( T_{rake,i} \) is the average total temperature as measured by the inlet rakes, \( T_{STD} \) is the standard day total temperature, and \( i \) is the traverse index. This correction procedure was applied to the measurements obtained at all traverse planes. The inlet rake total temperature was the arithmetic mean of the center three rake total temperature probes at \( \theta = 225^\circ \) and \( \theta = 315^\circ \).

The standard day correction for total pressure was done in the same manner as the total temperature, given by

\[ P_{t,cor} (r_i, \theta_i) = \left[ \frac{P_t (r_i, \theta_i)}{P_{rake,i}} \right] P_{STD}, \] (2.8)

where \( P_t (r_i, \theta_i) \) is the raw total pressure value, \( P_{rake,i} \) is the average total pressure as measured by the inlet rakes, \( P_{STD} \) is the standard day total pressure, and \( i \) is the traverse index. This correction procedure was applied to the measurements obtained
at all traverse planes. The inlet rake total pressure was the arithmetic mean of the center three rake total pressure probes at $\theta = 225^\circ$ and $\theta = 315^\circ$. 
CHAPTER 3

HOTWIRE MEASUREMENTS

Hot-wire anemometry is a well established technique that is used to obtain time resolved measurements of velocity \cite{39,48}. The hot-wire sensors are thin metallic elements, heated by an electric current and cooled by the incident flow. The heat transfer from the wire occurs through various effects in which the forced convection is the most predominant. Hence, the measurement is sensitive to both velocity and to the temperature of the flow. If the hot-wire sensor is used in a flow with an identical mean temperature as its calibration temperature, the uncertainty due to changes in the ambient temperature is minimized. There are standard methods for correcting small temperature changes in the flow for constant temperature anemometer (CTA) hot-wire systems, see \cite{6,7,39}. On the other hand, if the sensor is used in a flow with a different mean temperature than its calibration temperature, decoupling the temperature dependence and the flow velocity can be challenging, particularly when the Mach number is in the high-subsonic range. In this chapter, calibrating and using hot-wires in relatively high temperature, high Mach number flows with large temperature gradients will be discussed.

The energy balance for the hot-wire is given by \cite{45},

\begin{equation}
\text{electrical input power} = \text{convection to fluid} + \text{conduction to prongs.}
\end{equation}

The electrical power supplied to the sensor is measured as proportional to the
square of the anemometer output voltage. The convective heat transfer is related to the fluid Nusselt number. The conductive heat transfer from the wire is also a function of the fluid velocity. In steady state operation, the conduction effects are accounted for in the calibration which relates the overall cooling effects to the fluid velocity. In the CTA mode of operation the wire resistance is held constant by the anemometer feedback control system. Therefore, the energy balance given by equation 3.1 can be written as

\[
\frac{E_{in}^2}{R_w} = h A_w (T_w - T_x) = \frac{k}{d} A_w (Nu) (T_w - T_x),
\]

where \(E_{in}\) is the voltage across the sensor wire which is proportional to the output voltage of the anemometer \(E_m\), \(R_w\) is the wire resistance, \(h\) is the convective heat transfer coefficient, \(d\) is the wire diameter, \(A_w\) is the surface area of the wire, \(T_w\) is the wire temperature, \(T_x\) is the ambient flow temperature, \(k\) is the thermal conductivity of fluid, and \(Nu = hd/k\) is the Nusselt number. The Nusselt number for forced convection from the hotwire sensor in the subsonic and high-subsonic range depends on the Reynolds number \(Re\), Prandtl number \(Pr\) and, Mach number \(Ma\) (see Comte [18] and, Spangenberg [66]). However, calibrating the hot-wire sensor in the range of interest for Mach number reduces the uncertainty from the Mach number dependency. Taking into account that Prandtl number is a weak function of temperature, it can be assumed that \(Nu = Nu(Re)\), which denotes a functional dependency on Reynolds number.

King [32] suggested an empirical correlation for Nusselt number for infinite cylinders in flow which was used extensively in hot-wire anemometry as

\[
Nu = A + B (Re)^{0.5},
\]

where \(A\) and \(B\) are calibration constants. Collis and Williams [17] suggested that the
exponent of Reynolds number in the above correlation can be considered as a third calibration constant $n$, because the power of 0.5 is not suitable for all measurements. This can be written as

$$Nu = A + B (Re)^n.$$  \hfill (3.4)

As mentioned, it is essential to minimize the errors due to ambient fluid temperature changes in hot-wire calibration and application. Most of the existing correction factors are based on empirical correlation for the forced convective heat transfer around a cylinder introduced above. Bearman [6] introduced the most common temperature compensation factor which is given by

$$E_B = \left( \frac{T_w - T_c}{T_w - T_x} \right)^{1/2} E_m,$$  \hfill (3.5)

where $T_c$ is the calibration temperature. This correction factor is limited to small temperature changes, so that the fluid properties remain constant. Fluid properties are usually calculated at an average temperature between the ambient and wire temperature called the film temperature, $T_f = \frac{T_w + T_x}{2}$. Collis and Williams [17] show that, although the fluid properties are calculated at the film “temperature”, a parameter representing the temperature loading of the heated body is needed to better estimate the fluid temperature over the wire. The corrected voltage output suggested by Collis and Williams is

$$E_{CW}^2 = \left( \frac{T_f}{T_c} \right)^{-0.17} E_m^2.$$  \hfill (3.6)

They also emphasize that it is important to account for changes in the fluid properties such as thermal conductivity due to the temperature changes. They used the relationship given by Kannuluik and Carman [29] for calculating the thermal conductivity of air. Tables of experimentally measured values kinematic viscosity
introduced by Goldstein [22] can be used as well.

Hultmark [25] introduced a method to correct temperature drifts in the range of 15 K. This method is not directly based on empirical heat transfer correlations as it keeps the Nusselt number as an unknown function of the Reynolds number, hence the energy balance is

$$\frac{E_{m}^{2}}{R_{w}} = \frac{k}{d} Nu(Re) A(T_{w} - T_{x}).$$

(3.7)

Therefore, the velocity can be found as

$$\frac{U}{\nu} = f_{H} \left( \frac{E_{m}^{2}}{k(T_{w} - T_{x})} \right).$$

(3.8)

Kinematic viscosity $\nu$ and thermal conductivity, $k$ can be found as functions of temperature from the relationships given by Smits [63] and Kannuluik [28], respectively. A fourth order polynomial would be fitted to the calibration data to find the function $f_{H}$, as suggested by Perry [48]. It should be noted that in the proposed form of equation (3.8), the function $f_{H}$ is still restricted to the calibration temperature, and cannot be used for relatively large temperature fluctuations. According to Hultmark, calibrations obtained by this method at a certain temperature can be applied to flows with temperature changes of up to 15 K. However, their proposed method was not applicable to the 70 K range of temperature changes in this study.

Many of the existing temperature correction correlations such as the ones discussed here, are applicable to small to moderate variations in temperature [11]. However, Benjamin [4] proposed a temperature compensation method which can be applied to 200$K$ temperature change at very low Mach numbers ( $Ma < 0.03$ ).

Benjamin’s method of correction can be written as

$$E_{CB}^{2} = E_{m}^{2} \left( \frac{T_{w} - T_{c}}{T_{w} - T_{x}} \right)^{1.1}. \quad (3.9)$$
This correction factor has a different exponent for the non-dimensional temperature ratio compared to the correction factor proposed by Bearman (see equation (3.5)). They have shown that this small change in the exponent reduces the calibration uncertainty significantly in large temperature changes.

Mach number effects are mostly overlooked in evaluation of the aforementioned correction factors. Spangenberg [66] concluded that heat transfer from a hot-wire sensor has a Mach number dependency that is functionally different from its dependency on Reynolds number. He argued that of the three principal parameters, Mach number \((Ma)\), density \((\rho)\), and wire diameter \((d_w)\), hot-wires respond differently to changes in \(Ma\) from the way they do to changes in \(\rho\) or \(d_w\). This is true at lower speeds where a dependence on \(Ma\) amounts to dependence on velocity. The response to change in \(\rho\), however, was shown by him to be similar to the response to change in \(d\). He stated that the the \(\rho d\) product plays a separate role apart from being a constituent of Reynolds number. Thus, he used Knudsen number \((Kn = \lambda/d_w)\), where \(\lambda\) is the mean free path for the fluid over the wire, as a separate parameter representing the ratio of Reynolds number to the Mach number in order to investigate the dependency of Nusselt number on Mach number and Reynolds number. He showed that Mach number dependency is insignificant for \(Kn < 0.15\). This effect needs to be addressed more rigorously for using a single temperature calibration in a wide range of varying temperatures.

The work reported here was prompted by the need to measure velocity and turbulence characteristics of flow in an annular nozzle with a temperature profile that ranged from 580 K to 644 K. The temperature changes in this experiment were much larger than a few degrees of ambient temperature drift investigated in most of the literature. Also, the Mach number range of the flow during the measurements varied from 0.01 to 0.5, which adds to the complexity of the experimental measurements.

The ability of the existing methods, referred above, to account for temperature
changes were explored and found to be insufficient. An alternative method, using a collective mixture of the existing correction factors is introduced in this chapter. This method can be used to utilize a calibration obtained at a single temperature to be applied to the range of interest of temperatures. Furthermore, the design and manufacturing process of a hotwire probe which withstood the high Mach numbers and temperatures of the experiment is discussed. Measurements demanded the sensor wire to survive in a flow with the temperature of 644 K and Mach numbers as high as 0.9 for long periods of time. Therefore, a novel method of attaching the sensor wire to the hot-wire prongs was developed and is discussed in the following section.

3.1 Hotwire Probe Manufacturing

Hotwire Probes were designed and manufactured to operate in high temperature (from 583 K to 645 K) and high Mach number (from 0.05 to 0.9) flows. Figure 3.1 shows the schematic of a hotwire probe used in this work.

Figure 3.1. Schematic of Hotwire Probe

The probe body consisted of an outer stainless steel cover with a diameter of 6.35 mm. The outer shell stepped down to 4.7 mm in diameter close to the probe tip
in order to reduce the probe body interference with the flow. Two-bore ceramic tubing was closely fitted into the inner sleeve and further secured by Resbond\textsuperscript{TM}907GF high temperature adhesive (see table 3.1 for detailed characteristics of the adhesive materials). Two bare copper wires with the diameter of 1.02 mm were extended inside the two bores and were attached to the bases of the prongs using high temperature electrically conductive silver based paste (Pyro-Duct\textsuperscript{TM}597-A, see table 3.1). Temperature variations can change the electrical resistance of the probe significantly, thus the diameter of the copper wire was chosen as large as possible to minimize the uncertainties due to this temperature effect.

\begin{table}[!h]
\centering
\caption{Specifications of adhesive materials used in probe fabrication}
\begin{tabular}{llll}
\hline
Name & Service Temperature $K$ & Volume Resistivity ohm – cm & Thermal Conductivity W mK \\
\hline
Resbond\textsuperscript{TM}907GF & 1570 & $10^9$ & 0.86 \\
Resbond\textsuperscript{TM}919 & 1810 & $10^{11}$ & 0.58 \\
Pyro-Duct\textsuperscript{TM}597-A & 1200 & $2e^{-4}$ & 9.1 \\
\hline
\end{tabular}
\end{table}

The prongs were made from high carbon steel rods with a shank diameter of 0.46 mm. The tip of the rods were machined to a sharp point and slightly filed off to produce a small flat top surface for holding the sensor wire. They were secured by fitting into a two-bore ceramic disc, which was cut out from the two bore ceramic
tubing. The gap between the two ceramic parts was filled with high temperature electrically non-conductive cement (Resbond™ 919, see Table 3.1) to protect the connection between copper wires and the prongs as well as securing the ceramic disk in its location. The prongs were bent 90° and were extended 6.35mm away from the body. The tips of the prongs were 2.8 mm apart. High temperature electrically non-conductive epoxy was used between the two prongs to further secure them in position and damp any unwanted vibrations.

The sensor wire was Platinum 87%-Rhodium 13% wire with a diameter of 12.5 μm and tensile strength of 570 MPa, according to the manufacturer. Platinum-Rhodium wire can be used in temperatures as high as 1300 K. This alloy has a much lower tensile strength comparing to tungsten (1510 MPa). Therefore, the wire diameter was chosen to be larger than the more standard wire diameter of 5 μm. Length of the sensor wire was chosen to be in the range of 2.8 m to 3 mm in order to keep 200 < \( L_w/d < 400 \) and neglecting the end-conduction effects.

Platinum-Rhodium wire was welded to the prongs as shown in Figure 3.2a. Sensor wire was held in an apparatus attached to a micro-positioner. The wire was carefully placed on top of the prong and then sandwiched between a welding needle and the top surface of the prong. Electrical current was passed through the welding needle and into the prong which melted the sensor wire locally thus, forming a spot weld. However, the weld could potentially act as a stress concentration point and could not last in a high temperature and high Mach number flow. Hence, a thin layer of Pyro-Duct™ 597-A silver based paste, was applied on top of the spot weld to increase the mechanical strength of the joint as well as providing a better electrical and thermal connection. Hotwire probes prepared with this method survived in Mach 0.9 flow with total temperature of 644 K for more than 80 hours of testing over the period of several days.
Figure 3.2. Images of a Single Hotwire Prong Illustrating the Two-Step process for Attaching the sensor wire to the Prong, (a) Sensor Wire is Welded on the top surface of the Prong, (b) Spot Weld is Covered with a Thin Layer of Silver Paste

3.2 Calibration Method

The correction methods introduced in the previous chapter were insufficient for the intended application in this work. Thus, a new calibration equation for constant temperature mode of operation was proposed as

\[
E_c^2 \equiv E_m^2 \left( \frac{T_w - T_c}{T_w - T_x} \right)^r \left( \frac{T_f}{T_c} \right)^m = A + B \left( \frac{\rho U}{\mu} \right)^n.
\] (3.10)

where \( E_c \) is the corrected anemometer voltage output, \( U \) is the flow velocity, \( m \) and \( r \), are the calibration constants. Here \( \mu \) is the dynamic viscosity of fluid and \( \rho \) is the density of fluid. Both \( \mu \) and \( \rho \) were evaluated at the film temperature, \( T_f = \frac{1}{2} (T_w + T_x) \). The wire temperature can be found assuming a linear relationship between wire resistance and temperature, so that

\[
T_w = T_o + \frac{R - R_o}{\alpha R_o}.
\] (3.11)
where \( R_o \) is the reference wire resistance, \( T_o \) is the reference temperature at which the reference resistance was measured and \( \alpha \) is the temperature coefficient of resistance for the wire.

