11-2018

A Numerical and Experimental Evaluation of the Turbulent Heat Flux in a Heated Jet in Crossflow

Michael R. Borghi Jr.

Follow this and additional works at: https://commons.erau.edu/edt

Part of the Aerospace Engineering Commons

Scholarly Commons Citation
https://commons.erau.edu/edt/425
A NUMERICAL AND EXPERIMENTAL EVALUATION
OF THE TURBULENT HEAT FLUX IN
A HEATED JET IN CROSSFLOW

A Dissertation
Submitted to the Faculty
of
Embry-Riddle Aeronautical University
by
Michael R. Borghi Jr.

In Partial Fulfillment of the
Requirements for the Degree
of
Doctor of Philosophy in Aerospace Engineering

November 2018
Embry-Riddle Aeronautical University
Daytona Beach, Florida
A Numerical and Experimental Evaluation of the Turbulent Heat Flux in a Heated Jet in Crossflow

By

Michael R. Borghi, Jr.

This Dissertation was prepared under the direction of the candidate’s Dissertation Committee Chair, Dr. William Engblom, Department of Aerospace Engineering, and has been approved by the members of the dissertation committee. It was submitted to College of Engineering and was accepted in partial fulfillment of the requirements for the Degree of

Doctor of Philosophy in Aerospace Engineering

Dr. William Engblom, Ph.D
Committee Chair

Dr. Reda Mankbadi, Ph.D
Committee Member

Dr. Mark Ricklick, Ph.D
Committee Member

Dr. Tasos Lyrintzis, Ph.D
Committee Member

Dr. Tasos Lyrintzis, Ph.D
Department Chair, Aerospace Engineering

Dr. Maj Mirrani, Ph.D
Dean, College of Engineering

Dr. Lon D Moeller, J.D
Senior Vice President for Academic Affairs and Provost

Date

12-6-18
ACKNOWLEDGMENTS

I would like to thank my committee chair Dr. William Engblom for his continued support and encouragement over the nine years I have known and worked with him. Without him, I would not be the researcher I am today. I feel incredible honoured to have had the opportunity to work with him. I would like to thank my committee members, Dr. Tasos Lyrintzis, Dr. Reda Mankbadi, Dr. Mark Ricklick, and Dr. Nicholas Georgiadis, for the excellent and insightful advisement they have provided me throughout my academic career.

Additionally, I would like to extend my sincere gratitude to my colleagues at NASA Glenn Research Center. My supervisor, Dr. Ken Suder has graciously supported my quest for higher education throughout my time as a Pathways intern in the Turbomachinery and Turboelectric Systems branch. In addition to serving on my committee, Dr. Nicholas Georgiadis has been a true mentor and friend throughout my entire graduate education, something I am truly thankful for. I thank Mr. Douglas Thurman and Mr. Philip Poinsatte for their insight and support throughout my time at GRC. At my request, they collected and recollected the experimental data used herein, and selflessly allowed me to process the data however I saw fit. Without them this work would have not been possible. I would also like to thank Dr. Jim DeBonis,
Dr. Dennis Yoder, Dr. Mark Celestina, Dr. Ali Ameri, Mr. Manan Vyas, and Mr. Sameer Kulkarni for always providing excellent insight, suggestions and discussions.

A very special thanks is owed to my parents, Michael and Brenda Borghi. They have provided me with everything I could have ever wanted or needed. They have supported me in the hardest of times, and celebrated with me in the best. Their support means the world to me. I would like to thank my beautiful wife, Nisa, for her continued love and support. Her support throughout my education has made it all worth it. I look forward to our years together without school tagging along for the ride!

Finally, this research was sponsored by NASA’s Transformational Tools and Technologies Project of the Transformative Aeronautics Concepts Program under the Aeronautics Research Mission Directorate. The resources supporting this work were provided by the NASA High-End Computing Program through the NASA Advanced Supercomputing Division at NASA Ames Research Center. Their support is greatly appreciated.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIST OF TABLES</td>
<td>vii</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>viii</td>
</tr>
<tr>
<td>SYMBOLS</td>
<td>xiii</td>
</tr>
<tr>
<td>ABBREVIATIONS</td>
<td>xiv</td>
</tr>
<tr>
<td>NOMENCLATURE</td>
<td>xvi</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td>xix</td>
</tr>
<tr>
<td>1 INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1 The Physics of Film Cooling</td>
<td>1</td>
</tr>
<tr>
<td>1.1.1 The Jet in a Crossflow</td>
<td>6</td>
</tr>
<tr>
<td>1.1.2 Application of Numerical Methods to Film Cooling Problems</td>
<td>9</td>
</tr>
<tr>
<td>1.1.3 Turbulent Heat Flux</td>
<td>15</td>
</tr>
<tr>
<td>1.2 Governing Equations</td>
<td>19</td>
</tr>
<tr>
<td>1.2.1 Navier-Stokes Equations</td>
<td>19</td>
</tr>
<tr>
<td>1.2.2 The Averaged Navier-Stokes Equations</td>
<td>21</td>
</tr>
<tr>
<td>1.2.3 The Spatially Filtered Navier-Stokes Equations</td>
<td>29</td>
</tr>
<tr>
<td>1.3 Scale Resolving Simulations</td>
<td>35</td>
</tr>
<tr>
<td>1.3.1 Synthetic Turbulence Generation</td>
<td>46</td>
</tr>
<tr>
<td>2 EXPERIMENTAL AND NUMERICAL SETUP</td>
<td>53</td>
</tr>
<tr>
<td>2.1 Experimental Setup</td>
<td>53</td>
</tr>
<tr>
<td>2.1.1 CCA frequency compensation</td>
<td>58</td>
</tr>
<tr>
<td>2.2 Numerical Setup</td>
<td>62</td>
</tr>
<tr>
<td>2.2.1 RANS Simulations</td>
<td>63</td>
</tr>
<tr>
<td>2.2.2 Detached Eddy Simulations</td>
<td>66</td>
</tr>
</tbody>
</table>
### Contents

2.2.3 Large Eddy Simulations with Synthetic Turbulence .... 69

3 ANALYSIS OF EXPERIMENTAL RESULTS .......................... 75
   3.1 Effects of Numerical Temperature Compensation .......... 75
   3.2 Sensitivity of Anemometry Measurements to Probe Configuration ... 84

4 RESULTS OF NUMERICAL SIMULATIONS .......................... 94
   4.1 RANS Simulations ........................................ 94
      4.1.1 RANS Grid Sensitivity .................................. 94
      4.1.2 $\kappa-\epsilon$ RANS Comparison to Data ................. 97
   4.2 Detached Eddy Simulations ................................. 104
   4.3 Large Eddy Simulations with Synthetic Turbulence ........... 112
      4.3.1 Validation of Numerical Model ......................... 112
      4.3.2 Effects of Synthetic Turbulence on Inflow ............. 116

5 COMPARISON OF EXPERIMENTAL AND NUMERICAL RESULTS .... 131
   5.1 UV-Probe and Full-SEM ..................................... 131
   5.2 Two-Point Heat Flux Estimations .......................... 144
   5.3 Gradient Diffusion Hypothesis Evaluated ................... 151

6 CONCLUSIONS .................................................. 160

REFERENCES ..................................................... 163

A Rotation of the Reynolds Stress Tensor .......................... 170
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Coefficients used for Sutherland’s law of viscosity</td>
<td>21</td>
</tr>
<tr>
<td>1.2</td>
<td>Constants for the Spalart-Allmaras Turbulence model</td>
<td>43</td>
</tr>
<tr>
<td>2.1</td>
<td>Nominal experimental conditions</td>
<td>58</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>3</td>
</tr>
<tr>
<td>1.2</td>
<td>7</td>
</tr>
<tr>
<td>1.3</td>
<td>10</td>
</tr>
<tr>
<td>1.4</td>
<td>22</td>
</tr>
<tr>
<td>1.5</td>
<td>37</td>
</tr>
<tr>
<td>2.1</td>
<td>54</td>
</tr>
<tr>
<td>2.2</td>
<td>57</td>
</tr>
<tr>
<td>2.3</td>
<td>58</td>
</tr>
<tr>
<td>2.4</td>
<td>59</td>
</tr>
<tr>
<td>2.5</td>
<td>65</td>
</tr>
<tr>
<td>2.6</td>
<td>68</td>
</tr>
</tbody>
</table>

1.1 Representative sketch of control volume surrounding the boundary layer and cooling jet.

1.2 Schematic depiction of the effect of blowing ratio on a jet in crossflow. (a) Lower BR. (b) Higher BR. From Andreopoulos (1984).

1.3 Schematic depiction of the near field region of a jet in crossflow. From Fric (Fric, 1994).

1.4 Continuous time signal of a field variable, U-Velocity here. The dashed line represents the time average of the signal.

1.5 Schematic of the turbulent energy cascade.

2.1 SW-6 Facility in Center Wide Exhaust Configuration. From Wernet et. al. 2016.

2.2 U-W and U-V probes, with U.S. Nickel for scale. The X-wires (velocity) highlighted in blue, while the T-wires (temperature) are highlighted in yellow.

2.3 Sample of experimental temperature measurement. Black squares represent the probing locations for the experimental surveys.

2.4 Schematic representation and coordinates of CCA probe for use in the numerical compensation technique of Tagawa et al. From Tagawa et al. 2005.

2.5 O-H grid topology of the cooling tube exit plane. Edge of the cooling tube exit outlined in red. Every fifth grid line shown.

2.6 Computational domain used with the DES simulations. Insert: Topology of the tunnel-tube grid interface. Every third grid line is shown to better show the grid details.
<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.7</td>
<td>Schematic and isometric view of the computational domain. Isosurface of Q-criterion colored by temperature fluctuations.</td>
</tr>
<tr>
<td>3.1</td>
<td>-3 dB cutoff frequency as a function of velocity and temperature for the 1.2(\mu)m and 3.8(\mu)m CCA wires examined.</td>
</tr>
<tr>
<td>3.2</td>
<td>Estimation of the -3 dB cutoff frequency along the centerplane of the heated jet in crossflow, for the 1.2(\mu)m and 3.8(\mu)m CCA wires examined.</td>
</tr>
<tr>
<td>3.3</td>
<td>Transfer function computed from Eq. 2.8 representing the frequency response and compensation function at 10m/s and 300 K for both the 1.2 (\mu)m and 3.8 (\mu)m wires. -3 dB cutoff frequency labeled for reference.</td>
</tr>
<tr>
<td>3.4</td>
<td>Spectra of temperature signal from 3.8(\mu)m (left) and 1.2(\mu)m (right) CCA measurements with and without compensation.</td>
</tr>
<tr>
<td>3.5</td>
<td>Sample CCA temperature signal from 3.8(\mu)m (left) and 1.2(\mu)m (right) CCA measurements with and without compensation.</td>
</tr>
<tr>
<td>3.6</td>
<td>Centerline measurements of normalized temperature r.m.s fluctuations (\frac{T_{rms}}{\Delta T}) with 3.8(\mu)m (top) and 1.2(\mu)m (bottom) CCA wires with and without effects of compensation.</td>
</tr>
<tr>
<td>3.7</td>
<td>Percent increase in (\frac{T_{rms}}{\Delta T}) with 1.2(\mu)m and 3.8(\mu)m CCA surveys</td>
</tr>
<tr>
<td>3.8</td>
<td>Normalized temperature measurements, Mean and variance, with UV- and UW- xT probes at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00</td>
</tr>
<tr>
<td>3.9</td>
<td>Normalized velocity measurements, Mean and variance, with UV- and UW- xT probes at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00</td>
</tr>
<tr>
<td>3.10</td>
<td>Normalized velocity measurements, Mean and variance, with UV- and UW- xT probes at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00</td>
</tr>
<tr>
<td>3.11</td>
<td>Normalized heat fluxes, with UV- and UW- xT probes at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00</td>
</tr>
<tr>
<td>3.12</td>
<td>Local flow angles (\Theta_u) and (\Theta_w) computed from the Full-SEM LES</td>
</tr>
<tr>
<td>4.1</td>
<td>Streamwise and vertical velocity profiles taken at various X/D locations along the Z/D=0.0 plane. Dashed line: Very fine grid; Solid Line: Fine grid</td>
</tr>
<tr>
<td>Figure</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
</tr>
<tr>
<td>4.2 Normalized mean temperature and TKE profiles taken at various X/D locations along the Z/D=0.0 plane. Dashed line: Very fine grid; Solid Line: Fine grid</td>
<td>96</td>
</tr>
<tr>
<td>4.3 Normalized mean temperature and TKE profiles taken at various X/D locations along the Z/D=0.0 plane. Dashed line: Very fine grid; Solid Line: Fine grid</td>
<td>97</td>
</tr>
<tr>
<td>4.4 Normalized mean temperature for ( \kappa - \epsilon ) RANS and U-V x-T Probe</td>
<td>99</td>
</tr>
<tr>
<td>4.5 Normalized mean streamwise velocity for ( \kappa - \epsilon ) RANS and U-V x-T Probe</td>
<td>100</td>
</tr>
<tr>
<td>4.6 Normalized mean vertical velocity for ( \kappa - \epsilon ) RANS and U-V x-T Probe</td>
<td>101</td>
</tr>
<tr>
<td>4.7 Normalized streamwise turbulent heat flux, ( \overline{u'T} / U_\infty \Delta T ), for ( \kappa - \epsilon ) RANS and U-V x-T Probe</td>
<td>102</td>
</tr>
<tr>
<td>4.8 Normalized vertical turbulent heat flux, ( \overline{v'T} / U_\infty \Delta T ), for ( \kappa - \epsilon ) RANS and U-V x-T Probe</td>
<td>103</td>
</tr>
<tr>
<td>4.9 Normalized mean temperature for SA-DES and U-V x-T Probe. Slices at Z/D = 0.0 X/D = [2,4,6]</td>
<td>107</td>
</tr>
<tr>
<td>4.10 Normalized mean streamwise velocity for SA-DES and U-V x-T Probe. Slices at Z/D = 0.0 X/D = [2,4,6]</td>
<td>108</td>
</tr>
<tr>
<td>4.11 Normalized mean temperature fluctuations for SA-DES and U-V x-T Probe. Slices at Z/D = 0.0 X/D = [2,4,6]</td>
<td>109</td>
</tr>
<tr>
<td>4.12 Normalized mean streamwise velocity fluctuations for SA-DES and U-V x-T Probe. Slices at Z/D = 0.0 X/D = [2,4,6]</td>
<td>110</td>
</tr>
<tr>
<td>4.13 Turbulent viscosity ratio, ( \mu_t / \mu ), for the SA-DES case</td>
<td>111</td>
</tr>
<tr>
<td>4.14 Iso-countour of Q-Criterion (Q=50,000) colored by the normalized axial Reynolds stress.</td>
<td>114</td>
</tr>
<tr>
<td>4.15 Span averaged friction coefficient as a function of momentum thickness Reynolds number, ( Re_\theta )</td>
<td>114</td>
</tr>
<tr>
<td>4.16 Span averaged Reynolds stress profiles vs Reynolds number based on momentum thickness.</td>
<td>115</td>
</tr>
</tbody>
</table>
4.17 Root-mean-squared temperature fluctuations normalized by temperature difference at Z/D = 0.00, X/D = [2,4,6] for a coarse tube-SEM simulation, and a very fine no-SEM simulation. Very fine tube-SEM and U-V probe experimental data provided for reference. ................................. 118

4.18 Normalized Mean temperature profiles from LES and U-V xT-wire probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00 .......................... 123

4.19 Root-mean-squared temperature fluctuations normalized by temperature difference at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00 ............. 124

4.20 Mean streamwise velocity contours from LES and U-V xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00 .......................... 125

4.21 Mean vertical velocity contours from LES and U-V xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00 .......................... 126

4.22 Root-mean-squared streamwise velocity fluctuations contours from LES and U-V xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00 .................................. 127

4.23 Root-mean-squared vertical velocity fluctuations contours from LES and U-V xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00 128

4.24 Root-mean-squared spanwise velocity fluctuations contours from LES and U-W xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00 .................................. 129

4.25 Uncorrelated streamwise heat flux ($U_{rms}T_{rms}/U \Delta T$) contours from LES and U-V xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00 .......................... 130

5.1 Full-SEM LES (solid line) compared with CCA (temperature) and CTA (velocity) data taken with the UV-probe (symbols) ......................... 132

5.2 Normalized Mean (top) and fluctuating (bottom) temperature profiles from LES (left) and U-V xT-wire probe (right) at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00 .......................... 134

5.3 Mean (top) and fluctuating (bottom) streamwise velocity contours from LES (left) and U-V xT-wire Probe (right) at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00 .......................... 136
<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.4 Mean (top) and fluctuating (bottom) vertical velocity contours from LES (left) and U-V xT-wire Probe (right) at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00</td>
<td>137</td>
</tr>
<tr>
<td>5.5 Streamwise turbulent heat fluxes from LES and U-V xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00</td>
<td>141</td>
</tr>
<tr>
<td>5.6 Vertical turbulent heat fluxes from LES and U-V xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00</td>
<td>142</td>
</tr>
<tr>
<td>5.7 Spanwise turbulent heat fluxes from LES and U-V xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00. Note the difference in scale.</td>
<td>143</td>
</tr>
<tr>
<td>5.8 Schematic of the U-V xT-Probe highlighting the velocity-temperature offset.</td>
<td>145</td>
</tr>
<tr>
<td>5.9 Results of two-point heat flux study. Streamwise heat-fluxes at X/D = 3.00 (top left) Full-resolution LES, (top right) UV-Probe. The lower four images show the effects caused by increase probe offset. Black squares represent the experimental probe resolution.</td>
<td>146</td>
</tr>
<tr>
<td>5.10 Streamwise heat flux with effects of spanwise velocity-temperature offset included. Temperature offset of $\delta_{v-t}$ = 0.075D.</td>
<td>148</td>
</tr>
<tr>
<td>5.11 Vertical heat flux with effects of spanwise velocity-temperature offset included. Temperature offset of $\delta_{v-t}$ = 0.075D.</td>
<td>149</td>
</tr>
<tr>
<td>5.12 Streamwise turbulent heat fluxes from LES and $k - \epsilon$ RANS at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00</td>
<td>155</td>
</tr>
<tr>
<td>5.13 Vertical turbulent heat fluxes from LES and $k - \epsilon$ RANS at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00</td>
<td>156</td>
</tr>
<tr>
<td>5.14 Mean temperature contours with streamlines defined by the turbulent heat flux vector at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00</td>
<td>157</td>
</tr>
<tr>
<td>5.15 Mean temperature contours with streamlines defined by the turbulent heat flux vector at Z/D = 0.00, Y/D = 0.5</td>
<td>158</td>
</tr>
<tr>
<td>5.16 Evaluation of the turbulent heat flux vector gradient and its contribution to the energy equation. Heat lines defined by the turbulent heat flux vector included for reference. Slices at Z/D = 0.00, Y/D = 0.5</td>
<td>159</td>
</tr>
</tbody>
</table>
## SYMBOLS