Total temperature should be used in evaluating the correction factors of equation (3.10) as well as the film temperature to reduce the uncertainties due the Mach number variation. Therefore, all the fluid properties are evaluated at the film temperature using the total temperature as well. In the usual wire calibration, \( \rho u \) is found as a function of \( E_m \). However, if the calibration is obtained so that \( \rho u/\mu \) is calibrated against \( E_c \), then it is possible to find \( u \), the true velocity over the wire if \( T_x \) is known. \( T_x \) can be found via an independent total temperature measurement.

Dynamic viscosity can be found using the correlation given by Sutherland [67] as,

\[
\mu = \frac{C_1 T_x^{1.5}}{T_x + S}
\]

(3.12)

where \( C_1 = 1.45810^{-6} \frac{\text{Kg}}{\text{ms}\sqrt{\text{K}}} \) and \( S = 110.4 \text{ K} \). Replacing kinematic viscosity which is not a strong function of pressure, by density and dynamic viscosity which are fluid properties, kept both temperature and pressure sensitivity in the calibration equation.

Hotwire calibrations at three different temperatures were obtained to evaluate the proposed method. Figure 3.3a shows hotwire calibrations at 644 K, 616 K and 583 K. Here, the measured voltage output squared is plotted against the Reynolds number with calibration exponent. Equation (3.10) was used in obtaining the calibration lines, having \( T_c \) equal to the nominal calibration temperature. Nominal calibration temperature was initially chosen as the average total temperature of the flow, over the course of calibration process. Calibration constants including \( n \), were calculated to minimize the root mean square error between the measured velocity values and the fitted ones. The value of \( m \) was found to be close to \(-0.17\) suggested by Collis and Williams [17] for most of the cases. The value of \( r \) was found to be close to 1
as suggested by Bearman [6]. In some occasions \( r \) was found to be close to 1.1 as proposed by Benjamin [7]. However, it did not have a noticeable effect on the measurement uncertainties. It can be argued that for simplicity, the values of \( m = -0.17 \) and \( r = 1 \) can be accepted for simple application of this method. Although, it is suggested to allow the calibration constants to float, in order to achieve the best fit. The experimental measurements are scattered above and below the fitted curve without any systematic bias error with Reynolds number. This scatter was likely caused by the high turbulence levels of the flow during in-situ calibration as well as the overall uncertainty in total temperature and total pressure measurements used in the calibration process.

The calibration temperature \( T_c \), can be chosen to be different from the average total temperature during the calibration process. Thus, a calibration acquired at a certain temperature is valid at any given flow temperature by choosing the \( T_c \) to be equal to that temperature. For example, calibrations obtained at 616 K and 583 K can be applied to 644 K by setting \( T_c = 644 \) K in the respective calibration equations.

Figure 3.3b shows the comparison between calibration line acquired at 644 K to the calibrations obtained at 616 K and 583 K corrected for 644 K. The voltage output squared is increased linearly as \( Re^{0.479} \) is increased. The data show a very good collapse for the lowest speeds. At the highest speeds, for a fixed voltage some spread in the Reynolds number was observed. This spread can be attributed to the effects of compressibility as the highest Mach number is slightly higher than 0.5. Mach number has not been independently accounted for in this calibration, which caused a deviation at higher speeds. This is supported by Spangenberg’s [66] findings which suggested that \( E_{m}^{2} \) as a measure of the Nusselt number depends on the Mach number. Note again that total temperature was used in all of the above calculations as the use of free-stream static temperature resulted in a very strong Mach number sensitivity that has to be accounted for separately. Overall, Figure 3.3 demonstrates that by
using the proposed method, a hot-wire calibration obtained at a single temperature can be applied accurately to a wide range of temperatures.

![Graph showing hot-wire calibrations at different temperatures and corrected plots for a single temperature.](image)

Figure 3.3. Hotwire Calibrations, (a) Hotwire Calibrations at Different Temperatures. (b) Corrected Hotwire Calibration Plots for a single temperature.

The overall uncertainty was calculated to be less than ±1.3 m/s for the velocity measurements in this study. The uncertainty in the independent total temperature measurement was limited to no more than ±1 K random error, which introduced an uncertainty of less than 0.05 m/s in the velocity measurement. The uncertainty in the total and static pressure measurements were the last sources of the errors for this method. Experimental measurements obtained for this work had pressure uncertainty of ±34 Pa which introduced very insignificant uncertainty in the velocity measurement comparing to the uncertainty from the data scatter and can be neglected. Overall, it can be concluded with confidence that the uncertainty in this calibration method is directly dependent on the quality of velocity and temperature measurements during the calibration process, same as any other calibration method in the literature.
3.3 Validation

The mean velocity from hot-wire measurements, calculated using the suggested calibration method, is compared to the mean velocity derived from total pressure measurements at the same locations in Figure 3.4. This figure shows a radial traverse in a constant circumferential position for the non-uniform inlet total temperature distribution. The velocity is plotted against $\eta$. Figure 3.4a shows this comparison for the low Mach number flow regime at the inlet of the test section. Figure 3.4b shows the comparison between the two measurements at the exit measurement plane of the test section where the Mach numbers were significantly higher. In both cases, the hotwire measurements are well in the uncertainty bounds of the velocities calculated from the total pressure measurements which further ensures the validity of the proposed calibration method.

![Figure 3.4. Comparison between velocity from hotwire measurement and total pressure measurement, (a) Low Mach number comparison (b) High Mach number comparison.](image-url)
CHAPTER 4

EXPERIMENTAL RESULTS

This chapter will present experimental data obtained upstream and downstream of the annular nozzle described in chapter 2. Table 4.1 shows an overview of these measurements. Inlet 1 and inlet 2 refer to the inlet measurements planes 1 and 2 respectively. Two inlet conditions were examined. The first set had a nominally uniform total temperature distribution denoted with “U”. The non-Uniform total temperature distribution case “N-U”, had a span-wise variation of total temperature with strong span-wise gradients of total temperature at both inner and outer span. Both sets had nominally the same inlet total pressure and Mach number profiles.

TABLE 4.1

MEASURED EXPERIMENTAL FLOW PROPERTIES

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Temperature Profile</th>
<th>Planes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total Pressure</td>
<td>U,N-U</td>
<td>Inlet 2, Exit</td>
</tr>
<tr>
<td>Total Temperature</td>
<td>U,N-U</td>
<td>Inlet 2, Exit</td>
</tr>
<tr>
<td>Velocity</td>
<td>U,N-U</td>
<td>Inlet 1 and 2, Exit</td>
</tr>
<tr>
<td>Turbulence Intensity</td>
<td>U,N-U</td>
<td>Inlet 1 and 2, Exit</td>
</tr>
</tbody>
</table>

The total pressure distribution in both cases were similar to each other as discussed in chapter 2. Hence, any difference in the turbulent transport between the...
two cases was a result of the inlet temperature distribution. In the non-uniform case the inlet temperature distribution caused a non-uniform density profile at the inlet which affected the turbulent transports of thermal energy and momentum. In order to study this phenomenon, total temperature, total pressure and hotwire measurements were obtained at both the inlet and exit planes of the nozzle geometry. The experimental results are presented in the following sections.

4.1 Nozzle Inlet

Contour plots of total temperature and total pressure for the uniform total temperature distribution case are presented in Figure 4.1. Measurements were obtained from 3% span to 97% span with a circumferential extent of 20°. The contours presented here are limited to 10° circumferential extent as the circumferential periodicity of the geometry. Static pressure and wall temperature were measured at both OD and ID walls which are included in the presented contour plots. The walls are represented by the dotted lines in the Figures. The total pressure and total temperature were normalized based on equations 2.1 and 2.2, respectively.

Span-wise variation of the total pressure can be observed in Figure 4.1a with the highest values of 1 found close to the walls and the lowest of 0.994 near the mid-span. A pattern for local maxima and minima can be observed in the 25% and 75% span. This patterns was caused by the circumferential position of the turbulence grid. The total temperature shown in Figure 4.1b is mostly uniform across the span. Total temperature decreased close to the outer edge of the span by 0.5%. This was due to the heat transfer from the OD wall to the ambient.
Figure 4.1. Uniform total temperature distribution case, (a) Normalized inlet total pressure, (b) Normalized inlet total temperature.

The turbulence intensity distribution in the uniform distribution case are shown in Figure 4.2. Turbulence intensity was calculated as

\[ TI = \frac{u_{rms}}{U}. \]  

(4.1)

where \( u_{rms} \) is the rms velocity. The highest values of turbulence intensity can be found in the midspan and the minimum close to both walls. The area average turbulence intensity value was 16.5%. The pattern from the grid in the turbulence intensity contour is similar to the one of total pressure. These values which were anticipated from the multi-scale turbulence grid, correspond to a high turbulence flow which was of the interest in this study.

Contour plots of total temperature and total pressure for the non-uniform total temperature distribution case are presented in Figure 4.3. Measurements were obtained from 3% span to 97% span with a circumferential extent of 20°. The contours
Figure 4.2. Inlet turbulence intensity distribution in uniform case

presented here are limited to 10° circumferential extent in the same manner as the non-uniform case. The total pressure and total temperature were normalized based on equations 2.1 and 2.2, respectively.

The total pressure distribution in the non-uniform case was similar to the distribution in the uniform case. Slight differences in the contour plots of the two cases were due to more radial and circumferential traverse points in the non-uniform case. The pattern from the turbulence grid can be seen in Figure 4.3a. The total temperature has the highest value in the mid-span which is equal to the total temperature in the uniform case. The flow was colder at both walls due to the colder air which was injected in the rig behind the turbulence grid through openings at outer and inner walls.

The turbulence intensity distribution in the non-uniform case are shown in Figure 4.4. The location of high and low values of turbulence intensity in non-uniform case was similar to Figure 4.1b. The pattern for local maxima and minima seen in the 25% and 75% span were due to the circumferential position of the turbulence grid.
The average value of turbulence intensity was 17% which was slightly higher than the uniform case. This difference can be attributed to the more dense traverse points in the non-uniform case.

4.2 Nozzle Exit

Contour plots of total temperature and total pressure for the uniform case at the exit measurement plane are presented in Figure 4.5. Measurements were obtained from 14% span to 86% span with a circumferential extent of 20°. The contours presented here are limited to 10° circumferential extent as the circumferential periodicity of the geometry. The total pressure and total temperature were normalized based on equations 2.1 and 2.2 respectively.

The total temperature profile at the exit plane had the highest values close to the inner wall and lower values close to the OD wall. This can be contributed to the heat transfer from the outer walls of the test section to the ambient. The total

Figure 4.3. Non-uniform total temperature distribution case, (a) Normalized inlet total pressure, (b) Normalized inlet total temperature.
temperature variation was about 2 $K$ throughout the span thus, a nominally flat profile. The total pressure is presented in Figure 4.5a had the highest values close to both walls with the minimum at the mid-span. Turbulent mixing through the nozzle caused a relatively flat total pressure and total temperature distributions at the exit plane.

The turbulence intensity distribution is shown in Figure 4.6. The ratio of the maximum to the minimum value of the turbulence intensity was half the ratio of the inlet. There were random spots of high turbulence intensity values mostly concentrated closer to the OD wall. This can be contributed to the higher turbulence intensity values close to OD at the inlet plane as seen in Figure 4.2. The relative flatter exit profile was expected to be a result of the turbulent mixing through the nozzle.

Contour plots of total temperature and total pressure for the non-uniform case at the exit measurement plane are presented in Figure 4.7. Measurements were obtained from 14% span to 86% span with a circumferential extent of 20°. The contours pre-
Figure 4.5. Uniform total temperature distribution case, (a) Normalized exit total pressure, (b) Normalized exit total temperature.

Figure 4.6. Exit turbulence intensity distribution in uniform case

...presented here are limited to 10° circumferential extent as the circumferential periodicity of the geometry. The total pressure and total temperature were normalized based on equations 2.1 and 2.2, respectively.

The total temperature had the lowest values at the walls and the higher values in the mid-span. The ratio of the maximum to minimum total temperature was 1.02 which was much lower than the ratio of 1.12 at the inlet thus, a flatter exit profile relative to the inlet temperature distribution. This was caused by the high turbulence intensity at the inlet plane. As the flow moved through the nozzle, due to the turbulence mixing, the colder fluid at the walls mixed with the fluid at the mid-span thus, causing the flatter total temperature profile at the exit plane.

The total pressure profile at the exit plane is shown in figure 4.7a had the highest...
values close to the outer edges of the span and the lower values in middle of the span. This distribution is similar to the one shown in Figure 4.5a.