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Delta$</td>
<td>Change or filter width</td>
</tr>
<tr>
<td>$\delta_{ij}$</td>
<td>Kronecker delta</td>
</tr>
<tr>
<td>$FFT$</td>
<td>Fast Fourier Transform</td>
</tr>
<tr>
<td>$G$</td>
<td>Filter kernel</td>
</tr>
<tr>
<td>$IFFT$</td>
<td>Inverse Fast Fourier Transform</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>AUSM</td>
<td>Advection Upstream Splitting Method</td>
</tr>
<tr>
<td>BR</td>
<td>Blowing Ratio</td>
</tr>
<tr>
<td>CCA</td>
<td>Constant Current Anemometry</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CRVP</td>
<td>Counter Rotating Vortex Pair</td>
</tr>
<tr>
<td>CTA</td>
<td>Constant Temperature Anemometry</td>
</tr>
<tr>
<td>DES</td>
<td>Detached Eddy Simulation</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>GDH</td>
<td>Gradient Diffusion Hypothesis</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>MILES</td>
<td>Monotonicity-preserving Implicit Large Eddy Simulation</td>
</tr>
<tr>
<td>PSP</td>
<td>Pressure Sensitive Paint</td>
</tr>
<tr>
<td>PIV</td>
<td>Particle Image Velocimetry</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Averaged Navier Stokes</td>
</tr>
<tr>
<td>SGS</td>
<td>Sub-Grid Scale</td>
</tr>
<tr>
<td>SRS</td>
<td>Scale Resolving Simulation</td>
</tr>
<tr>
<td>TKE</td>
<td>Turbulent Kinetic Energy</td>
</tr>
<tr>
<td>RMS</td>
<td>Root Mean Squared</td>
</tr>
</tbody>
</table>
SA  Spalart-Allmaras turbulence model

SST  Shear Stress Transport turbulence model
**NOMENCLATURE**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Area</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Thermal diffusivity</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Specific heat at constant pressure</td>
</tr>
<tr>
<td>$C_v$</td>
<td>Specific heat at constant volume</td>
</tr>
<tr>
<td>$c_i$</td>
<td>Specific heat of $i^{th}$ CCA component</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Boundary layer thickness</td>
</tr>
<tr>
<td>$D$</td>
<td>Cooling hole diameter</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
</tr>
<tr>
<td>$\rho_j$</td>
<td>Jet density</td>
</tr>
<tr>
<td>$\rho_\infty$</td>
<td>Freestream density</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>Massflow rate</td>
</tr>
<tr>
<td>$\dot{m}_\infty$</td>
<td>Freestream massflow rate</td>
</tr>
<tr>
<td>$\dot{m}_\infty$</td>
<td>Jet massflow rate</td>
</tr>
<tr>
<td>$e$</td>
<td>Specific internal energy</td>
</tr>
<tr>
<td>$\eta$</td>
<td>Film effectiveness</td>
</tr>
<tr>
<td>$\eta_k$</td>
<td>Kolmogorov scale</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>Turbulent dissipation rate</td>
</tr>
<tr>
<td>$F_c$</td>
<td>Cutoff frequency of probe</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>$f_{ci}$</td>
<td>Cutoff frequency of $i^{th}$ CCA component</td>
</tr>
<tr>
<td>$h$</td>
<td>Heat transfer coefficient or specific enthalpy</td>
</tr>
<tr>
<td>$H$</td>
<td>Compensation function for CCA signal</td>
</tr>
<tr>
<td>$I_t$</td>
<td>Turbulent intensity</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>Turbulent kinetic energy or wavenumber</td>
</tr>
<tr>
<td>$Kn$</td>
<td>Knudsen number</td>
</tr>
<tr>
<td>$k$</td>
<td>Thermal conductivity</td>
</tr>
<tr>
<td>$Ma$</td>
<td>Mach number</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Viscosity</td>
</tr>
<tr>
<td>$\mu_t$</td>
<td>Turbulent viscosity</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity</td>
</tr>
<tr>
<td>$Nu$</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>$\overline{T}$</td>
<td>Mean temperature</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$Pr$</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>$Pr_t$</td>
<td>Turbulent Prandtl number</td>
</tr>
<tr>
<td>$\dot{q}$</td>
<td>Specific rate of heat transfer</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$\sigma_{ij}$</td>
<td>Viscous stress tensor</td>
</tr>
<tr>
<td>$s_{ij}$</td>
<td>Rate of strain tensor</td>
</tr>
<tr>
<td>$\tau_i$</td>
<td>Time constant of $i^{th}$ CCA component</td>
</tr>
<tr>
<td>$\tau_w$</td>
<td>Wall shear stress</td>
</tr>
<tr>
<td>Symbol</td>
<td>Definition</td>
</tr>
<tr>
<td>--------</td>
<td>------------------------------------------------</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature or Period</td>
</tr>
<tr>
<td>$T_{aw}$</td>
<td>Adiabatic wall temperature</td>
</tr>
<tr>
<td>$T_\infty$</td>
<td>Freestream temperature</td>
</tr>
<tr>
<td>$T_j$</td>
<td>Jet temperature</td>
</tr>
<tr>
<td>$T_w$</td>
<td>Wall temperature</td>
</tr>
<tr>
<td>$T_{inlet}$</td>
<td>Turbine inlet temperature</td>
</tr>
<tr>
<td>$\theta$</td>
<td>Nondimensional temperature</td>
</tr>
<tr>
<td>$\Theta_u$</td>
<td>Flow direction angle in U-V plane</td>
</tr>
<tr>
<td>$\Theta_w$</td>
<td>Flow direction angle in U-W plane</td>
</tr>
<tr>
<td>$U_j$</td>
<td>Jet velocity</td>
</tr>
<tr>
<td>$U_\infty$</td>
<td>Freestream velocity</td>
</tr>
<tr>
<td>$u, v, w$</td>
<td>Cartesian velocity components</td>
</tr>
<tr>
<td>$u'T', v'T', w'T'$</td>
<td>Cartesian turbulent heat flux components</td>
</tr>
<tr>
<td>$v$</td>
<td>velocity</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Specific turbulent dissipation rate or Spectral frequency</td>
</tr>
<tr>
<td>$\Omega_i$</td>
<td>CCA wire parameter</td>
</tr>
<tr>
<td>$x^+, y^+, z^+$</td>
<td>Cartesian viscous wall units</td>
</tr>
<tr>
<td>$Y$</td>
<td>Spectrum of uncompensated temperature signal</td>
</tr>
<tr>
<td>$Y_c$</td>
<td>Spectrum of compensated temperature signal</td>
</tr>
</tbody>
</table>
ABSTRACT

The injection of fully-developed turbulent heated air from a tube into a cooler turbulent duct flow is examined, as an analogy to film cooled turbine blades. Scale Resolving Simulations (SRS) are used to examine the flow numerically. A Detached Eddy Simulation (DES) methodology is examined but found to be ineffective at correctly capturing the physics of the flow. A Large Eddy Simulation (LES) numerical model is developed and applied in which tube and duct turbulence inflow effects are emulated using a divergence-free synthetic eddy method (SEM). The LES sensitivity to the synthetic inflow turbulence is examined with a series of simulations with the SEM inflow toggled on and off. The effects of turbulence in the coolant tube are found to the most critical for accurate prediction. For direct comparison, a hot-wire experiment is conducted within the ERB test cell SW-6 at NASA Glenn Research Center. Excellent agreement is obtained for these numerical and experimental results related to velocity, temperature, and heat flux, for a blowing ratio of 1.2, and involving a 36 K temperature difference. The relative effect on the solutions of tube and duct inflow turbulence is systematically evaluated. The impact of inherent low-pass filtering of temperature measurements and probe wire offset on the experimental results are addressed. The validity of the gradient diffusion hypothesis, fundamental to Reynolds-Averaged Navier-Stokes (RANS) models, is evaluated.
1. INTRODUCTION

1.1 The Physics of Film Cooling

Film cooling is an active cooling technique universally used in gas turbine engines to protect the metal components from the harsh conditions found in the hot section (turbine and nozzle) of the engine. Typically, cooler air is bled from the compressor or core-bypass flow and blown through and then out of channels built within the turbine blades. Because a turbine’s overall thermal efficiency is tied to the turbine inlet temperature ($T_{inlet}$), there is a strong desire to increase the allowable temperatures within the engine. As a result, film cooling has remained an active area of research over the previous five decades (Goldstein, Eckert, & Ramsey, 1968; Bogard & Thole, 2006; Mahesh, 2013). This sustained research has led to the $T_{inlet}$ temperatures, which can range from 1200$^\circ$C to 1600$^\circ$C, to surpass the allowable temperatures of the metal alloys the blades are commonly made from (Bogard & Thole, 2006; Han & Ekkad, 2001). The push for improved performance and efficiency will continue to drive hot-section temperatures higher. Even with advanced materials such as ceramic matrix composites (Ruggles-Wrenn & Jones, 2012) and thermal barriers/coatings, there remains a need for active cooling.
The intent of introducing a secondary film of fluid along a surface - turbine blade, rocket nozzle or otherwise - is to reduce the rate of convective heat transfer into the object. This is accomplished through the reduction of the temperature of fluid nearest the surface of interest. Newton’s law of cooling states that the rate of heat transfer is proportional to the temperature difference between the surface and surrounding fluid. Specifically,

\[ \dot{q} = \frac{\dot{Q}}{A} = h(T_w - T_\infty) \]  

(1.1)

Where \( h \) is the heat transfer coefficient in \( \frac{W}{m^2K} \) or equivalent units. \( T_w \) is the temperature of the wall, and \( T_\infty \) is the temperature of the bulk fluid passing over the wall in \( K \). It should be noted that the heat transfer coefficient is a function of the local aerodynamic conditions, including the introduction of a cooling fluid.