The turbulence intensity profile in the exit plane can be observed in Figure 4.8. The average turbulence intensity value was similar to the uniform case. The turbulence intensity had the highest values at the both edges of the span and the lowest values in the mid-span. This was in contrast with the inlet distribution of the turbulence intensity shown in Figure 4.4. It can be due to the mixing through the nozzle which caused the turbulent flow to migrate closer to the walls while the flow in the midspan was much less turbulent.
CHAPTER 5

COMPUTATIONAL SETUP

Steady state RANS simulations were obtained to study the capability of existing turbulence and turbulent Prandtl number models in modeling the turbulence flow through the nozzle geometry described in chapter 2. The numerical domain was similar to the experimental geometry. The experimental measurements described in chapter 4 were used to prepare the inflow boundary conditions. The assumptions made to make the flow problem amenable to computational simulations are discussed in section 5.1. The meshes generated for this study will be discussed in this chapter along with the utilized solver. The details of the model equations solved in this analysis are discussed in the final section.

5.1 Geometry and Boundary Conditions

A schematic of the flow geometry is shown in Figure 5.1. A 10° slice of the full annulus was chosen for this study. Rotational periodic boundary condition was applied to the side boundaries of the model to account for the periodicity in the experimental measurements. Table 5.1 shows an overview of the considered turbulence models along with the necessary boundary conditions for each model.
TABLE 5.1

TURBULENCE MODELS INVESTIGATED IN THE NUMERICAL ANALYSIS

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>Inlet B.C.s</th>
<th>Exit B.C.</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k - \epsilon$</td>
<td>Total Pressure and Temperature, Turbulent kinetic energy, turbulence dissipation</td>
<td>Static Pressure</td>
</tr>
<tr>
<td>Realizable $k - \epsilon$</td>
<td>Total Pressure and Temperature, Turbulent kinetic energy, Turbulence dissipation</td>
<td>Static Pressure</td>
</tr>
<tr>
<td>Spalart-Allmaras</td>
<td>Total Pressure and Temperature, Turbulent viscosity ratio</td>
<td>Static Pressure</td>
</tr>
<tr>
<td>$k - \omega$</td>
<td>Total Pressure and Temperature, Turbulent kinetic energy, Specific turbulence dissipation</td>
<td>Static Pressure</td>
</tr>
<tr>
<td>Kato-Launder $k - \omega$</td>
<td>Total Pressure and Temperature, Turbulent kinetic energy, Specific turbulence dissipation</td>
<td>Static Pressure</td>
</tr>
<tr>
<td>SST $k - \omega$</td>
<td>Total Pressure and Temperature, Turbulent kinetic energy, Specific turbulence dissipation</td>
<td>Static Pressure</td>
</tr>
<tr>
<td>Reynolds Stress Model</td>
<td>Total Pressure and Temperature, Turbulent kinetic energy, Specific turbulence dissipation</td>
<td>Static Pressure</td>
</tr>
</tbody>
</table>
Total pressure and total temperature were measured experimentally as discussed in the previous chapter. Also, wall temperatures were measured through the contraction which were used as constant wall temperature boundary conditions in the numerical simulations. The turbulence kinetic energy was used in most of the evaluated models. It was obtained by taking the trace of the Reynolds stress tensor as

$$k = \frac{1}{2} u'_i u'_i. \quad (5.1)$$

The turbulent velocity in the axial direction, $u'$, was measured experimentally. Isotropic turbulent flow assumption was made to calculate the turbulent kinetic energy. The solver presumed an isotropic turbulent flow as well. Thus, the turbulence kinetic energy at the inlet boundary condition has been divided equally between the three main directions. The turbulent kinetic energy was calculated from the experimental data as

$$k = \frac{3}{2} (u')^2. \quad (5.2)$$

With the isotropic turbulence assumption, the dissipation can be approximated by (see \[46\])
\[ \epsilon = 15\nu \left( \frac{\partial u'}{\partial x} \right)^2. \] (5.3)

Hotwire measurements were obtained at the inlet measurement planes 1 and 2 separated by a distance of \( \Delta x_{inlet} = 23 \) mm. Turbulent dissipation \( \epsilon \) can be measured between these two planes using equation 5.3 as

\[ \epsilon \approx 15\nu \left[ \frac{\left( \frac{u'^2}{\text{plane 2}} \right) - \left( \frac{u'^2}{\text{plane 1}} \right)}{\Delta x_{inlet}} \right]. \] (5.4)

Using the stream-wise Taylor microscale \( \lambda_u \), a theoretical estimate of the dissipation can be calculated based on the equation 5.4

\[ \epsilon = 30\nu \frac{(u')^2}{\lambda_u^2}. \] (5.5)

Turbulence dissipation was calculated based on the equations 5.4 and 5.5. The experimental results were suspected to be inadequate as the measurement planes were positioned too far from each other. However, the dissipation calculated from the equation 5.4 were close to the theoretical calculations. Hence, the turbulent dissipation for the computational boundary condition were calculated using equation 5.5.

Specific turbulence dissipation \( \omega \), was needed to setup the boundary conditions for the \( k - \omega \) turbulence model and its variations which was calculated as

\[ \omega = \frac{\epsilon}{C_{\mu}k}. \] (5.6)

where \( C_{\mu} \) is a turbulence model constant mostly equal to 0.09.

The experimental data showed a slight aperiodicity. It is important for the numerical solution to start with a truly periodic boundary condition. In order to satisfy this condition, the data were processed as follows. A Fourier transform in circumfer-
ential direction was obtained. The first six modes were retained to reconstruct the flow field. The rest of the modes were not considered to remove the high wavenumber discontinuous information. Figure 5.2 shows the inlet boundary conditions for the non-uniform inlet total temperature distribution.

![Figure 5.2](image)

Figure 5.2. Computational inlet boundary conditions of non-uniform cases, (a) Total pressure. (b) Total temperature. (c) Turbulence kinetic energy (d) Turbulent dissipation (e) Specific turbulent dissipation

5.2 Solver

The numerical results were obtained computationally using ANSYS FLUENT® 17. Details of the solver algorithm and structure is provided by ANSYS® [2]. The solver was configured to solve a three-dimensional, compressible turbulent flow problem using the pressure based solver along with the coupled energy equation. The
pressure-velocity coupling was treated with a coupled scheme. The gradient spatial
discretization was set to least squares cell based method while other spatial discretiza-
tion options were set to the second order upwind method. The flow courant number
was chosen to be 200. The under-relaxation factors were set to the default values.
The conservation of mass flow rate and total enthalpy was monitored along with the
heat transfer from the system through each computational run.

5.3 Mesh

The computational grid was generated using Pointwise® 17.3R5. The mesh topol-
yogy for the nozzle is shown in Figure 5.3. Details of the meshing software is provided
by Pointwise [49]. Structured mesh was chosen for the entire domain space of this
simulation. The nozzle domain consisted of approximately 8.2 million cells. Grid
points adjacent to the wall boundaries in the inlet were spaced such that $y^+ \approx 1$.
Inlet Reynolds number was measured to be $1.023 \times 10^5$ based on the height of the
inlet which was slightly higher than the measured Reynolds number $9.7 \times 10^5$ at the
exit. Thus, the grid points adjacent to the exit wall boundaries were positioned closer
than the distance of $y^+ \approx 1$.

![Figure 5.3. Grids generated for the computational study, (a) Nozzle grid. (b) Nozzle wall grid.](image)

Four additional grids were generated to study the mesh dependency of computa-
tional results. Structured mesh was chosen for all the generated grids with the cell spacing of \( y^+ \sim 1 \) adjacent to the wall boundaries. Table 5.2 shows the number of cells in each grid. Mesh dependency study was performed for each turbulence model and inlet temperature distribution case separately.

### Table 5.2

**NUMBER OF CELLS IN THE COMPUTATIONAL GRIDS.**

<table>
<thead>
<tr>
<th>Grid</th>
<th>Number of Cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>1.5 million</td>
</tr>
<tr>
<td>II</td>
<td>3.5 million</td>
</tr>
<tr>
<td>III</td>
<td>8.2 million</td>
</tr>
<tr>
<td>IV</td>
<td>14.5 million</td>
</tr>
<tr>
<td>V</td>
<td>26.9 million</td>
</tr>
</tbody>
</table>

Circumferential averaged total pressure and temperature, mean velocity and turbulence intensity of the grids were compared at the exit plane to evaluate the grid convergence. Figure 5.4 shows an example of such comparisons made for the non-uniform case with SST \( k-\omega \) turbulence model and constant turbulent Prandtl number. Total temperature and the axial velocity were identical across all the grids. Grids III, IV and V had identical total pressure distributions while the meshes with the lower cell count had slightly higher values of total pressure. There was less than 0.01% difference in the turbulence intensity values between all five grids. Based on the presented results grid III was chosen for this specific study. Moreover, these results were mostly consistent across the different turbulence models for both inlet total temperature distribution cases. Hence, the grid III was chosen for all the
computational analysis in this study.

Figure 5.4. Comparison between the computational results across the different grids, (a) Total pressure. (b) Total temperature. (c) Mean axial velocity. (d) Turbulence intensity.

5.4 Turbulence Modeling

Turbulence models are of sets of partial differential equations, in some cases supplemented with appropriate algebraic equations that predict the effects of turbulence. For a given flow, with defining the appropriate initial boundary conditions, these equations can be solved numerically. Various turbulence models listed in table 5.1
were studied in this work to assess their performance in the complex environment of the problem in hand as well as quantifying the turbulent Prandtl number. The models introduced in this section were studied in more detail as they showed more prominent results in predicting the experimental measurements.

5.4.1 SST $k - \omega$

The two equations SST $k - \omega$ turbulence model developed by Menter effectively blends the robust and accurate formulation of the $k - \omega$ model in the near-wall region with the free-stream independence of the $k - \epsilon$ model away from the walls. This model does not involve damping functions unlike any other two-equation model which allows simple Dirichlet boundary conditions to be simplified. $k - \omega$ model is superior to other models because of its simplicity and numerical stability. It is the model of choice on the sublayer of the boundary layer as well is in the logarithmic part of the boundary layer. However, in the wake region of the boundary layer, the $k - \omega$ model has a very strong sensitivity to the free stream values of specific dissipation rate ($\omega$) thus, it should be abandoned in favor of the $k - \epsilon$ model.

To achieve the desired features in each of the regions discussed above, the standard high Reynolds number $k - \epsilon$ formulation is transformed to a $k - \omega$ model. It is then multiplied by a blending function $(1 - F_1)$ and added to the original $k - \omega$ model to obtain the $k - \omega$ SST formulation. $F_1$ is designed to be 1 in the sublayer and logarithmic region of the boundary layer and to gradually switch to zero in the wake region.

Two equation turbulence models have two extra transport equations to represent the properties of the flow. This allows for the model to account for history effects like convection and diffusion of turbulent energy. In this case the two transported variables are turbulent kinetic energy, $k$ (see equation 5.7) and turbulent specific dissipation rate $\omega$ (see equation 5.8). $k$ determines the energy of
turbulent flow while $\epsilon$ determines the scale of turbulence. The transport equations for the SST $k\omega$ model were solved using FLUENT®.

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k, \quad (5.7)$$

and,

$$\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_j} (\rho \omega u_j) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega. \quad (5.8)$$

In these equations, $G_k$ is generation of turbulent kinetic energy due to mean velocity gradients, $G_\omega$ is the generation of turbulent specific dissipation rate due to mean velocity gradients, $\Gamma_k$ is the effective diffusivity of turbulent kinetic energy, $\Gamma_\omega$ is the effective diffusivity of specific dissipation rate, $Y_k$ is the dissipation of turbulent kinetic energy, $Y_\omega$ is dissipation of specific dissipation rate, $S_\omega$ and, $S_\omega$ are the user defined terms. Details of each term can be found in the FLUENT® theory guide [21].

The energy equation which was solved along with the aforementioned transport equations is

$$\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\bar{\nu} (\rho E + p)) = \nabla \cdot \left( k_{\text{eff}} \nabla T - \sum_j h_j \overline{J}_j + (\tau_{\text{eff}} \cdot \bar{\nu}) \right) + S_h, \quad (5.9)$$

where $k_{\text{eff}} \nabla T$ is the energy transfer due conduction, $\sum_j h_j \overline{J}_j$ is the energy transfer due species diffusion, $\tau_{\text{eff}} \cdot \bar{\nu}$ is the energy transfer due viscous dissipation and, $S_h$ is the volumetric heat source. The term of interest in the energy equation is the effective conductivity $k_{\text{eff}}$, as underlined in the equation [5.9]. This term is defined as,

$$k_{\text{eff}} = k + k_t. \quad (5.10)$$

Turbulent conductivity $k_t$ is defined by
This study investigates the effect of various turbulent Prandtl number models on the effectiveness of the computational analysis in predicting the experimental measurements. $Pr_t$ can be defined as a constant value in FLUENT® or as a function by utilizing user defined functions (UDF).