As the primary intent of the film cooling is to protect the structural integrity of an object such as a turbine blade, a film cooling effectiveness parameter, \( \eta \), is often defined. This effectiveness parameter is intended to quantify how well the film cooling system is displacing the hot bulk flow away from the surface.

\[ \eta = \frac{T_{aw} - T_\infty}{T_j - T_\infty} \]  

(1.2)

Here \( T_{aw} \) is the local temperature, \( T_j \) is the temperature of the jet, and \( T_\infty \) is the freestream bulk temperature. When applied along the wall, a non-dimensional
temperature distribution, is obtained. The film effectiveness can be applied away from the wall, with the local temperature replacing the adiabatic temperature, to evaluate mixing effectiveness. A simple 2-D control volume analysis can be performed in an attempt to estimate \( \eta \). A mass flow and enthalpy balance may be done on a control volume (dashed lines) surrounding the boundary layer and film cooling hole, shown in Fig. 1.1.

\[
\dot{m} = \dot{m}_j + \dot{m}_\infty = \int_0^\delta \rho(y)U(y)dy
\]

\hspace{1cm} (1.3)
Using an enthalpy balance based on figure 1.1 (b) the average temperature $\bar{T}$ within the boundary layer is be found as,

$$\bar{T} = T_\infty + \frac{\int_0^\delta \rho U(y)C_p(T - T_\infty)dy}{\int_0^\delta \rho U(y)C_p dy}$$  \hspace{1cm} (1.4)$$

The massflow averaged specific heat in the boundary layer $\bar{c}_p$ is defined as,

$$\bar{c}_p = \frac{\dot{m}_j c_{pj} + \dot{m}_\infty c_{p\infty}}{\dot{m}_j + \dot{m}_\infty}$$  \hspace{1cm} (1.5)$$

An enthalpy balance can be rewritten as,

$$(\dot{m}_j + \dot{m}_\infty)\bar{c}_p \bar{T} = \dot{m}_j c_{pj} T_j + \dot{m}_\infty c_{p\infty} T_\infty$$  \hspace{1cm} (1.6)$$

and further rearranged using (1.5) and (1.6) to give,

$$\frac{\bar{T} - T_\infty}{T_j - T_\infty} = \eta = \frac{1}{1 + \frac{\dot{m}_\infty c_{p\infty}}{\dot{m}_j c_{pj}}}$$  \hspace{1cm} (1.7)$$

Equation 1.7 shows that $\eta$ can be directly related to the ratio of the thermal conductances ($\dot{m}c_p$) within the jet to that which is entrained into the boundary layer from the bulk flow. However, the contribution entrained from the bulk flow is not trivial to evaluate. Goldstein (1971) as well as Sou (1985) provide a review of the analytical approaches taken within the literature to approximate $\dot{m}_\infty$. One major flaw is that the assumed velocity profiles are typically taken for a growing
boundary layer over a flat plate without injection. These profiles are used without
adjustment for the increased momentum thickness caused by the injection of cooling
flow. The determination of exactly where the boundary layer starts in relation to
the injection site is also a major problem. However, the largest problem for this
analysis could be the 2-D nature (slot injection) of the control volume analysis. Slot
injectors are uncommon in film cooling applications as they are unreliable in their
operation, with uneven blowing common. Additionally, structural design constraints
imposed on turbine blades are difficult to meet with slot injectors. As a result of
these limitations, discrete cooling holes are used. The highly 3-D nature of discrete
injection holes makes a closed form analysis impractical. Due to these limitations
early research was forced to rely on experimental testing.

Goldstein (1971) also provides a summary of early experimental work done on film
cooling. Many different configurations were tested including porous walls,
backwards facing steps, and 3-D inclined jets, at a range of different blowing ratios
and Reynolds numbers. One notable work reviewed is that of Goldstein et al.
(1968) in which 2-D and 3-D cooling holes at different blowing ratios are tested.
Later work by Goldstein et al. (1974) measured the effect of different densities and
hole shapes on the film cooling effectiveness.

More recently, advanced experimental techniques have been applied to the jet in
crossflow problem, and more specifically, to turbine film cooling. With the advent of
techniques including particle image velocimetry (PIV) and pressure sensitive paint
(PSP) a more fundamental understanding of film cooling is possible. With higher
resolution experimental data being obtained, experimentalists are able to focus on the individual effects of various flow parameters. Forth et al. (1985) for example, identified the marked effect of density ratio \((DR)\) on film cooling effectiveness. Wright et al. (2011a) used PSP to look at the effect of density and shaped holes on cooling effectiveness. Chen et al. (2015) evaluated the effect of injecting the coolant flow at an angle askew to the core flow direction, finding much improved distribution of cooling flow. Bons et al. (1996) and Wright et al. (2011b) have looked at the effect freestream turbulence has on film cooling effectiveness, finding significant increase in effectiveness at higher turbulence intensities, \(I_t\). Some experimental investigations have moved away from the flat plate surrogate and use more realistic conditions and geometries (Dunn & Mathison, 2014; Waye & Bogard, 2007).

1.1.1 The Jet in a Crossflow

As mentioned previously, film cooling applications are a small subset of a larger set of flow phenomena: jets in crossflows. Jets in a crossflow are evident in numerous physical problems, from film cooling of turbine blades to the venting of exhaust gas from a smoke tower. A key parameter in classifying the jet in crossflow problem is the blowing ratio,

\[
BR = \frac{\rho_j U_j}{\rho_\infty U_\infty}
\]  

(1.8)

Where \(\rho_j\) and \(U_j\) are the density and velocity of the jet respectively, and \(\rho_\infty\) and \(U_\infty\) are the properties of the bulk crossflow. The forces driving the flow are
Figure 1.2: Schematic depiction of the effect of blowing ratio on a jet in crossflow. (a) Lower $BR$. (b) Higher $BR$. From Andreopoulos (1984).

of different natures for blowing ratios which are low ($BR \in [0.5, 2]$) as compared to higher blowing ratios ($BR > 2$) (Andreopoulos & Rodi, 1984). For higher blowing ratios the region near the jet is driven by inviscid fluid dynamics, with turbulence effects not playing a role until further downstream. Lower blowing ratios however have near fields which are dominated by turbulence effects, and have the added
complication of largely non-uniform velocity profiles at the jet exit. Figure 1.2 shows schematic depictions of the impact BR has on jet dynamics.

As shown in Fig. 1.2 as \( BR \) increases the more the jet is lifted off the surface. This lifting causes flow from either side of the jet to move towards the plane of symmetry. This flow then recirculates and moves upstream towards the jet and ultimately entrained into the jet. This increased recirculation region in higher blowing ratio flows leads to low film cooling efficiency, and is ultimately to the conclusion higher blowing ratios \( (BR > 2) \) are not suitable for film cooling applications. However, sufficient blowing must be applied to not allow for hot flow ingestion within the cooling passages.

Fric and Roshko (1994) identify four main regions with the near field region of the jet in crossflow. These regions are:

- **Jet shear-layer vortices** - these vortices have their origin within the boundary layer within the jet, and grow due to a Kelvin-Helmholtz instability within the jet-freestream shear-layer (Fric & Roshko, 1994).

- **System of horse shoe vortices** - these vortices are the result of an adverse pressure gradient just upstream of the jet, along with vorticity which exists within the boundary layer upstream of the jet (Coussement, Gicquel, & Degrez, 2012).
• Counter-rotating vortex pair (CRVP) - dominate the flow downstream of the injection site. According to Cortelezzi & Karagozian (2001) these are a result of a process of rolling up, tilting, and folding over shear layer vortices.

• Wake vortices - the least understood of the near field phenomena, are likely generated from the boundary layer downstream of the jet. These vortices will play a large role in the effectiveness of film cooling applications of a jet in crossflow (Fric & Roshko, 1994).

Of these four regions two are inherently unsteady phenomenon: the jet shear-layer and wake vortices. These will require an eddy-resolving time accurate method in order to be captured by numerical simulations. The other regions of the flow, while containing unsteady components, have strong base flow representation and are captured by steady flow methods. Figure 1.3 shows a schematic depiction of the four dominate regions of the near field region of a jet in crossflow.

1.1.2 Application of Numerical Methods to Film Cooling Problems

The jet in crossflow problem as applied to film cooling applications is a challenging problem numerically. This is largely due to the highly complex, turbulence dominated near region in the wake of the jet. Only the dominate CRVP and horse shoe vortices are easily captured by steady RANS simulations. This can be seen in the work by Leylek and Zerkle (1994) who used a $k - \epsilon$ turbulence model on a coarse grid, comprised of roughly 200,000 nodes.
Walters and Leylek (2000) published a comprehensive four part series of papers detailing film cooling physics. In Part I use of wall-functions are compared with two-layer $k - \varepsilon$ based turbulence model for streamwise injection with cylindrical holes. In Part II McGovern and Leylek (2000) followed a similar procedure to investigate compound-angle injection with cylindrical holes. Part III has Hyames and Leylek (2000) expanding the study to streamwise injection through shaped holes. Finally, Part IV has Brittingham and Leylek (2000) examining compound angle injection with shaped holes. The authors reported the use of the more detailed two-layer model as necessary but not sufficient for capturing the complex physics of film cooling.

Currently, limitations still exist with conventional turbulence closures to the RANS equations, as applied to film cooling configurations. Hoda and Acharya (2000)
completed a numerical study evaluating several different turbulence models’ ability to cope with the highly complex near region of the jet wake. Several different turbulent closures were evaluated including high and low-\(Re\) formulations of the \(k - \omega\) and \(k - \epsilon\) two equation models, as well as non-linear eddy viscosity models of Mayong and Kasagi, and Speziale. In general the models over predict the peak jet velocities in the wake region of the jet, over predict vertical penetration, and under predict lateral penetration. The authors further concluded that the tested models gave ”...overly simplistic predictions for the highly complex flow field...” and that ”better resolution in the near-wall region is clearly needed”. Hoda et al. (2000) then extended the work to include Reynolds Stress Transport (RST) models. Their work found a significant under prediction in lateral shear stress, \(u'w'\) across the range of turbulence models tested. To evaluate and isolate the shortcomings of the standard \(k - \epsilon\) turbulence model Muldoon and Acharya (2000) compared the different terms in the modeled TKE equation with their exact counterparts found through a DNS simulation. Muldoon and Acharya found that the Boussinesq gradient approximation was reasonably accurate, but the expression for the eddy viscosity within the standard \(k - \epsilon\) model was found to be highly inaccurate. The authors provided two alternative damping functions which greatly improve the predicted distribution of eddy viscosity from the \(k - \epsilon\) model.

Due to the inability of RANS simulations to adequately capture the near wake region, substantial research has recently been done in large-eddy simulations (LES) of jets in crossflows. Tyagi and Acharya (2003) performed LES of an inclined
cylindrical jet on a uniform grid of just over one million cells. The computational domain extends \([-5D,12D]\) X \([-1D,4D]\) X \([-3D,3D]\) in the streamwise, vertical, and spanwise directions; respectively, with cell spacing of \([0.1D X 0.05D X 0.1D]\).

Blowing ratios of \(BR = 0.5\) and \(BR = 1.0\) were considered. Temporal advancement was done with an explicit second order Adams-Bashforth scheme, while a mixed 3\(^{rd}/4^{th}\) order finite difference scheme was used for spatial discretization. Clear improvement over RANS based simulation were seen in the averaged velocity profiles at downstream locations. However, limited comparisons with experimental Reynolds stresses raises questions on how well the near region of the jet was resolved.

Renze et al. (2008) used LES to examine the film cooling problem when density gradients are present. Two density ratios are achieved through the use of air-to-air injection and \(CO_2\)-to-air injection. Both density ratios were examined through a range of blowing ratios ranging from \(BR = 0.1\) to \(BR = 0.5\). The computational domain extends \([-7.6D,28D]\) X \([0D,4D]\) X \([-1.5D,1.5D]\) in the streamwise, vertical, and spanwise directions respectively, with cell spacing ranging from \([1.5^+,75^+]\) X \([1^+,50^+]\) X \([1.5^+,12^+]\) in wall viscous \((y^+)\) units. An additional simulation is done concurrently to simulate the incoming boundary layer using rescaling techniques, where the turbulent boundary layer profile and quantities are rescaled to match the desired inflow profile. A multi-specie, mixed central/upwind AUSM scheme was used within an Monotonicity-preserving Implicit Large Eddy Simulation (MILES) implementation, with no subgrid scale (SGS) treatment. Very good agreement is
predicted for the averaged velocity profiles. Agreement is not as good for the rms velocity profiles.

Johnson and Kapat (2013) performed LES calculations of a cylindrical film cooling hole to examine the unsteady jet in crossflow interactions, and the effect blowing ratio. A rescaling technique was used to simulate the incoming boundary layer. The computational domain extends \([-5D,11D]\) X \([0D,6D]\) X \([-1.5D,1.5D]\) in the streamwise, vertical, and spanwise directions respectively. Wall normal spacing was less than unity for the first cell off the wall, with a growth rate of 1.2. Axial and lateral spacing ranged from \([15^{+},40^{+}]\) wall units. A dynamic Smagorinsky subgrid model was used, along with a constant subgrid Prandtl number. The authors report favourable results with the exception of the regions just downstream of the injection site. No benefit was observed from using the dynamic Smagorinsky for turbulent heat fluxes compared to using a constant subgrid Prandtl number.

Other research into turbine film cooling with LES has moved towards larger more complex problems. Ziefle and Kleiser (2013) have looked at the effect of crossflow turbulence intensity on film cooling flow structure. Sarkar and Babu (2015) studied the effect of a transient wake on film cooling injections along the leading edge. Martini et al. (2006) studied more complex film cooling geometries involving trailing edge cutbacks on turbine airfoils.

The fourth region (identified by Fric and Roshko) just downstream of the injection site is especially difficult to adequately resolve. The wake vortices are often left
unresolved. This is related to the low blowing ratios of interest for film cooling applications. If the jet “lies down” along the surface of interest, the scale of the wake vortices drastically reduces. The wake vortices become a part of the turbulent boundary layer beneath the jet. Sagaut et al. state LES of a turbulent boundary layer should be considered quasi-DNS as the computational cost required is only an order of magnitude less than that of DNS. It has also been suggested that the spacing normal to the wall for wall bounded LES simulations should be on the order of $\Delta y^+ = 1$ carried across the entire boundary layer (Sagaut, Deck, & Terracol, 2013). These grid requirements impose excessive computational costs.

In LES cases, the generation of realistic turbulent inlet flow conditions is often crucial. In many cases, without a forced disturbance, a steady inflow condition will remain in a stable laminar condition and may never fully transition to turbulent flow. A long standing practice has been to use a precursor simulation to generate the inflow boundary data via a procedure such as recycling and rescaling (Johnson & Kapat, 2013; Lund, Wu, & Squires, 1998). This approach however requires a relatively simple geometry and significant extra computational expense and time. These limitations have led to the development of synthetic turbulence generation methods. In earlier works, random noise was applied to a mean flow as a method of perturbing the freestream flow. The lack of coherent structures in time and space led to a rapid decay of the turbulent energy (Jarrin, Addad, & Laurence, 2003). Klein et al. (2003) developed a methodology to digitally filter random data onto the inlet mesh. Jarrin et al. (2006; 2009) developed the synthetic eddy method
(SEM), which takes a mean velocity profile and Reynolds stress tensor to create coherent turbulent structures at the inflow during the runtime of the simulation. Recently, developments have been made to the SEM approach to improve the accuracy and efficiency, as well as the ability to produce a divergence-free turbulent flow field (Poletto, Craft, & Revell, 2013; Patruno & Ricci, 2017; Skillen, Revell, & Craft, 2016). For the present study, the divergence-free synthetic eddy method is selected for use in the LES simulations.

1.1.3 Turbulent Heat Flux

In addition to the previously discussed momentum closures, the accurate prediction of turbulent heat transfer is critical for the design of film cooling and other aerothermal systems. In RANS solvers the turbulent heat flux, $\overline{\rho u'_i h'}$, found in the Favre-averaged energy equation (1.9), is responsible for the transport of heat due to turbulent motion. Like the Reynolds stress, the turbulent heat flux requires a turbulent closure.

$$\begin{gathered}
\frac{\partial}{\partial t} \left( \bar{p} \left( \bar{e} + \frac{1}{2} \bar{u}_i \bar{u}_i + k \right) \right) + \frac{\partial}{\partial x_j} \left( \bar{p} \bar{u}_j \left( \bar{h} + \frac{1}{2} \bar{u}_j \bar{u}_j + k \right) \right) = \\
\frac{\partial}{\partial x_j} \left( -q_j - \rho \bar{u}'_j h' + \sigma_{ji} \bar{u}_i - \frac{1}{2} \rho \bar{u}'_j \bar{u}'_j \right) + \\
\frac{\partial}{\partial x_j} \left( \bar{u}_i \left( \sigma_{ij} - \rho \bar{u}'_i \bar{u}'_j \right) \right)
\end{gathered}$$ (1.9)

In conventional RANS solvers the closure is often accomplished through the use of the gradient diffusion hypothesis (GDH). The GDH defines a dynamic similarity
between heat and momentum transfer through the use of a constant turbulent
Prandtl number $Pr_t = \frac{\nu_t}{a_t}$ and mean temperature gradient,

$$\frac{\rho u'_j h'}{Pr_t \partial h/\partial x_j} = -\frac{\mu_t}{Pr_t c_p \partial T/\partial x_j}$$

As will be presented in this work, simple GDH based approaches fall short of
providing adequate turbulent heat flux predictions and therefore thermal
distributions. Variable $Pr_t$ models allow for the scaling of heat fluxes, however they
are unable to affect the flux directionality. As a result, variable $Pr_t$ models have
had difficulty matching experimental data (Wassel & Catton, 1973; Bradshaw,
Launder, & Lumley, 1991; Sommer, So, & Zhang, 1993; Ivanova, Noll, Domenico, &
Aigner, 2008). The GDH can be extended to a scalar-flux model through the
inclusion of the Reynolds stress tensor to account for heat flux anisotropy (Daly &
Harlow, 1970; Abe & Suga, 2001; Ling, Ryan, Bodart, & Eaton, 2016). These
higher order extensions of the GDH require accurate Reynolds stress predictions,
which pose additional challenges.