5.4.2 Reynolds Stress Model

In the Reynolds Stress Model (RSM), the differential equations governing the transport of the individual kinematic Reynolds stresses $\overline{u_i u_j}$ are solved along for the dissipation rate $\epsilon$ or specific dissipation rate $\omega$. The latter would provide a length or time scale for the turbulence. Thus, the turbulent-viscosity assumption that $-\overline{u_i u_j}$ is locally determined by $\frac{\partial \overline{u_i}}{\partial x_j}$ is unnecessary. In the Reynolds stress model, the mean convection of the Reynolds stresses is balanced by four processes-production, dissipation, redistribution, and turbulent transport. These equations \[\text{33}\] can be written as

$$k_t = \frac{c_p \mu_t}{Pr_t}$$  \hspace{1cm} (5.11)
\[
\frac{\partial}{\partial t} \left( \rho u'_i u'_j \right) + \frac{\partial}{\partial x_k} \left( \rho u_k u'_i u'_j \right) = - \frac{\partial}{\partial x_k} \left[ \rho u'_i u'_j u'_k + p' \left( \delta_{kj} u'_i + \delta_{ik} u'_j \right) \right] \\
+ \frac{\partial}{\partial x_k} \left[ \mu \frac{\partial}{\partial x_k} (u'_i u'_j) \right] - \rho \left( u'_i u'_k \frac{\partial u_j}{\partial x_k} + u'_j u'_k \frac{\partial u_i}{\partial x_k} \right) \\
- \rho \beta \left( g_i u'_j \theta + g_j u'_i \theta \right) + p' \left( \frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right) - 2\mu \frac{\partial u'_i}{\partial x_j} \frac{\partial u'_j}{\partial x_i} \\
- 2\rho \Omega_k \left( u'_k u'_m \epsilon_{ikm} + u'_i u'_m \epsilon_{jkm} \right) + S_u \\
\]

(5.12)

In the equation above, \( C_{ij} \) is convection, \( D_{L,ij} \) is molecular diffusion, \( P_{ij} \) is stress production, \( F_{ij} \) is production by system rotation, \( D_{T,ij} \) is turbulent diffusion. \( G_{ij} \) is buoyancy production, \( P_{ij} \) is pressure strain and, \( \epsilon_{ij} \) is the dissipation term [50]. Among these terms \( C_{ij}, \) \( D_{L,ij}, \) \( P_{ij} \) and, \( F_{ij} \) do not require any modeling. However, the rest of the terms should be modeled to close the exact equations. The details of such modelings can be found in the FLUENT® theory guide [21].

In comparison with the two equation models, Reynolds stress models have been demonstrated to be superior to two equation models. However, they are somewhat more difficult and costly to implement because there are seven turbulence equation to be solved (for \( u_i u_j \) and \( \epsilon \), or \( \omega \).
Detailed comparisons between numerical solutions and experimental measurements are presented in this chapter. The process used is outlined as follows. In the first section, various turbulence models with a fixed turbulent Prandtl number were used to obtain a numerical solution. The efficacy of the turbulence models were evaluated by comparing the circumferentially averaged total pressure, total temperature and turbulence intensity values from the numerical simulation and the experimental measurements at the exit measurement plane. These comparisons included the magnitude as well as the shape of the profiles. The SST $k-\omega$ model was found to be superior to the other models in predicting the experimental measurements.

SST $k-\omega$ turbulence model was used with variety of turbulent Prandtl numbers and turbulent Prandtl number models. The results were compared with experimental measurements in the same manner as the previous step in order to identify the effective best turbulent Prandtl number for the case of study. The detailed comparisons are presented in the second section. The turbulent Prandtl number was found to be approximately 0.5. None of the turbulent Prandtl number models predicted the correct value.

Total pressure and total temperature profiles were chosen as critical flow properties to evaluate turbulence and turbulent Prandtl number models in the process of comparing numerical results with experimental measurements. Both the magnitude and the shape of the total pressure profiles from the CFD and experiment were compared to each other. Near wall comparison of these properties were neglected due
to the lack of experimental measurements near the wall to set the appropriate inlet boundary conditions.

The experimental total temperature measurements at the exit plane had an uncertainty of $\pm 1 \, K$. For purpose of the comparison, it was important to recognize that there was an uncertainty in the overall enthalpy flux that affects the comparison of the exit total temperature magnitudes between CFD and experiment. Both CFD and experiment were adiabatic and while the inlet boundary condition was set as close to the experimental boundary condition as possible, the uncertainty in the inlet temperature measurements and the limited traverse points caused an uncertainty in the overall enthalpy flux through the numerical domain. The enthalpy flux from the CFD was found to be within 1% of the experimental enthalpy flux in all computational cases. However, a new variable $\theta_t$ was introduced to eliminate the net enthalpy difference between the numerical simulations and experimental measurements as

$$
\theta_t = \frac{T_t(\eta) - \langle T_t(\eta) \rangle}{T_{t_{max}}},
$$

(6.1)

where $T_t(\eta)$ is the circumferentially averaged total temperature. Therefore, by using this variable, the comparison was only made by the shape of the total temperature profiles rather than the magnitudes in order to investigate the turbulent mixing of thermal energy.

The turbulence intensity comparison had the lowest priority as the goal of the analysis was not to necessarily predict the exact turbulence characteristics of the nozzle exit plane. The isotropic turbulence assumption made in the computational analysis despite the anisotropy in the experiments, cause an uncertainty in the turbulence properties predicted by the CFD. The CFD was expected to over-predict the turbulent kinetic energy at the exit plane due to this assumption. The axial velocity profile from the numerical solution was compared to the experimental measurements as well. However these comparisons are not included here. It was shown in the ex-
perimental measurements that the mean values of velocity measured by hotwire were 
similar to the mean velocity distribution calculated from total pressure and static 
pressure at the exit plane. Thus, the comparison between total pressure profiles from 
the numerical solution and the experimental measurements provided a comparison 
between the mean axial velocities as well.

6.1 Turbulence Models

The turbulence models under consideration in this study are shown in table 6.1. 
These models can be divided in 4 groups, $k - \epsilon$ family, Spalart-Allamaras (SA), 
Reynolds stress model and, $k - \omega$ family. The results from the first three groups are 
reviewed briefly while the SST $k - \omega$ is presented with more details as it performed 
well in predicting the experimental measurements.

The realizable $k - \epsilon$ turbulence model was the first model used in this study. 
The term “realizable” means that the model satisfies certain mathematical constraints 
on the Reynolds stresses. First, if $k \frac{\partial U}{\partial x} > \frac{1}{3C_{\mu}}$, then the term $\overline{u^2}$ becomes negative 
which is unrealistic. Realizable $k - \epsilon$ model will satisfy the Schwartz inequality, 
$(\overline{u_\alpha u_\beta})^2 \leq \overline{u_\alpha^2 u_\beta^2}$ as well.

Figure 6.1 shows a comparison between the circumferentially averaged total pres-
sure and total temperature from experimental measurements and the computational 
model based on realizable $k - \epsilon$ turbulence model in the non-uniform inlet temperature 
distribution case. The experimental normalized total pressure had a local minimum 
value of approximately 0.9986 near the center of span. The magnitude of total pres-
sure increased consistently moving away from the midspan and shows a maximum 
value close to unity at the outer edges of the span. The computational total pressure 
had a local maximum value of 0.999 at $\eta = 0.35$, with the magnitude of total pressure 
decreasing rapidly close to the outer and inner walls. The magnitude of the computa-
tional total pressure was within the uncertainty bars of the experimental data at the
mid-span with a different trend in the slope of the profile. The CFD under-predicts the total pressure magnitude by 0.4% near the walls at $\eta = 0.1$ and 0.9. The $k - \epsilon$ turbulence model does not predict the magnitude or the shape of the total pressure profile correctly. This can be attributed to a known shortcoming of this model in the flows with large pressure gradients[76].

The experimental normalized total temperature in Figure 6.1b had a local maximum value of 0.008 near the center of the span. The magnitude of total temperature decreased more rapidly moving away from the mid-span toward the outer wall than toward the inner wall. Normalized total temperature $\theta_t$, had a value of $-0.003$ at $\eta = 0.14$ and at $\eta = 0.84$ had the minimum value of $-0.013$. The lower $\theta_t$ values at the OD wall were due to the lower total temperatures close to the OD in the inlet of the nozzle. The computational normalized total temperature had a local maximum of approximately 0.004 close to the center of the span. The magnitude of total temperature decreased moving away from the mid-span toward the outer edges of the span. CFD under-predicted the slope of the temperature profile at the mid-span.
and over-predicted it closer to the walls. Overall, the computational profile of $\theta_i$ was flatter compared to the experimental profile due to the over-prediction of thermal mixing by $k-\epsilon$ turbulence model.

In the standard $k-\epsilon$ turbulence model, the turbulence viscosity is calculated as

$$\nu_t = C_\mu \frac{k^2}{\epsilon}.$$  \hspace{1cm} (6.2)

This formulation\cite{16} does not provide accurate estimations for the eddy viscosity where the Reynolds number is low for example, close to the solid walls. Thus the RNG variation of $k-\epsilon$ model was evaluated as well, which incorporates a differential equation for calculating effective viscosity accounting for low-Reynolds number effects\cite{77}. However, this formulation did not improve the computational predictions and was not adequate to account for the aforementioned shortcoming in the $k-\epsilon$ turbulence model either.

![Figure 6.2](image)

Figure 6.2. Comparison between experimental measurements and computational analysis based on SA model, (a) Circumferentially averaged total pressure. (b) circumferentially averaged total temperature.

The SA model\cite{65} is a one equation model that solves a modeled transport equation for the kinematic eddy viscosity, $\nu$. It was designed specifically for aerospace
applications involving wall-bounded flows [21]. Figure 6.2a shows a comparison between the circumferentially averaged normalized total pressure and total temperature from experimental measurements and the computational model based on realizable SA turbulence model in the non-uniform inlet temperature distribution case.

Computational normalized total pressure had a local maximum of 0.997 at approximately $\eta = 0.4$. The magnitude of the pressure decreased as moving away from the center of the span. Computational total pressure decreased more rapidly closer to the outer edges of the span. Total pressure was under-predicted by 0.7% at $\eta = 0.14$ and, $\eta = 0.86$, by approximately 0.15% at the mid-span. Overall the SA model under-predicted the magnitude of total pressure at the exit measurement plane significantly.

The computational normalized total temperature had an approximate constant value of $\theta_t = 0.001$ from $\eta = 0.2$ to $\eta = 0.7$ and it decreased rapidly outside this range. The shape of the total temperature profile predicted by the SA turbulence model was considerably flatter comparing to the experimental profile due to the over-prediction of thermal mixing by this turbulence model.

A disadvantage of standard two-equation turbulence models is the excessive generation of turbulence energy, $G_k$, in the vicinity of stagnation points and walls [30]. Kato proposed a formulation for $G_k$ to limit the excessive production of the turbulence energy caused by high level of shear strain rate $S$. Figure 6.3 shows a comparison between the circumferentially averaged normalized total pressure and total temperature from experimental measurements and the computational model based on Kato-Launder $k – \omega$ turbulence model in the non-uniform inlet temperature distribution case.

The computational total pressure had a maximum value of 0.9993 at $\eta = 0.14$ which was 0.05% lower than the experimental total pressure. The magnitude of the total pressure decreased moving away from the inner wall and had a local minimum
of 0.9989 at \( \eta = 0.72 \). The computational total pressure increased slightly moving toward the OD wall away from the location of the local minimum. Although the computational magnitudes of total pressure were within the uncertainty bounds of the experimental measurements, the shape of the profile differed from the experimental slope significantly. The spread of total pressure in the experiment was 0.0012 compared to the computational spread of 0.0004. The location of the computational local minimum was skewed considerably toward the outer wall as well. It was also noticeable that the computational total temperature distribution was similar to the distribution predicted by the realizable \( k - \epsilon \) turbulence model which was discussed earlier in this section.

Figure 6.4 shows a comparison between the circumferentially averaged normalized total pressure and total temperature from experimental measurements and the computational model based on Reynolds Stress Model (RSM) turbulence model in the non-uniform inlet temperature distribution case.

The computational total pressure had a local minimum of 0.9975 at the center
of the span. The magnitude of total pressure increased as moving away from the mid-span toward the edges of the outer span. The shape of the profile was similar to the circumferential averaged total pressure at the inlet as shown in figure 2.3b. The magnitudes predicted by Reynolds stress model were lower than the experimental measurements throughout the span of the exit measurement plane.

Computational normalized total temperature profile had a constant value of $\theta_t = 0$ over the span of the exit measurement plane. The profile was completely flat and does not match the shape of the experimental measurements. RSM over-predicted the thermal mixing through the nozzle significantly. In the present case, the Reynolds stress components except $u^2$ were not measured experimentally. Instead, turbulent kinetic energy $k$ and specific dissipation rate $\omega$, were used to define the inlet boundary conditions along with estimates for the remaining components of the Reynolds stress tensor. The estimate was obtained using the isotropy indicator measurements performed by Ranade [52] as explained in chapter 5. RSM solves the complete Reynolds stress tensor hence, it should provide more accurate numerical solution than two
equation turbulence models. However, the simulation performed using RSM and
the aforementioned set of boundary conditions were inadequate for predicting the
turbulent transport of energy for the case of interest.

Figure 6.5 shows a comparison between the circumferentially averaged normal-
ized total pressure and total temperature from experimental measurements and the
computational model based on SST $k - \omega$ turbulence model in the non-uniform inlet
temperature distribution case.

Figure 6.5. Comparison between experimental measurements and
computational analysis based on SST $k - \omega$ model, (a) Circumferentially
averaged total pressure. (b) circumferentially averaged total temperature.

The computational normalized total pressure had a local minimum of 0.9984 at the
mid-span. The magnitude of total pressure increased moving away from the mid-span
with a local maximum of 0.9995 at $\eta = 0.2$ and a local maximum of approximately
0.9984 at $\eta = 0.8$. Total pressure decreased after the local maxima toward the walls
on both sides of the span. The predicted magnitude of total pressure was within
the uncertainty bounds of the experimental measurements throughout the range of
comparison except closer to the OD which was 0.08% lower. The difference between
the predicted total pressure and the measured one can be due to the inlet boundary
conditions. The experimental measurements at the inlet were obtained up to $\eta = 0.97$ and still showed an increase in the total pressure value. To interpolate between the measured value and the wall static pressure, a linear change of the total pressure was assumed. If the real experimental total pressure still increased passed $\eta = 0.97$ before dropping down to the static pressure at the wall, it can explain the slight difference between the CFD and the experimental measurements at the exit plane.