The validation of RANS turbulent heat flux models for design purposes (Deng, Wu,
& Xi, 2001; Karcz & Badur, 2003) are limited by a lack of high quality experimental
and numerical data for complex flow fields, with the relevant turbulent quantities.
In fact, some studies have reported aphysical trends in the turbulent quantities
measured (Crabb, Durao, & Whitelaw, 1981). Temperature fluctuations ($T'$) and
turbulent heat fluxes ($\overline{u'T}$) are key parameters which are not commonly or easily
measured. High frequency temperature measurements using fine-wire thermometers are difficult to take due to the low cut off frequencies operating in constant current mode (CCA) (Smits, Perry, & Hoffmann, 1978; Childs, Greenwood, & Long, 2000).

In his dissertation, Khalkhal (1997) proposed a numerical compensation technique to account for the low pass filtering of temperature signals measured through the use of CCA. Through the selection of an observed and desired cutoff frequency the spectra of the temperature signal can be boosted. This method required empirical knowledge of cutoff frequencies for various wires, and did not account for other second order effects. Tagawa et al. (2005) extended the compensation technique by analytically solving the heat equation across a CCA wire, and obtaining a transfer function for the frequency response. This enables a priori estimation of cutoff frequencies and corrections thereof based on flow conditions and physical parameters of the wire only. More recently, Arwatz et al. (2013) further extended the method seen in Tagawa’s work to include additional empirically derived parameters quantifying the performance of the CCA probes. The additional data is obtained through additional bench top experiments.

For turbulent heat flux measurements, simultaneous measurements of velocity and temperature must be taken. Constant temperature anemometry (CTA) or hotwires are often used to obtain velocity measurements. Multi-sensor probes consisting of fine wires for temperature and velocity measurements are often used. Kohli et al. (2005) used the combination of CCA temperature measurements and Laser Doppler Velocimetry (LDV), to investigate the heat fluxes and turbulent heat transport, in a
film cooling configuration. The physical separation of these measurements (i.e.,
distance between velocity and temperature sensors) can introduce significant error
to the correlation.

Recently, researchers have published film cooling work done with MRI-based
Magnetic Resonance Velocimetry (MRV). Coletti et al. (2013) apply MRV to the
film cooling problem and measure mean velocity and the distribution the injected
contaminate. This is used to validate accompanying LES results which examine the
turbulent heat flux. This work is based on the assumption that the turbulent
Prandtl ($Pr_t$) and turbulent Schmidt ($Sc_t$) numbers are equal. Additionally the
MRV technique does not allow for the direct measurement of scalar fluxes for
additional validation of the numerical simulations.

Accurate and efficient numerical techniques, validated by detailed experiments, are
sought to support advancement of turbine film cooling technologies. For the present
study, multiple hot-wire probes and post-processing techniques are utilized in an
effort to obtain a high-quality turbulence-related data set. This data set will be
used in conjunction with the present well resolved LES simulations to provide
insight into the the turbulent transport of heat within film cooling flows.

Additionally, the effects imparted by the physical geometry of the experimental
probes is examined $a$ $posteriori$ with the LES results.
1.2 Governing Equations

1.2.1 Navier-Stokes Equations

The Navier-Stokes equations are the set of partial differential equations which govern fluid motion acting as a continuum. Here they will be presented with the assumption of Newtonian flow. A Newtonian fluid is one which sees a linear relationship between the viscous stress and rate of strain tensors. They are comprised of the continuity equation, three momentum equations, and an energy equation.

Continuity equation:

\[ \frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \]  \hspace{1cm} (1.11)

Momentum equations:

\[ \frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_j u_i}{\partial x_i} = -\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ji}}{\partial x_j} \]  \hspace{1cm} (1.12)

Energy equation:

\[ \frac{\partial}{\partial t} \left[ \rho \left( e + \frac{1}{2} u_i u_i \right) \right] + \frac{\partial}{\partial x_j} \left[ \rho u_j \left( h + \frac{1}{2} u_i u_i \right) \right] = \frac{\partial}{\partial x_j} \left( u_i \sigma_{ij} \right) - \frac{\partial q_j}{\partial x_j} \]  \hspace{1cm} (1.13)
Equations 1.11-1.13 are the Navier-Stokes equations for a compressible fluid. The total energy of the fluid, $E$, can be described:

$$E = \rho \left( e + \frac{1}{2} \rho u_i u_i \right)$$  \hspace{1cm} (1.14)

Assuming a Newtonian fluid, $\sigma_{ij}$ is the viscous stress tensor, $S_{ij}$ is the rate of strain tensor, as described below:

$$\sigma_{ij} = 2\mu s_{ij} - \frac{2}{3}\mu \delta_{ij} \frac{\partial u_k}{\partial x_k}$$  \hspace{1cm} (1.15)

$$s_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$  \hspace{1cm} (1.16)

The laminar dynamic viscosity, $\mu$, can be represented through Sutherland’s law of viscosity for ideal gases,

$$\mu(T) = \mu_0 \left( \frac{T_0 + C}{T} \right)^{3/2} \left( \frac{T}{T_0} \right)^{3/2}$$  \hspace{1cm} (1.17)

where $\mu_0$ and $T_0$ and $C$ are the reference dynamic viscosity, temperature and Sutherland’s constant respectively for the specified gas. For air, these values are shown in Table 1.1. In the heat flux term, $q_j$, $k$ is the thermal conductivity of the gas. In conventional computational methods Fourier’s law is used to represent the heat flux,

$$q_j = -k \frac{\partial T}{\partial x_j} = -\mu \frac{\partial h}{Pr \partial x_j}$$  \hspace{1cm} (1.18)
The Prandtl number, \( Pr \), is a dimensionless number that represents the ratio between momentum diffusivity and thermal diffusivity. Explicitly it is defined as,

\[
Pr = \frac{\nu}{\alpha} = \frac{\mu c_p}{k}
\]  

(1.19)

Table 1.1: Coefficients used for Sutherland’s law of viscosity

<table>
<thead>
<tr>
<th>Coefficients</th>
<th>Value for Air</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \mu_0 ) ([\text{kg} \cdot \text{ms}^{-1}])</td>
<td>(1.716 \times 10^{-5})</td>
</tr>
<tr>
<td>( T_0 ) ([\text{K}])</td>
<td>273.15</td>
</tr>
<tr>
<td>( C ) ([\text{K}])</td>
<td>110.4</td>
</tr>
</tbody>
</table>

The ideal gas equation is included for the equation of state:

\[
p = \rho RT
\]

(1.20)

1.2.2 The Averaged Navier-Stokes Equations

The application of the Navier-Stokes equations (1.11-1.13) to a wide range of practical engineering problems does not always necessitate the use of the full unsteady set of equations. Often, a steady-state mean solution can provide the desired quantities at a significantly reduced computational cost. Techniques have been developed to simplify the full set of Navier-Stokes equation into a steady form. The most common techniques are Reynolds and Favre averaging. In the following
sections both the Reynolds averaged (time) and Favre (mass weighted time) averaged equations will be presented.

Reynolds Averaging

The Reynolds decomposition is a key part of the averaging process commonly applied to the Navier-Stokes equations. The time average of a continuous time signal of a field variable, \( f \), at point \( p \) in space, as shown in Fig. 1.4. This averaging process is defined,

\[
\overline{f(p)} = \frac{1}{t_2 - t_1} \int_{t_1}^{t_2} f(p, t) dt \tag{1.21}
\]

The aim of the Reynolds decomposition is to separate the steady base flow and fluctuating components from the instantaneous signal. It must be ensured that the

Figure 1.4: Continuous time signal of a field variable, U-Velocity here. The dashed line represents the time average of the signal.
period which the Reynolds averaged is taken over (i.e. \( t_2 - t_1 \)) is larger than the characteristics time scales of the turbulence which are present in the flow. The instantaneous signal can now be defined as,

\[
f(p, t) = \overline{f(p)} + f'(p, t)
\]  

(1.22)

The Reynolds decomposition is to be applied to the Navier-Stokes equations to obtain a set of relations for the mean flow quantities. Before this can be done a few important relations should be defined for the decomposed field variables.

\[
\overline{f'} = 0 \\
\overline{f_1 f_2'} = 0 \\
\overline{f_1' f_2'} \neq 0
\]  

(1.23, 1.24, 1.25)

By definition the fluctuating component of a single variable time signal has a zero time average, as the steady base component was removed with the decomposition. Similarly, the product of a mean component and a fluctuating one also goes to zero when averaged. The product of two fluctuations does not necessarily go to zero when the time average is applied. With these rules in mind the Reynolds decomposition
(1.22) can now be applied to the variables within the Navier-Stokes equations to form the incompressible Reynolds Averaged Navier-Stokes (RANS) equations.

\[
\frac{\partial u_j}{\partial x_j} = 0 \tag{1.26}
\]

\[
\rho \frac{\partial u_j u_i}{\partial x_i} = - \frac{\partial P}{\partial x_i} + \rho \frac{\partial}{\partial x_j} \left( 2\mu S_{ji} - \bar{u}'_j u'_i \right) \tag{1.27}
\]

Here a time average has been used to define the mean and fluctuating components of the signal. For stationary processes other averaging techniques, such as ensemble averaging, are appropriate with no loss of generality seen in the RANS equations above. For incompressible flows the coupling between energy and momentum equations is often very weak, therefore the energy equation is omitted here. The Favre-averaged energy equation will be discussed in the next section.