The computational normalized total temperature had a local maximum of 0.08 at the center of the span. The magnitude of total pressure decreased moving away from the center of the span toward the walls. The predicted magnitudes by the turbulence model were within the bounds experimental measurements throughout the range of comparison except close to the walls.

It is useful to define a quantitative metric to compare the shape of the circumferentially-averaged profiles of total pressure and total temperature between the experimental measurements and computational solutions. A $L^2$ type norm can be used to quantify the effectiveness of turbulence models in predicting the experimental total temperature and total pressure measurements. It can be defined for the normalized total pressure as

$$S_{\Pi_t} = \left[ \sum (\Pi_{t,exp} - \Pi_{t,cfd})^2 \right]^{\frac{1}{2}}. \quad (6.3)$$

The L2-norm for the normalized total temperature is

$$S_{\Theta_t} = \left[ \sum (\Theta_{t,exp} - \Theta_{t,cfd})^2 \right]^{\frac{1}{2}}. \quad (6.4)$$

Figure 6.6 shows the $S_{\Pi_t}$ and $S_{\Theta_t}$ for the circumferentially averaged normalized total pressure and total temperature. The SST $k-\omega$ and Kato-Lauder $k-\omega$ models had the lowest value of $S_{\Pi_t}$. The circumferentially averaged normalized total pressure for Kato-Lauder $k-\omega$ model shown in figure 6.3 had a relatively flatter shape than
the circumferentially averaged experimental total pressure profile. Thus, it can be inferred from figure 6.6a that the shape and absolute values of the numerical total pressure based on the SST $k - \omega$ model, matched the experimental measurements better compared to the other turbulence models considered in this study. Based on $S_{\Theta_t}$ shown in figure 6.6b, Kato-Launder $k - \omega$, realizable $k - \epsilon$ and, SST $k - \omega$ models had similar performance. The realizable $k - \epsilon$ model resulted in a higher value of $S_{\Pi_t}$. It is important to note that, the effectiveness of the turbulence models cannot be evaluated solely based on the value of $S_{\Theta_t}$. Considering the shape of the normalized total temperature profiles between the numerical simulations and the experimental measurements (see figures 6.3 and, 6.5), it can be concluded that the SST $k - \omega$ had the best overall performance among the turbulence models.

![Figure 6.6. L2-norm between the experimental measurements and numerical solutions for various turbulence models, (a) $S_{\Pi_t}$ for circumferentially averaged total pressure. (b) $S_{\Theta_t}$ for circumferentially averaged total temperature.](image)

The computational normalized total temperature based on the SST $k - \omega$ model had a higher gradient than the experimental measurements. This was due to the under-predicting of thermal mixing by the turbulence model. This rate could be changed by changing the turbulent Prandtl number. Decreasing $Pr_t$, should increase
the thermal mixing in the turbulence model thus, decreasing the gradient of total
temperature in the exit measurement plane. This will be investigated in the next
section.

6.2 Effect of Turbulent Prandtl Number

Turbulent Prandtl number relates the eddy viscosity defined in the SST $k - \omega$
model to the eddy diffusivity of heat in the energy equation as shown by equation
5.11. The eddy viscosity is modeled as

$$\nu_t = \frac{k}{\omega} \times \frac{1}{\max\left[\frac{1}{\alpha^*}, \frac{SF_2}{\alpha_1, \omega}\right]},$$  \hspace{1cm} (6.5)

where $S$ is the strain rate tensor, $F_2$ is a blending function, $\alpha_1$ is a model constant and,
$\alpha^*$ is a function of turbulent Reynolds number. Varying turbulent Prandtl number
while directly affects the energy equation and turbulent heat transport, indirectly
changes the momentum transport in the model by changing the flow distribution as
the energy equation and the model transport equations are coupled.

![Figure 6.7](image_url)

Figure 6.7. Comparison between experimental measurements and
computational analysis with varying constant turbulent Prandtl number
based on SST $k - \omega$ model, (a) Circumferentially averaged total pressure.
(b) circumferentially averaged total temperature.
The turbulent Prandtl number is defined as a constant in the turbulence model. By varying this constant value and comparing the computational results with the experimental measurements, some physical insights can be gained on how the momentum and thermal energy are transported by turbulent fluctuations in this flow. Figure 6.7 shows the effect of turbulent Prandtl number on the total pressure and total temperature distribution at the exit plane of the nozzle.

Figure 6.7a shows a comparison between the normalized experimental total pressure measurements and the computational total pressure calculated based on the SST $k - \omega$ turbulence model with varying constant turbulent Prandtl numbers. The computational total pressure profiles had local minimum values at the midspan. The magnitude of total pressure increased as moving away from the mid-span toward the outer edges of the span with local maximums close to $\eta = 0.2$ and $\eta = 0.8$. The rate of change of the total pressure decreased by decreasing the turbulent Prandtl number. The magnitudes of predicted total pressure values were within the uncertainty bounds of the experimental measurements for all the values of turbulent Prandtl number except $Pr_t = 0.3$ at the midspan.

Figure 6.7b shows a comparison between the normalized experimental total temperature measurements and the computational total temperature calculated based on the SST $k - \omega$ turbulence model with varying constant turbulent Prandtl numbers. The computational total temperature profiles had local maximum values at the midspan. The magnitude of total temperature decreased as moving away from the mid-span toward the outer edges of the span. The rate of change of the total temperature decreased by decreasing the turbulent Prandtl number. The magnitude of the predicted total temperature profiles were within the uncertainty bounds of the experimental measurements throughout the span for turbulent Prandtl number of 0.5.

The turbulent Prandtl number value of 0.5 can be also confirmed by comparing the
$L^2$ norms for normalized total pressure and total temperature, defined by equations 6.3 and 6.4 respectively. Figure 6.8 shows the $S_{\Pi_t}$ and $S_{\Theta_t}$ calculated between the experimental measurements and numerical solutions based on the SST $k - \omega$ turbulence model with varying constant turbulent Prandtl number.

![Graphs showing $S_{\Pi_t}$ and $S_{\Theta_t}$](image)

Figure 6.8. L2-norm between the experimental measurements and numerical solutions based on SST $k - \omega$ turbulence model, (a) $S_{\Pi_t}$ for circumferentially averaged total pressure. (b) $S_{\Theta_t}$ for circumferentially averaged total temperature.

$S_{\Pi_t}$ had an absolute minimum value for the turbulent Prandtl number of 1 as shown in figure 6.8a. Its value increased more rapidly by decreasing the turbulent Prandtl number. This local minimum can be attributed to the relatively significant deviation of the computational results from the experimental measurements close to the walls as shown in figure 6.7a. The lack of inlet experimental measurements close to the annulus walls could affect the computational inlet boundary conditions as explained before. Thus, it was more critical to consider the $S_{\Theta_t}$ profile shown in figure 6.8b to quantify the turbulent Prandtl number. The $S_{\Theta_t}$ had a local minimum for the turbulent Prandtl number value of 0.5. It increased with a much greater rate by decreasing the turbulent Prandtl number in comparison with increasing it. There was approximately 0.3% averaged difference between the absolute computa-
tional and experimental total temperature values for this turbulent Prandtl number due to the enthalpy difference between the experiment and the CFD as explained in the beginning of the chapter.

Figure 6.9. Comparison between experimental measurements and computational analysis with $Pr_t = 0.5$ based on SST $k - \omega$ model, (a) Circumferentially averaged total pressure. (b) circumferentially averaged total temperature.

Figure 6.9 shows the comparison between experimental and computational data for turbulent Prandtl number value $Pr_t = 0.5$ based on the SST $k - \omega$ turbulence model. The numerical total temperature was similar to the experimental measurements while the total pressure was about 0.08% lower than the experimental measurements close to the wall as discussed in the previous section. This simulation achieved the normalized total temperature $\Theta_t$ values that matched the shape of the experimental measurements closely as expected. It could be of interest to note that the absolute total temperature from the computational solution could be matched to the absolute experimental values, by matching the computational enthalpy flux at the computational inlet to the experimental enthalpy flux at the exit measurement plane.

The constant value of 0.5 for the turbulent Prandtl number can be used to evalu-
Figure 6.10. Comparison between various turbulent Prandtl number models

date the turbulent Prandtl number models introduced in the chapter. Figure 6.10 shows the comparison between the averaged turbulent Prandtl number values from various models and the constant value of 0.5 calculated based on the CFD and experimental measurements. Computational data from the exit plane were used in calculating the turbulent Prandtl number values from these models. Dunn’s and both Kay’s models predicted a turbulent Prandtl number of close to 0.85. Aoki’s model under-predicted the turbulent Prandtl number by approximately 60%. Graber’s model predicted a value close to 1. Diessler and Dunn predicted turbulent Prandtl number values of over 1. None of these models were capable of predicting the analogy between momentum and heat transfer in this problem. Thus, it is of interest to develop a turbulent Prandtl number model that is not only capable of capturing the physics of this problem but is also applicable in other types of flows as well.
CHAPTER 7

THEORETICAL APPROACH TO TURBULENT PRANDTL NUMBER

Majority of the analytical models for turbulent transport processes calculate the eddy diffusivities of momentum, heat, or passively convected matter. From such calculations, the turbulent Prandtl number can be calculated as the ratio of eddy viscosity to the eddy diffusivity of heat. A significant group of the analytical models use the mixing length concept as the basis for approaching the turbulent Prandtl number modeling. Such analytical models seek to account for the exchange of momentum and heat between the instantaneous surroundings and an element of fluid which moves across gradients of velocity and temperature. This element mixes with the surrounding fluid at the end of its movement. Hence, the amount of the transferred thermal energy or momentum is calculated using the appropriate properties of the moving element at the end of its movement, accordingly.

The early studies based on the mixing length theory did not account for the loss of heat or momentum through the movement of the element ([26], [20], and [41]). The thermal and momentum mixing lengths were also presumed to be equal in those analyses. This led to $Pr_t = 1$ known as the Reynolds analogy, which is shown to be incorrect for most of gas and liquid metal flows. Jenkins [26] and Diessler [20] initiated the theoretical modeling of turbulent Prandtl number, and their works can still be considered as a strong starting point for studying this phenomenon. The model proposed by Jenkins considered the motion of a spherical element of fluid across the uniform velocity and temperature gradients, taking the radius of the particle to be equal to the mixing length. Deissler, on the other hand considered a stream of fluid
moving across uniform velocity and temperature gradients and accounts for heat transfer from a spherical element of fluid in the stream. Deissler’s model differed fundamentally from the Jenkins’s approach in taking the transfers to and from the fluid element to be controlled by diffusive processes external to the moving element rather than within it. Therefore, Deissler had more freedom in specifying the transfer mechanisms. It also gave the ability to the model to prescribe momentum transfer by a different process than heat transfer, although he did not investigate this aspect completely.

In the mixing length theory of turbulent transfer, a portion of the fluid is assumed to originate from a point in the fluid and travel transversely at a constant temperature and velocity due to the turbulent motions. The moving fluid particle has the mean temperature and momentum of the fluid appropriate to its point of origin in the flow. This portion of the fluid is considered to be a stream of fluid, all parts of which move with approximately the same velocity. It will travel a distance of one mixing length, and will mix with the fluid, assuming the average temperature and velocity at the location of mixing. Mixing length is defined as the distance which a particle travels across the flow before mixing with its new environment.

Adopting the Lagrangian point of view, a simple mathematical model for the diffusion action of turbulent mixing across the mean streamlines of a parallel flow can be developed as

\[
\frac{dP_y}{dt} = -\delta (P_y - \overline{P}).
\] (7.1)

This specifies the change of the general property \( P \) when a fluid particle moves through a distribution \( \overline{P}(y) \). The constant \( \delta \), relates the rate of change to the deviation of \( P_y \) from the local mean value which depends on the efficiency with which the property is exchanged with its surroundings by the molecular diffusion and other transfer mechanisms. The turbulent Prandtl number can be calculated by
first defining $\delta$ and then finding a solution to the equation \[7.1\] This will be discussed in this chapter.

7.1 Re-Derivation of Deissler’s Turbulent Prandtl Number Model

Two time averaged stream surfaces 1 and 2 are considered in the flow as shown in Figure 7.1 in order to define and solve the model equation \[7.1\]. The stream surfaces are separated by the small distance $l$, the average mixing length at the particular location in the flow. For example, if pipe flow is considered, stream surfaces are concentric cylindrical surfaces with the same axis as the pipe. The fluid particles traveling between the two stream surfaces can be modeled as a stream-tube originating from surface 1 due to the turbulent fluctuations and reaching the surface 2.

Deissler [20], proposed a turbulent Prandtl model by accounting for heat transfer from the stream-tube while moving from surface 1 to 2. In this case, the fluid element originates from the surface 1 where it has the time average temperature of the fluid at that surface, $T_1$ and arrives at the surface 2 with a lower temperature $T_{P2}$, where it mixes with the surrounding fluid assuming the time average temperature of surface

![Figure 7.1. Flow particle moving from stream surface 1 to stream surface 2](image-url)
2, $T_2$. The key assumption in this analysis is that, although the mean flow can be compressible, the turbulent fluctuating velocities are small enough to consider the turbulent mixing process as an incompressible phenomenon.

In order to find out the heat transported from the surface 1 to surface 2 by the turbulent motion, an infinitesimal control volume can be considered around the stream surface 2 as shown in Figure 7.2. The stream tube that originated from the stream surface 1 enters the control volume. A stream of fluid will leave the control volume in an overall steady state process.

![Figure 7.2. Control volume around the stream surface 2](image)

The continuity equation for the control volume is

$$\frac{\partial}{\partial t} \int_{CV} \rho dV + \oint_{CS} \rho \vec{v} \cdot \vec{n} \, dA = 0.$$  \hspace{1cm} (7.2)

The term $\frac{\partial}{\partial t} \int_{CV} \rho dV$ is zero for the steady state conditions considered in this case. Thus, the equation above can be written as

$$\dot{m}_m = \dot{m}_{out}.$$  \hspace{1cm} (7.3)
The integral form of the energy equation for this control volume is

\[ \dot{Q} - \dot{W} = \frac{\partial}{\partial t} \int_{CV} \rho e_C dV + \oint_{CS} \rho \left( h + \frac{1}{2}v^2 + gz \right) \mathbf{v} \cdot \mathbf{n} \, dA, \]  

which can be simplified as

\[ \frac{d (E_{CV})}{dt} = m_i (h_{in} - h_{out}) = \dot{m} c_p (T_P - T_2). \]  

Thus the thermal energy transported from surface 1 to surface 2 by the turbulent motion is

\[ Q_t = \dot{m} c_p (T_P - T_2) = \rho v' c_p A_{ps} (T_P - T_2), \]  

where \( \dot{m} \) is the mass flow rate in the stream tube, \( c_p \) is the specific heat, \( T_P \) is the temperature of flow particle reaching the surface 2. \( \dot{m} \) is defined as \( \rho v' A_{ps} \), where \( \rho \) is the fluid density, \( v' \) is a characteristic turbulent velocity normal to the stream surfaces, \( A_{ps} \) is the area that comes in contact with the stream-tube leaving surface 1. Hence, the turbulent heat transfer per unit area per unit time from surface 1 to 2 can be written as

\[ q_t = \rho v' c_p (T_P - T_2) \Rightarrow q_t = \rho v' c_p \phi_h (T_1 - T_2), \]  

where

\[ \phi_h = \frac{T_P - T_2}{T_1 - T_2}. \]  

Equation 7.7 is manipulated to replace the temperature difference between the particle and stream surface 2 with the difference between the mean temperatures of the two stream surfaces. This difference then can be replaced by the gradient of temperature among the two surfaces. The mixing length \( l \), is assumed to be small
enough to consider a linear change of temperature between the two stream surfaces. Thus, the gradient of the temperature between the two surfaces can be written as

$$\bar{T}_1 - \bar{T}_2 = -l\frac{dT}{dy}. \quad (7.9)$$

The non-dimensional term $\phi_h$ determines the ratio of the temperature difference between the particle when it reaches the stream surface 2 and the mean temperature of the stream surface 2 to the difference between the initial temperature of the particle and the mean temperature of stream surface 2. It defines the heat loss process illustrating with this analysis which can be better understood by studying the limits of it. $\phi_h = 1$ means that the temperature of the particle has not changed during the movement from surface 1 to 2 thus, bringing all of its energy to the stream surface 2. On the other hand, $\phi_h = 0$ occurs if the particle has already reached the temperature of stream surface 2 while arriving there thus, the particle has transferred all of its thermal energy to the surrounding fluid. The value of $\phi_h$ is expected to be between these extreme limits as the limits are not likely to represent the true physics of turbulent flow. Replacing the temperature gradient defined by equation 7.9, the equation 7.10 becomes

$$q_t = -\rho v' c_p \phi_h l \frac{dT}{dy}. \quad (7.10)$$

Equation 7.10 is the turbulent heat transfer per unit time per unit area in this model. In order to calculate the eddy diffusivity of heat from this equation, it is necessary to compare this modeled formulation with the thermal turbulent transfer defined by the energy equation. The time averaged energy equation is given by

$$\bar{u}_i \frac{\partial T}{\partial x_i} = \alpha \frac{\partial^2 T}{\partial x_i^2} - \frac{\partial}{\partial x_i} \left( u'_i T' \right), \quad (7.11)$$

where $\alpha$ is the coefficient of thermal diffusivity. The second term on the right hand
side of the equation is the turbulent transport of heat which can be simplified in the case of two dimensional turbulent flow described in section 1.1 as

\[
\frac{\partial}{\partial x_i} (-u'_iT') = \frac{\partial}{\partial y} (-v'T') ,
\]

(7.12)

where \( v' = u_2 \) and, \( \frac{\partial}{\partial x} (-u'T') = 0 \). Adopting the definition for the eddy diffusivity of heat \( (\epsilon_h) \), the above equation can be written as

\[
\frac{\partial}{\partial y} (-v'T') = \frac{\partial}{\partial y} \left( -\epsilon_h \frac{dT}{dy} \right).
\]

(7.13)

This defines the rate of turbulent diffusion of thermal energy in the direction normal to the stream-surfaces. Therefore, the turbulent transfer of thermal energy per unit time, per unit area over the mixing length based on equation 7.11 is given by

\[
q_t = \rho c_p (-v'T') = -\rho c_p \epsilon_h \frac{dT}{dy}.
\]

(7.14)

The amount of heat flux in equation 7.14 should be equal to the amount calculated by equation 7.10. Comparing these two equations the eddy diffusivity of heat can be defined as

\[
\epsilon_h = v' \phi_h l.
\]

(7.15)

Hence, the eddy diffusivity of heat in this model depends on the mixing length, the normal turbulent velocity component that can be defined using the physical flow properties. The remaining unknown in the equation 7.15 is the non-dimensional number \( \phi_h \). In order to define this ratio, it is essential to find out the temperature of the particle when it reaches the stream surface 2. The differential element of fluid as shown in Figure 7.1 is considered to calculate \( T_{P_2} \). Heat is transferred out of the
element while it is moving between the stream surfaces by convection. The amount of heat transfer from the element can be related to the change in the temperature of the particle as

\[ h' A_{pp} (T_P - \overline{T}) \, dt = -\rho V_p c_p \frac{dT_P}{dt} \, dt, \tag{7.16} \]

where \( h' \) is the convective heat transfer coefficient, \( A_{pp} \) is the surface of the differential fluid element, \( T_p \) is the particle temperature, \( T \) is the temperature of fluid surrounding the fluid element, and \( V_p \) is the volume of the differential element of the fluid. \( v' \) is assumed to be constant through the time of flight thus, \( dt = \frac{ds}{v'} \). The equation 7.16 becomes

\[ h' A_{pp} (T_P - \overline{T}) = -\rho V_p c_p v' \frac{dT_P}{ds}. \tag{7.17} \]

The constants in this equation can be grouped together to define a new constant as defined below,

\[ \delta_h = \frac{h' A_{pp}}{\rho c_p V_p v'}. \tag{7.18} \]

Therefore, the equation 7.17 can be written as

\[ \frac{dT_P}{ds} = -\delta_h (T_P - \overline{T}). \tag{7.19} \]

The ratio between the volume and the surface area of the element \( \frac{V_p}{A_{pp}} \) can be replaced by a new length scale \( d \) representing a diameter for the element. Equation 7.18 becomes

\[ \delta_h = \frac{4 h'}{\rho c_p v'd} = \frac{4 k \left( \frac{h'd}{T} \right)}{\rho c_p v'd^2} = \frac{4 k N u}{\rho c_p v'd^2}. \tag{7.20} \]

Note that the equation 7.19 is comparable to the equation 7.1. Deissler assumed a
laminar heat transfer from the stream tube to the surrounding flow. Therefore, he stated that \( h' \) is proportional to \( k/\delta \), where \( \delta \) is the thermal boundary layer thickness around the stream tube.

Utilizing the linear temperature change assumption made earlier, the flow temperature surrounding the fluid element at any location \( s \) can be derived by \( \bar{T} = \bar{T}_1 + s \frac{dT}{dy} \). The differential equation governing the temperature of the fluid element is given by

\[
\begin{align*}
\frac{dT_P}{ds} + \delta_h T_P &= -\delta_h \left( \bar{T}_1 + s \frac{dT}{dy} \right) \\
T_P (s = 0) &= \bar{T}_1.
\end{align*}
\] (7.21)

This equation was solved to find \( T_{P_2} = T_P (s) \mid_{s=l} \), and as a result \( \phi_h \) was calculated as

\[
\phi_{h_0} = \frac{1}{\delta_h l} \left( 1 - e^{-\delta_h l} \right). \tag{7.22}
\]

The eddy diffusivity of heat can be derived based on equations 7.15 and 7.22 as

\[
\epsilon_{h_0} = \frac{v' l}{\delta_h l} \left( 1 - e^{-\delta_h l} \right). \tag{7.23}
\]

In order to calculate the turbulent Prandtl number, the eddy diffusivity of momentum needs to be defined in the same manner as the eddy diffusivity of heat. The turbulent shear stress can be obtained from the momentum equation. The time-averaged momentum equation is

\[
\rho U_j \frac{\partial U_i}{\partial x_j} = \rho f_i + \frac{\partial}{\partial x_j} \left[ -p \delta_{ij} + \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \rho u'_i u'_j \right]. \tag{7.24}
\]

The turbulent shear stress in this case can be written as

\[
- \rho \bar{u'} \bar{v'} = - \rho \epsilon_m \frac{\partial \bar{U}}{\partial y}. \tag{7.25}
\]
Diessler [20] did not account for the momentum transfer from the stream-tube moving between the two surfaces. He proposed that the momentum equation can be written as

\[ M_t = -\rho v' l \frac{dU}{dy}. \]  

(7.26)

Comparing equations 7.26 and 7.25, the eddy viscosity for momentum can be computed using

\[ \epsilon_m = v' l. \]  

(7.27)

Thus, Deissler’s turbulent Prandtl number can be defined as

\[ Pr_{t, Deissler} = \frac{\epsilon_m}{\epsilon_h} = \frac{v' l}{v' l \phi_{h_0}} = \frac{1}{\phi_{h_0}}. \]  

(7.28)

The non-dimensional number \( \phi_{h_0} \) is found earlier in equation 7.23, thus,

\[ Pr_{t, Deissler} = \frac{\delta_h l}{1 - e^{-\delta_h l}}. \]  

(7.29)

Deissler’s analysis stopped at equation 7.29. He found the numerical value of constant \( \delta_h l \) as a function of the mean flow Peclet number \( Pe \), based on the experimental data.

7.2 Addition of Momentum Loss (\( \phi_m \)) to Diessler’s Model

The most important shortcoming of the model explained in section 7.1 is the lack of accounting for the momentum change of the particle in the time of movement in the same manner as the thermal energy. A fluid element shown in Figure 7.3 is considered to investigate modeling the momentum change of the particle. The fluid particle leaves the stream-surface 1 with the momentum appropriate to that surface.
characterized by the mean axial velocity $U_1$. It reaches the second stream surface with an axial velocity $U_{P_2}$ which is lower than $U_2$.

![Figure 7.3. Flow particle moving from stream surface 1 to stream surface 2](image)

The momentum transported by eddy between the two surfaces is given by

$$\dot{m} (U_{P_2} - U_2) = \rho v' \phi_m A_{ps} (U_1 - U_2), \hspace{1cm} (7.30)$$

where

$$\phi_m = \frac{U_{P_2} - U_2}{U_1 - U_2}. \hspace{1cm} (7.31)$$

Equation (7.30) is rearranged so that the velocity difference between the particle and stream surface 2 is replaced by the difference between the mean axial velocities of the stream surface 1 and 2 which can be inferred from the flow. This velocity difference can be replaced by the mean flow velocity gradient. A non-dimensional number $\phi_m$ is introduced here in the same manner as the $\phi_h$ for the thermal energy transport. It defines the ratio between the difference between the particle’s terminal axial velocity and the mean axial velocity of the stream surface 2 to the difference between the particle’s initial velocity and the mean axial velocity of the stream surface 2. Hence,
\( \phi_m = 1 \) represents the condition where the axial velocity of the particle has not changed during the movement from surface 1 to 2, bringing all of its momentum to the stream surface 2. On the other hand, \( \phi_m = 0 \) occurs if the particle has already transferred all of its momentum during the movement thus, reaching the stream surface 2 with the same axial velocity appropriate to that surface. The value of \( \phi_m \) is expected to be in between these extreme limits analogous to the arguments made regarding \( \phi_h \).