**Favre-Averaged Equations**

Often when dealing with compressibility and heat transfer effects it is convenient to use a mass weighted time average for the Navier-stokes equations. The Reynolds decomposition of the density field provides,

\[
\rho = \bar{\rho} + \rho' \tag{1.28}
\]
The time averaging process described above can now be reformed as,

$$\tilde{f}(p) = \frac{1}{\tilde{p}(t_2 - t_1)} \int_{t_1}^{t_2} \rho(p, t) f(p, t) dt$$ (1.29)

Applying to the conservative momentum variable, $\rho u_j$,

$$\bar{\rho} \tilde{u}_j = \rho \bar{u}_j$$ (1.30)

With this mass averaged velocity a new Favre decomposition can be formulated,

$$f_i = \tilde{f}_i + f''_i$$ (1.31)

Applying this technique to the Navier-Stokes (1.11-1.13) the Favre-averaged Navier-Stokes can be obtained (Wilcox, 2004),

$$\frac{\partial \bar{p}}{\partial t} + \frac{\partial}{\partial x_i} (\bar{p} \tilde{u}_i) = 0$$ (1.32)

$$\frac{\partial}{\partial t} (\bar{p} \tilde{u}_i) + \frac{\partial}{\partial x_j} (\bar{p} \tilde{u}_j \tilde{u}_i) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} (\sigma_{ji} - \rho \bar{u}_j^\prime \bar{u}_i^\prime)$$ (1.33)

$$\frac{\partial}{\partial t} \left( \bar{p} \left( \bar{e} + \frac{1}{2} \tilde{u}_i \tilde{u}_i \right) + \frac{1}{2} \rho \bar{u}_j^\prime \bar{u}_i^\prime \right) + \frac{\partial}{\partial x_j} \left( \bar{p} \tilde{u}_j \left( \bar{h} + \frac{1}{2} \tilde{u}_i \tilde{u}_i \right) + \tilde{u}_j \rho \bar{u}_j^\prime \bar{u}_i^\prime \right) =$$

$$\frac{\partial}{\partial x_j} \left( -q_j - \rho \bar{u}_j^\prime \bar{h}^\prime + \sigma_{ji} \bar{u}_i^\prime - \frac{1}{2} \rho \bar{u}_j^\prime \bar{u}_i^\prime \bar{u}_i^\prime \right) + \frac{\partial}{\partial x_j} \left( \tilde{u}_i \left( \sigma_{ji} - \rho \bar{u}_j^\prime \bar{u}_i^\prime \right) \right)$$ (1.34)
**Turbulent Closures: The Reynolds Stress and Turbulent Heat Flux**

The Favre-Averaged Navier-Stokes equations (1.32-1.34) above are very similar in form to the general Navier-Stokes defined in section 1.2.1 with a couple of key additional terms. These additional terms are unclosed and must be modeled to “close” and solve the equations.

The first of these terms is the Reynolds stress tensor, \( \bar{\rho}u_i''u_j'' \) (compressible) or \( \bar{u}_i'u_j' \) (incompressible) which appear in the right hand side of the momentum equation (1.33). This symmetric tensor provides 6 additional unknowns, which represent the mean transfer of momentum through turbulent motion. In classical RANS simulations the Reynolds stress is modeled through the use of the Boussinesq approximation and the turbulent viscosity concept.

\[
-\rho u_i''u_j'' = \sigma_{ij}^T = 2\mu_T \left( S_{ij} - \frac{1}{3} \frac{\partial \bar{u}_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \bar{\rho}k \delta_{ij} \tag{1.35}
\]

The term, \( \mu_T \), is the eddy viscosity and \( k \) is the turbulent kinetic energy (TKE). TKE is measure of the energy contained within the turbulent motion in the flow, and is defined as

\[
k = \frac{1}{2} \delta_{ij} \sigma_{kk}^T = \frac{1}{2}(u_1'u_1' + u_2'u_2' + u_3'u_3') \tag{1.36}
\]

The Boussinesq approach dictates that the turbulent stress is proportional to the mean velocity strain rate by the eddy viscosity term. This is consistent with the laminar stress tensor (1.15) with the additional \(-\frac{2}{3} \bar{\rho}k \delta_{ij} \) term. This term is included
to ensure the trace of $\sigma_{ij}^T$ is $-2\bar{p}k$. The addition of this extra term is necessary to represent the correct TKE in the momentum equations. With Boussinesq approximation included, the final term on the RHS of the momentum equation can be recast as,

$$\frac{\partial}{\partial x_j} \left( \sigma_{ji} + \sigma_{ij}^T \right)$$

(1.37)

$$\frac{\partial}{\partial x_j} \left[ (\mu + \mu_T) \left( 2\tilde{S}_{ij} - \frac{1}{3} \frac{\partial \tilde{u}_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \bar{p}k \delta_{ij} \right]$$

(1.38)

A further approximation for low speed flows is to replace the deviatoric strain tensor, which is trace-less, with the mean strain tensor,

$$\frac{\partial}{\partial x_j} \left( (\mu + \mu_T) 2\tilde{S}_{ij} \right)$$

(1.39)

In the averaged energy equation (1.34) the turbulent heat flux vector, $\overline{\rho u_i u_i'} h'$, arises during the time-averaging process and requires a turbulent closure similar to the Reynolds stress. Physically, this heat flux represents the transport of heat within the flow due to turbulent motion. In conventional RANS solvers the closure often accomplished through the use of the gradient diffusion hypothesis (GDH). The GDH defines a dynamic similarity between heat and momentum transfer through the use of a constant turbulent Prandtl number $Pr_t = \frac{\mu_t}{\alpha_t}$ and mean temperature gradient,

$$\overline{\rho u'_t h'} = -\frac{\mu_t}{Pr_t} \frac{\partial \tilde{h}}{\partial x_j} = -\frac{\mu_t c_p}{Pr_t} \frac{\partial \tilde{T}}{\partial x_j}$$

(1.40)
The heat flux terms within the energy equation can be combined in a similar
fashion to the viscous stress terms in the momentum equation. The first term on
the RHS of the energy equation becomes,

$$\frac{\partial}{\partial x_j} \left( -\left( \frac{\mu}{Pr} + \frac{\mu_T}{Pr_T} \right) c_p \frac{\partial \bar{T}}{\partial x_j} + ... \right)$$

(1.41)

Variable $Pr_t$ models exist to attempt better estimate the distribution of heat fluxes,
but have had difficulty matching experimental data (Wassel & Catton, 1973;
Bradshaw et al., 1991; Sommer et al., 1993; Ivanova et al., 2008). The GDH can be
extended to a scalar-flux model through the inclusion of the Reynolds stress tensor
to account for heat flux anisotropy (Daly & Harlow, 1970; Abe & Suga, 2001; Ling
et al., 2016). The generalized gradient diffusion hypothesis (GGDH) of Daly can be
expressed,

$$\overline{\rho u_i u^H} = -C_{GGDH} \tau \overline{\rho u_i u^\rho} c_p \frac{\partial \bar{T}}{\partial x_j}$$

(1.42)

The leading constant $C_{GGDH}$ is tunable and similar to the turbulent Prandtl
number in the basic GDH model, while $\tau$ is representative of the turbulent time
scale. The GGDH has been found to under predict the streamwise flux, $\bar{u}^\rho T^H$
compared to the vertical component, $\bar{v}^\rho T^H$ (Suga, 1995). This led to the
development of a higher order implementation, the higher-order generalized gradient
diffusion hypothesis (HPGGDH) (Abe & Suga, 2001).

$$\overline{\rho u_i u^H} = -C_{HOGGDH} \tau \overline{\rho u_k u^\rho} \overline{\rho u_k u^\rho} c_p \frac{\partial \bar{T}}{\partial x_j}$$

(1.43)
Both of these higher order extensions of the GDH require accurate Reynolds stress predictions, which pose separate challenges altogether.

Two additional terms remain on the right hand side of the energy equation. The first of these, $\overline{\sigma_{ij}u_i^0}$, corresponds to energy transport by molecular diffusion through turbulent motion. The second term, $\frac{1}{2}\overline{\rho u_j^0 u_i^0 u_i^0}$, is represents the average transport of TKE through turbulent motion.

### 1.2.3 The Spatially Filtered Navier-Stokes Equations

As will be discussed further in the next section, it is sometimes desirable to spatially filter the the Navier-Stokes equations, instead of time averaging them. The spatial filtering provides for a length-scale based separation of variable rather than the temporal separation utilized in the RANS equations. This is achieved through the use of a spatial filter function, $G$. To obtain the filtered state variable, $\overline{\phi}$, the convolution of the filter and instantaneous variable is taken,

$$\overline{\phi}(\vec{x}, t) = \int_{-\infty}^{\infty} G\left(\frac{\vec{x} - \zeta}{\Delta}\right) \phi(\zeta, t) \, d^3 \zeta$$  \hspace{1cm} (1.44)
This process will remove features from the flow field which are larger than the filters cutoff width, \( \Delta \). Several filters are used in fluids research, with the box filter (Eq. 1.45), and gaussian filter (Eq. 1.46) being some of the most common.

\[
G_{\text{Box}}(\zeta) = \begin{cases} 
\frac{1}{\Delta} & \zeta \leq \frac{\Delta}{2} \\
0 & \zeta > \frac{\Delta}{2}
\end{cases}
\] (1.45)

\[
G_{\text{Gauss}}(\zeta) = \left( \frac{6}{\pi \Delta^2} \right)^{1/2} e^{-\frac{6\zeta^2}{\Delta^2}}
\] (1.46)

The filtering process, \( \langle \phi \rangle \), will commute with differentiation and conventional averaging (notated \( <> \) here),

\[
\frac{\partial \overline{\phi}}{\partial t} = \overline{\left( \frac{\partial \phi}{\partial t} \right)}
\] (1.47)

\[
\langle \phi \rangle = \overline{\phi}
\] (1.48)

With these properties in mind, a length-scale based decomposition, similar to that done with the RANS equations, is now carried out.

\[
\phi'(x, t) \equiv \phi(x, t) - \overline{\phi(x, t)}
\] (1.49)

Here, \( \phi' \), is the field associated with the small length scales, sometimes referred to as the residual field. This residual field is similar in appearance to the fluctuating field.
in Reynolds decomposition, though it should be noted that the filtered residual field is not always zero.

$$\overline{\varphi}(x, t) \neq 0$$ (1.50)

Applying the spatial filter to the Navier-Stokes equations (Eq. 1.11-1.13) a set of equations can be formed for the large scale contents of the flow. The resulting equations are what is generally referred to as the Large Eddy Simulation (LES) equations.

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0$$ (1.51)

applying the Favre-average,

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho} u_i}{\partial x_i} = 0$$ (1.52)

The LES momentum equation is,

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}$$ (1.53)

applying the Favre-average,

$$\frac{\partial \bar{\rho} u_i}{\partial t} + \frac{\partial \bar{\rho} \bar{u}_i u_j}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \bar{\tau}_{ij}}{\partial x_j}$$ (1.54)
The second term on the left hand side can be expanded upon:

\[
\frac{\partial \tilde{p} \tilde{u}_i \tilde{u}_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \tilde{p} (\bar{u}_i + u'_i) (\bar{u}_j + u'_j) \right) \quad (1.55)
\]

\[
= \frac{\partial}{\partial x_j} \left( \tilde{p} \left( \tilde{u}_i \tilde{u}_j + \bar{u}_i u'_j + \bar{u}_j u'_i + u'_i u'_j \right) \right) \quad (1.56)
\]

This can be decomposed further to enable the separation of a resolved (larger than the filter) and unresolved length scale (smaller than the filter) equation (Garnier, Adams, & Sagaut, 2009).

\[
\frac{\partial \tilde{p} \tilde{u}_i \tilde{u}_j}{\partial x_j} = \frac{\partial}{\partial x_j} (\tilde{p} \tilde{u}_i \tilde{u}_j) + \frac{\partial}{\partial x_j} (L + C + R) \quad (1.57)
\]

\[
L = \tilde{p} \left( \tilde{u}_i \tilde{u}_j - \bar{u}_i \bar{u}_j \right) \quad (1.58)
\]

\[
C = \tilde{p} \left( \tilde{u}_i u'_j + u'_i \tilde{u}_j \right) \quad (1.59)
\]

\[
R = \tilde{p} \left( u'_i u'_j \right) \quad (1.60)
\]

The first term on the right hand side of Eq. 1.57 represents the desired resolved convective flux of the momentum equation. The terms L, C, and R, are the Leonard, Cross, and Reynolds stresses, respectively. The Leonard stress (Eq. 1.58) contains only resolved quantities, though can be inconvenient to compute. This leads to its association with the other unresolved stresses. The Cross stress (Eq. 1.59) is the result of interactions between the resolved and unresolved scales and therefore must be modeled. The Reynolds stress (Eq. 1.60) accounts for interactions which occur within the unresolved scales alone and also requires a modeled closure.
These three stress terms are often combined into a single term for convenience. This
term is referred to as the sub-grid-scale (SGS) stress, $\tau_{ij}^{SGS}$.

$$\tau_{ij}^{SGS} = p(u_i u_j - \tilde{u}_i \tilde{u}_j) \quad (1.61)$$

Finally, the spatially filtered momentum equation for LES can be written as, (1.35)

$$\frac{\partial \rho \tilde{u}_i}{\partial t} + \frac{\partial \rho \tilde{u}_i \tilde{u}_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \frac{\partial \tau_{ij}^{SGS}}{\partial x_j} \quad (1.62)$$

Filtering the energy equation (1.12) gives,

$$\frac{\partial}{\partial t} \left[ \rho e + \frac{1}{2} \rho u_i u_i \right] + \frac{\partial}{\partial x_j} \left[ \rho u_j h + \frac{1}{2} \rho u_j u_i u_i \right] = \frac{\partial}{\partial x_j} (u_i \sigma_{ij}) - \frac{\partial q_j}{\partial x_j} \quad (1.63)$$

after Favre-averaging,

$$\frac{\partial}{\partial t} \left[ \bar{\rho} e + \frac{1}{2} \bar{\rho} \bar{u}_i \bar{u}_i \right] + \frac{\partial}{\partial x_j} \left[ \bar{\rho} u_j h + \frac{1}{2} \bar{\rho} u_j u_i u_i \right] = \frac{\partial}{\partial x_j} (u_i \sigma_{ij}) - \frac{\partial q_j}{\partial x_j} \quad (1.64)$$

The second term on the left hand side of (1.64) can be treated in the same fashion
as the convective term in the momentum equation. Separating the resolved and
unresolved components, leads to:

$$\frac{1}{2} \bar{\rho} u_i \bar{u}_i = \frac{1}{2} \bar{\rho} \tilde{u}_i \tilde{u}_i + \frac{1}{2} \rho (u_i \bar{u}_i - \tilde{u}_i \tilde{u}_i) \quad (1.65)$$

$$= \frac{1}{2} \bar{\rho} \tilde{u}_i \tilde{u}_i + \tilde{k} \quad (1.66)$$
here, $\tilde{k}$, is the unresolved kinetic energy which must be modeled. The filtered velocity-enthalpy term on the left hand side gets an analogous decomposition, separating the resolved and unresolved heat fluxes.

$$
\bar{p}_{uj}h = \bar{p}_{uj}\tilde{h} + \bar{p}_{j}(\bar{u}_{j}h - \bar{u}_{j}\tilde{h})
$$

(1.67)

$$
= \bar{p}_{uj}\tilde{h} + q_{j}^{SGS}
$$

(1.68)

The remaining convective term on the left hand side of Eq. (1.64) contains a triple product and is expanded,

$$
\frac{1}{2}\bar{p}_{uj}u_{i}\tilde{u}_{i} = \frac{1}{2}\bar{p}(\bar{u}_{j} + u'_{j})(\bar{u}_{i} + u'_{i})
$$

(1.69)

$$
= \frac{1}{2}\bar{p}\left[\bar{u}_{j}\bar{u}_{i}\tilde{u}_{i} + 2\bar{u}_{j}\bar{u}_{i}u'_{i} + \bar{u}_{j}\bar{u}_{i}\tilde{u}_{i} + \bar{u}_{j}\bar{u}_{i}\tilde{u}_{i} + 2\bar{u}_{j}\bar{u}_{i}u'_{i} + \bar{u}_{j}\bar{u}_{i}\tilde{u}_{i}\right]
$$

(1.70)

and rearranged to separate the resolved and unresolved length scales:

$$
\frac{1}{2}\bar{p}_{uj}u_{i}\tilde{u}_{i} = \frac{1}{2}\bar{p}_{uj}\tilde{u}_{i}\tilde{u}_{i} + \frac{1}{2}\bar{p}\left[\bar{u}_{j}\bar{u}_{i}\tilde{u}_{i} - \bar{u}_{j}\bar{u}_{i}\tilde{u}_{i} + 2\bar{u}_{j}\bar{u}_{i}u'_{i}\right]
$$

(1.71)

$$
+ \frac{1}{2}\bar{p}\left[\bar{u}_{j}\bar{u}_{i}u'_{i} + \bar{u}_{j}\bar{u}_{i}\tilde{u}_{i} + 2\bar{u}_{j}\bar{u}_{i}\tilde{u}_{i} + \bar{u}_{j}\bar{u}_{i}\tilde{u}_{i}\right]
$$

With the exception of the blue terms, all of the unresolved terms are small and therefore typically neglected. The first blue term can be approximated as,

$$
\frac{1}{2}\bar{p}_{uj}\tilde{u}_{i}u'_{i} \approx \bar{u}_{j}\tilde{k}
$$

(1.72)
and the second blue term is approximated,

\[ \overline{p u_j i u_i} \approx \bar{u}_i \sigma_{ij}^{SGS} \]  

(1.73)

Moving now to the first term on the right hand side of Eq. (1.64), once again a decomposition of resolved and unresolved scales is employed,

\[ u_i \sigma_{ij} = \tilde{u}_i \sigma_{ij} + \left( u_i \sigma_{ij} - \tilde{u}_i \sigma_{ij} \right) \]  

(1.74)

\[ u_i \sigma_{ij} \approx \tilde{u}_i \sigma_{