The assumption that axial mean velocity is changing linearly between the surfaces can be made here the same as temperature. This translates in to \( U_1 - U_2 = -l \frac{dU}{dy} \). Hence, the turbulent momentum transfer per unit time per unit area is

\[
M_t = -\rho v' \phi_m l \frac{dU}{dy}. \tag{7.32}
\]

Comparing equations 7.26 and 7.25 the eddy viscosity for momentum can be computed using

\[
\epsilon_m = v'l \phi_m. \tag{7.33}
\]

Hence, using the eddy diffusivity of heat defined by the equation 7.15, the turbulent Prandtl number can be defined as

\[
Pr_t = \frac{\epsilon_m}{\epsilon_h} = \frac{v'l \phi_m}{v'l \phi_h} = \frac{\phi_m}{\phi_h} \tag{7.34}
\]

In order to model the momentum loss characterized by \( \phi_m \), the fluid element shown in Figure 7.3 is considered again. The difference between the velocity of particle and the surrounding flow while the particle is moving between the stream surfaces cause a drag force acting on the fluid particle. The velocity of the particle \( U_P \), can be derived by balancing the momentum change in the particle and the drag force acting on the element of fluid. This balance can be written as
\[ F_d = ma \Rightarrow -\frac{1}{2} \rho C_d A_s \left( U_P - \overline{U}(y) \right)^2 = m \frac{dU_P}{dt}. \]  

(7.35)

Recounting the assumption that \( v' \) is constant for the fluid element, \( dt = \frac{ds}{v'} \), the above equation can be rewritten as

\[-\frac{1}{2} \rho C_d A_s \left( U_P - \overline{U}(y) \right)^2 = \rho V_p v' \frac{dU_P}{ds}, \]  

(7.36)

where

\[ \delta_m = \frac{C_d A_s}{2v' V_p}. \]  

(7.37)

Equation 7.36 can be simplified as

\[ \frac{dU_P}{ds} = -\delta_m (U_P - \overline{U}), \]  

(7.38)

The drag coefficient \( C_d \) for a cylinder in the flow can be calculated from the equation given by Cheng [12] as

\[ C_D = 11 \text{Re}^{-0.75} + 0.9 \left[ 1 - e^{-\frac{1000}{\text{Re}}} \right] + 1.2 \left[ 1 - e^{-\left( \frac{\text{Re}}{3000} \right)^{0.7}} \right]. \]  

(7.39)

The mixing length is small enough to assume a linear change in velocity as,

\[ \overline{U}(s) = U_1 + s \frac{d\overline{U}}{ds}. \]  

(7.40)

Hence, the difference between the axial velocity in the two stream surfaces can be found by
\[ \overline{U}_1 - \overline{U}_2 = -l \frac{d\overline{U}}{dy}. \]  

Thus, the differential equation governing the velocity of this fluid particle is

\[
\begin{cases}
\frac{dU_P}{ds} = -\delta_m (U_p - U)^2 \\
U_P(s = 0) = \overline{U}_1.
\end{cases}
\]  

(7.42)

The above equation can be solved to find \( U_P|_{s=l} \) and as a result, \( \phi_m \) for the modified Deissler’s model can be evaluated as

\[
\phi_m = \frac{-1 + e^{2(\delta_m \frac{d\overline{U}}{dy})^\frac{1}{2} l}}{(\delta_m \frac{d\overline{U}}{dy})^\frac{1}{2} l \left[ 1 + e^2 (\delta_m \frac{d\overline{U}}{dy})^\frac{1}{2} l \right]^\frac{1}{2}}.
\]  

(7.43)

Hence, equation 7.29 becomes

\[
Pr_{t,\text{ModDeis}} = \left[ \frac{\delta_h l}{(\delta_m \frac{d\overline{U}}{dy})^\frac{1}{2} l} \left[ -1 + e^{2(\delta_m \frac{d\overline{U}}{dy})^\frac{1}{2} l} \right] \left( \frac{1}{1 - e^{-\delta_h l}} \right) \right].
\]  

(7.44)

The modified Deissler’s turbulent Prandtl number model presented in equation 7.44 should provide more realistic predictions comparing to his original model given by equation 7.29.

7.3 Multi-Scale Turbulent Prandtl Number Model

7.3.1 Intermediate Mixing Length

In the model constructed so far, the mixing process can be modeled as two different phenomena. One involves the transfer of a property by the large scale turbulent motions. The other one is defined by the small scale turbulent diffusion of the prop-
erty in the stream surfaces. The first mechanism occurs while the particle is moving and the other where the particle mixes in the stream surfaces. For example, in the case of turbulent transport of heat the first mechanism is the convective heat transfer which depends on the turbulent Reynolds number around the stream tube. The two physical phenomena are considered to happen simultaneously. While this may sound redundant, they will both be shown to be necessary for this analysis. The modified Deissler model can be expanded by deviating from the classic definition of the mixing length theory. An intermediate mixing length can be introduced where the time scale of turbulent diffusion in a designated stream surface is larger than the time scale of the large scale turbulent motions. Therefore, it can be argued that the fluid particle may leave a stream surface that it reaches, before assuming the mean temperature appropriate to that surface. This can happen where there is high turbulence taking into account both transfer mechanism explained earlier. For example, a differential element of fluid leaves stream surface 1 with the mean temperature $T_1$ appropriate to that level. It arrives to the stream surface 2 with a temperature of $T_{P2}$ and reaches an intermediate temperature $T_{int}$, before leaving surface 2 toward stream-surface 3. This analysis accounts for the history of the particles moving between the two surfaces. It can be shown that considering more than three surfaces does not impact the end result of turbulent Prandtl number significantly. This process is shown in Figure 7.4.

7.3.2 Modeling the Multi-Scale Turbulent Prandtl Number

Assuming that the mass flow rate through the stream tube moving between the stream surfaces 1 and 2 is equal to the mass flow rate of the stream tube moving between surface 2 and 3, the total turbulent heat transfer is

$$ dq_t = \dot{m}c_p \left[ (T_{P2} - T_{int}) + (T_{P3} - T_3) \right]. $$

(7.45)
Having \( T_2 - T_3 = T_1 - T_2 \), the above equation can be expanded as

\[
dq_t = \rho c_p v' \left[ \left( \frac{T_{P_2} - T_{int}}{T_1 - T_2} \right) + \left( \frac{T_{P_3} - T_3}{T_1 - T_2} \right) \right] (T_1 - T_2).
\] (7.46)

Using equation 7.9, the amount of heat transfer per unit area per unit time can be calculated from

\[
dq_t = -\rho c_p v'l (\phi_{h_{12}} + \phi_{h_{23}}) \frac{dT}{dy},
\] (7.47)

where

\[
\phi_{h_{12}} = \frac{T_{P_2} - T_{int}}{T_1 - T_2},
\] (7.48)

\[
\phi_{h_{23}} = \frac{T_{P_3} - T_3}{T_1 - T_2},
\] (7.49)

and

\[
T_{int} = T_{P_2} + n_h (T_2 - T_{P_2}), \quad 0 \leq n_h \leq 1.
\] (7.50)
If $n_h = 0$, the fluid particle temperature remains constant through its stay in the stream surface 2. This represents the case that the large scale turbulent motion time scales are much faster than the small scale turbulent diffusion time scales thus, the particle does not stay at surface 2 for enough time to exchange thermal energy with the surrounding fluid. On the other hand, if $n_h = 1$, the particle is completely mixed with the surrounding fluid in the surface 2 assuming the appropriate temperature of that surface, which is discussed before. Based on equation 7.14, the eddy diffusivity of heat in this case can be defined as

$$
\epsilon_{h_B} = v' I (\phi_{h_{12}} + \phi_{h_{23}}).
$$

The stream-tube shown in Figure 7.4 leaves stream-surface 1 with the average mean axial velocity $U_1$ appropriate to that level. It reaches the second surface with a mean axial velocity $U_{P_2}$, and reaches an intermediate velocity of $U_{int}$ before leaving toward surface 3. The intermediate velocity can be modeled as

$$
U_{int} = U_{P_2} + n_m \left( \frac{U_2}{U_1} - U_{P_2} \right),
$$

$$
0 \leq U_{int} \leq 1.
$$

If $n_m = 0$, it means that the particle does not mix at the second surface at all and leaves it instantly toward the third surface. On the other hand, if $n_m = 1$, the particle stays at stream-surface 2, and completely mixes with the surrounding fluid, assuming its mean velocity. The amount of momentum transferred by this fluid particle to the surrounding flow having $U_1 - U_2 = U_2 - U_3$ can be written as

$$
dM_t = \rho v' \left[ \left( \frac{U_{P_2} - U_{int}}{U_1 - U_2} \right) + \left( \frac{U_{P_3} - U_2}{U_2 - U_3} \right) \right] (U_1 - U_2).
$$

The momentum transfer per unit time per unit area with the assumption that

95
axial velocity changes linearly between the considered stream surfaces is given by

\[ dM_t = -\rho v' l \left( \phi_{m12} + \phi_{m23} \right) \frac{dU}{dy}. \]  

(7.54)

Based on the definition of turbulent shear stress in equation 7.25, the eddy viscosity for this case is given by

\[ \epsilon_{mB} = v' l \left( \phi_{m12} + \phi_{m23} \right). \]  

(7.55)

Hence, the turbulent Prandtl number is

\[ Pr_{t,MS} = \frac{\epsilon_{mB}}{\epsilon_{hB}} = \frac{v' l \left( \phi_{m12} + \phi_{m23} \right)}{v' l \left( \phi_{h12} + \phi_{h23} \right)} = \frac{\phi_{m12} + \phi_{m23}}{\phi_{h12} + \phi_{h23}}. \]  

(7.56)

The sets of differential equations defining the temperature of the flow element through movements between the three stream surfaces are derived the same as the two surface model. The temperature of the particle between surface 1 and 2 is defined as,

\[ \frac{dT_P}{ds} + \delta_h T_P = \delta_h \left( \frac{T_1}{1} + s \frac{dT}{dy} \right) \]

(7.57)

\[ T_P (s = 0) = T_1, \]

and for movement from stream-surface 2 to 3,

\[ \frac{dT_P}{ds} + \delta_h T_P = \delta_h \left( \frac{T_1}{1} + (s + l) \frac{dT}{dy} \right) \]

(7.58)

\[ T_P (s = 0) = T_{int}. \]

The temperature of fluid element at surfaces 2, and 3 can be derived by solving the above set of equations. Hence, the eddy diffusivity of heat can be calculated based on these known temperatures.

The momentum transfer per unit time per unit area is given by equation 7.54. The
eddy viscosity can be derived as defined by equation 7.55. The momentum loss can be analyzed in the same fashion as the previous case by considering a drag force acting on the fluid element moving from one stream-surface to the other. The differential equation defining the particle velocity through the time of flight from surface 1 to 2 is

\[
\begin{align*}
\frac{dU_p}{ds} - \delta_M \left[ U_p - \left( U_1 + s \frac{dU}{dy} \right) \right]^2 &= 0 \\
U_p(s = 0) &= U_1,
\end{align*}
\]

(7.59)

and for movement from stream-surface 2 to 3,

\[
\begin{align*}
\frac{dU_p}{ds} - \delta_M \left[ U_p - \left( U_1 + (s + l) \frac{dU}{dy} \right) \right]^2 &= 0 \\
U_p(s = 0) &= U_{int}.
\end{align*}
\]

(7.60)

The velocity of the particle reaching each stream-surface can be found by solving the above set of differential equations. Thus, the eddy viscosity can be calculated and the turbulent Prandtl number in this case would be given by

\[
Pr_{t,MS} = \left[ \frac{\delta_h l}{\left( \delta_m \frac{dU}{dy} \right)^{\frac{3}{2}} l} \right] \frac{e^{2\delta_h l}}{\left( (1 - n_h) + (1 + n_h) e^{\delta_h l} \right) \left( -1 + e^{\delta_h l} \right)} \left\{ \begin{array}{c} n_m \left[ -1 + e^{2\left( \delta_m \frac{dU}{dy} \right)^{\frac{1}{2}} l} \right] \\
1 + e^{2\left( \delta_m \frac{dU}{dy} \right)^{\frac{1}{2}} l} \\
1 + e^{4\left( \delta_m \frac{dU}{dy} \right)^{\frac{1}{2}} l} \end{array} \right\} \left[ (n_m - 2) \left[ -1 + e^{4\left( \delta_m \frac{dU}{dy} \right)^{\frac{1}{2}} l} \right] \right] + \left[ (n_m - 2) - 2n_m e^{2\left( \delta_m \frac{dU}{dy} \right)^{\frac{1}{2}} l} \right].
\]

(7.61)

This equation should provide more realistic predictions of the turbulent Prandtl number compared to Deissler’s original and modified models.
7.3.3 Non-Dimensional Terms in Turbulent Prandtl Number Model Equations

The analysis presented in the previous sections can be further investigated by introducing non-dimensional terms instead of the unknown constant. The repeating term $\delta_h l$ in the turbulent Prandtl number models can be rearranged as

$$
\delta_h l = 4 \left( \frac{k}{\mu c_p} \right) \left( \frac{\rho \nu}{\rho^2 d} \right) \left( \frac{l}{d} \right) Nu(Re_w) = 4 \left( \frac{1}{Re_{\infty} Pr} \right) \left( \frac{l}{d} \right) Nu(Re_w).
$$

(7.62)

where, the Nusselt number $Nu$, for a cylinder in the flow can be calculated from the equation given by Churchill [16] as

$$
Nu_d = 0.3 + \frac{0.62 Re_{\infty}^{\frac{5}{8}} Pr^{\frac{1}{3}}}{1 + \left( \frac{0.4}{Pr} \right)^{\frac{3}{4}}} \left[ 1 + \left( \frac{Re}{282000} \right)^{\frac{5}{8}} \right]^{-\frac{4}{5}}.
$$

(7.63)

Therefore, Deissler’s turbulent Prandtl number model given by equation [7.29] can be rewritten in terms of non-dimensional numbers as

$$
Pr_{t, nondies} = 4 \left( \frac{1}{Re_{\infty} Pr} \right) \left( \frac{l}{d} \right) Nu(Re_w) \left( \frac{1}{1 - e^{-4 \left( \frac{1}{Re_{\infty} Pr} \right)^{\frac{1}{3}} Nu(Re_w)}} \right).
$$

(7.64)

This is slightly different from Deissler’s approach as he did not consider the heat transfer from the stream tube to be a Reynolds number dependent mechanism. Representing Diessler’s turbulent Prandtl number model [7.29] in terms of non-dimensional numbers makes it possible to evaluate his model without using the model constants. The constants are limited to the experimental measurements and the uncertainties involved with the measurements which were used to define them in Deissler’s analysis. The non-dimensional numbers can be define in any type of flow either experimentally
or numerically to evaluate this turbulent Prandtl number model.

The recurring term in the equation 7.44 is \((\frac{dU}{dy} \delta_m)\frac{1}{2} l\). This term can expanded having \(\delta_m\) defined by equation 7.37, and \(\frac{V_P}{A_o} = d\) as

\[
\left(\frac{dU}{dy} \delta_m\right) \frac{1}{2} l = \left[ \frac{C_d (l) (d) (ds)}{2\pi (d^2) (ds)} \left(\frac{dU}{dy}\right) l^2 \right]^{\frac{1}{2}}. 
\]

Equation 7.65 can be rearranged in terms of non-dimensional numbers as

\[
\left(\frac{dU}{dy} \delta_m\right) \frac{1}{2} l = \left[ \left(\frac{1}{2\pi}\right) \left(\frac{l}{d}\right) C_d (Re_{u'}) \left(\frac{l}{v'} \frac{dU}{dy}\right) \right]^{\frac{1}{2}}. 
\]

Therefore, the turbulent Prandtl number defined by equation 7.44 can be written in terms of non-dimensional numbers as

\[
Pr_t,ModDeis = \left\{ \frac{4 \left(\frac{1}{Re_{u'} Pr}\right) \left(\frac{l}{d}\right) Nu (Re_{u'})}{\left(\frac{l}{2\pi} \left(\frac{l}{v'} \frac{dU}{dy}\right) C_d (Re_{u'})\right)^{\frac{1}{2}}} \right\} \left\{ \left(\frac{-1 + e^2 \left[\frac{1}{2\pi} \left(\frac{l}{d}\right) \frac{dU}{dy}\right] C_d (Re_{u'})}{1 + e^2 \left[\frac{1}{2\pi} \left(\frac{l}{d}\right) \frac{dU}{dy}\right] C_d (Re_{u'})}\right)^{\frac{1}{2}} \right\} \left[ \frac{1}{1 - e^{-4 \left(Re_{u'} Pr\right) \left(\frac{l}{d}\right) Nu (Re_{u'})}} \right]. 
\]

Lastly, the multi-scale turbulent Prandtl number model defined in equation 7.61 can also be rearranged in terms of non-dimensional numbers as
These non-dimensional numbers can be divided into two groups. The first group can be calculated based on the properties of the fluid flow which include \( Re_{v'0} \), \( Pr \), \( Nu(Re_{v'}) \) and, \( C_d(Re_{v'}) \). While the second group consists of the numbers that are exclusive to the model including \( l \), \( n_m \), \( n_h \) and, \( l v_0 dy \). Noting that \( l v_0 dy \), can be calculated from the flow measurements however, \( l \), the mixing length is chosen to be equal to the turbulent integral length scale.

In order to investigate how the proposed equation depends on the values of \( n_m \) and, \( n_h \), a typical turbulent flow of air is considered where \( Pr = 0.7 \), turbulence intensity is 5\%, and the turbulence isotropy indicator of \( \frac{v'}{u'} = 0.6 \). Recalling that \( n_m, n_h = 0 \) is the extreme case that the flow particle does not mix with the surrounding fluid in the mid-stream surface at all; and \( n_m, n_h = 1 \) is the other extreme case where the particle completely mixes in the stream surface 2 and does not leave the mid-plane. It is sensible to neglect the extremes and investigate the effects of \( n_m \) and \( n_h \) somewhere.
between the extremes. Figure 7.5 shows the turbulent Prandtl number normalized by the maximum turbulent Prandtl number value versus $0.2 \leq n_m, n_h \leq 0.8$. These results are independent of the Reynolds number. The turbulent Prandtl number’s value changes approximately $\pm 10\%$ for the range of $n_m$ and $n_h$. The maximum value of the turbulent Prandtl number occurs where $n_m$ has the maximum considered value and $n_h$ has the minimum value. The minimum turbulent Prandtl number value occurs where $n_m$ has the minimum considered value and $n_h$ has the maximum value.

![Figure 7.5. Variation of the turbulent Prandtl number model with $n_m$ and $n_h$.](image)

The process of turbulent diffusion in a stream surface is expected to be relatively similar for both momentum and thermal energy thus, it can be expected that the values of $n_h$ and $n_m$ should be close to each other. Therefore the edges of the contour plot show in Figure 7.5 can be neglected where the values of $n_h$ and $n_m$ are drastically different. It can be inferred that by choosing $n_m$ and $n_h$ to be 0.5, the model should predict the turbulent Prandtl number with an acceptable uncertainty. The value of 0.5 can be considered as a constant value for $n_h$ and $n_m$ from now on in this model.

Assuming that the values for all the non-dimensional terms explained above, are known in a specific flow, $\frac{l}{d}$ becomes the only independent variable in equation 7.68.
The non-dimensional \( \frac{l}{d} \) defines the ratio of the distance between the stream surfaces and the diameter of the stream tubes moving between the surfaces. \( l \) is chosen to be the turbulent integral length scale of the flow however, \( d \) is not specifically defined in this model. Thus, \( \frac{l}{d} \) is considered as an independent non-dimensional value for evaluating this model. The functional dependency of equation 7.68 on \( \frac{l}{d} \) is examined here.

The turbulent Prandtl number model has an absolute minimum value at \( \frac{l}{d}|_{\text{min}} > 1 \). It increases rapidly by increasing \( \frac{l}{d} \) and tends to a value of 1 when \( \frac{l}{d} \) approaches zero. Large values of \( \frac{l}{d} \gg \frac{l}{d}|_{\text{min}} \) represent a case where the diameters of the stream tubes are significantly smaller than the integral length scale. These values produce unphysically large turbulent Prandtl number values as the first term in equation 7.68 defines the shape of the proposed function in this range. This term is proportional to \( (\frac{l}{d})^\frac{1}{2} \) which increases rapidly by increasing \( \frac{l}{d} \). Also, for very large values of \( \frac{l}{d} \), the stream tube diameter \( d \) becomes very small which decreases the turbulent Reynolds number for the stream-tube. The drag coefficient \( C_d (Re_{u'}) \), which was defined by equation 7.39 in this model is not valid for low Reynolds numbers as it produces large unphysical values (see Figure 7.6).

![Figure 7.6. Variation of drag coefficient with turbulent Reynolds number](image)

102
On the other hand, the proposed turbulent Prandtl number model does not make correct predictions for $\frac{l}{d} << 1$. This range represents a case where the diometer of the stream tube is much larger than the integral length scale separating the stream surfaces. In this case, the fluid element considered in the analysis presented in this chapter would have a very large surface area. Thus, the amount of thermal energy and momentum transferred out of or into this element would be unphysically large. If $\frac{l}{d}$ is sufficiently small, then $\phi_m \approx \phi_h << 1$, and the fluid element is always in approximate equilibrium with its surrounding. This invalidates the initial assumption in the analysis that the turbulent mixing was modeled as two different phenomenons. The proposed turbulent Prandtl number model is evaluated in various flows in the next section.

7.3.4 Validation

Different flow types were chosen to evaluate the multi-scale turbulent Prandtl number model. The investigated cases include the experimental measurements obtained in the Notre Dame annular nozzle facility, inner turbulent boundary layer of flow over a flat plate, turbulent jet flow and, turbulent pipe flow. The non-dimensional terms in the equation 7.68 were calculated using the experimental measurements provided for each case of study. The proposed model was then investigated over the range of $1 \leq \frac{l}{d} \leq 10$ to find the absolute minimum value of turbulent Prandtl number at $\frac{l}{d}-min$ in each case. This minimum value was then compared to the experimentally measured turbulent Prandtl number in each flow type.

The experiments explained in chapter 4 present a case for evaluating the proposed model. It is an accelerated flow with averaged inlet turbulence intensity of 16%, $\frac{dC}{dy}$ of 120 1/s, and $\left\langle u'^2 \right\rangle^{1/2} = 5$ m/s. The mixing length was chosen to be the integral length scale calculated from the experimental hotwire measurements. Figure 7.7 shows the comparison between the proposed turbulent Prandtl model based on the
above measurements and the computational result for turbulent Prandtl number from chapter 5. These results are presented for a range of $1 \leq \frac{l}{d} \leq 10$. The proposed turbulent Prandtl number model provides a value of 0.5 for $\frac{l}{d} \approx 4.75$ where the $\frac{l}{d}$ that minimizes the turbulent Prandtl number value. There is a good agreement between the proposed model and the numerical study. Thus, it can be argued that the model can capture the turbulent value of Prandtl model when the turbulent transfer of heat is much greater than of the momentum.

Snijders et al. [64] and Hishida [1] studied the turbulent Prandtl number in the inner boundary layer of a flat plate separately. The non-dimensional numbers in the proposed model were calculated based on the experimental data obtained by Snijders. Figure 7.8 shows the comparison between the experimentally measured turbulent Prandtl number by Snijders and the proposed theoretical model. Snijders measured a turbulent Prandtl number value of 0.9 ± 0.1 which is shown by the red box. Hishida measured a turbulent Prandtl number value of 0.9 ± 0.02 which is shown by the blue box. The range for $\frac{l}{d}$ considered here is the same as the range in the previous comparison. The minimum value of turbulent Prandtl number which is 0.89 occurred at $\frac{l}{d} = 1.4$. The agreement between the predicted value and both
measurement shows that the model is capable of capturing the value of turbulent Prandtl number when the turbulent transfer mechanisms of momentum and heat are relatively similar.

![Comparison between the proposed turbulent number and Snijders' experimental measurements](image)

Figure 7.8. Comparison between the proposed turbulent number and Snijders’ experimental measurements

Turbulent jet flow was investigated as another type of flow for evaluating the proposed model. Chua [14] experimentally studied the turbulent Prandtl number in a circular jet and measured a turbulent Prandtl number value of 0.625 ± 0.2. Figure 7.9 shows the comparison between Chua’s results and the proposed model. The red box indicates the uncertainty bounds of Chua’s measurements. The minimum predicted turbulent Prandtl number is 0.65 which occurs at $\frac{l}{d} = 3.4$. It can be seen from this comparison that the model can estimate the value of turbulent Prandtl number in this case where the turbulent heat transfer is more than the turbulent momentum transfer.

Ludwig [40] studied the turbulent pipe flow and experimentally measured the turbulent Prandtl number along the pipe radius. Detailed experimental measurements obtained by Laufer [33] in turbulent pipe flow, were used to evaluate the proposed turbulent Prandtl model against Ludweig’s measurements. In order to calculate $\frac{dT}{dy}$ for the turbulent Prandtl number model a third order function was fitted to the
experimental measurements of $\overline{U}$ versus $y$.

Figure 7.10 shows the comparison between the proposed model and the experimental measurements across the radius of the pipe. The abscissas represents the distance from the pipe wall normalized by the pipe radius. The value of $\frac{l}{d}$ for each point was chosen at the point that the turbulent Prandtl number value is minimum. The proposed model can predict the turbulent Prandtl within 3% accuracy over the radius of the pipe except closest to the wall. There is 7% deviation from the model prediction and the measured turbulent Prandtl number close to the wall. This was due to the uncertainty in the experimental measurements used for this prediction. The experimental data closest to the wall provided by Laufer was scarce and some of the needed values were not even measured close to the wall. The comparison shown in Figure 7.8 gives a better example to evaluate the predictions of the proposed model for flow close to the solid walls. There is a local maxima in the predicted turbulent Prandtl number values at $\frac{r}{R} = 0.82$. This was due to the third order fitted function explained above. The third order function fitted to experimental measurements of $\overline{U}$ will have a second order $\frac{d\overline{U}}{dy}$ function thus, the local maximum. Overall, this com-
parison shows that the model is capable of predicting the correct turbulent Prandtl number across the pipe’s radius.

\[ Pr_t \]

![Graph showing comparison between the model and Ludweig's experimental measurements in turbulent pipe flow](image)

Figure 7.10. Comparison between the proposed turbulent number and Ludweig’s experimental measurements in turbulent pipe flow
Turbulent flow through a nozzle with high inlet turbulence intensity and high temperature gradients was studied both experimentally and computationally in this work. Experimental measurements were obtained at the nozzle inlet and exit. These included total temperature, total pressure and hotwire measurements. Hotwire measurements were obtained at two inlet measurement planes separated by 23 mm distance from each other so that, the turbulence decay could be calculated experimentally.

In order to obtain the hotwire measurements in this flow with high temperature gradients, a novel calibration technique was developed. This method allowed for a hot-wire calibration obtained at a constant temperature to be applied to a wide range of ambient temperature, varying within ±35K. It is expected that this method can be applied to ranges beyond the temperature range of interest in this study. It was shown that if total temperature is used in the hot-wire calibration equation, the uncertainties in the calibration due to Mach number sensitivity can also be minimized.

A computational geometry identical to the experimental nozzle was developed. Experimental measurements were used to define the numerical boundary conditions in evaluating the effectiveness of various turbulence models in this accelerated highly turbulent flow. Over the range of the turbulence models utilized in this study, the SST $k$ – $\omega$ model performed the best.

SST $k$ – $\omega$ model was used to study the turbulent Prandtl number in this flow. Turbulent Prandtl number defines the relationship between the eddy diffusivity of
heat and eddy viscosity in the turbulence model thus, defining the analogy between the turbulent transport mechanisms of thermal energy and momentum. It was concluded that the turbulent Prandtl number of 0.5 defines the aforementioned analogy in this flow. This means that the turbulence transports thermal energy more rapidly than the momentum. This was due to the inlet total temperature distribution for the non-uniform case of study. The numerical simulations with turbulent Prandtl number of 0.9 matched the experimental measurements for the uniform inlet total temperature distribution case. It can be inferred from the simulations that, the transport mechanism of thermal energy and momentum are relatively similar to each-other with thermal transport being slightly more than momentum for the uniform case.

An analytical model for the turbulent Prandtl number was developed based on the intermediate mixing length concept. The model predicts the turbulent Prandtl number by using some flow properties discussed in chapter 4 along with model constants. This model predicted the correct turbulent Prandtl number for the considered nozzle flow. The model was also validated with independent experimental measurements of turbulent Prandtl number in jet flow, boundary layer and turbulent pipe flow.


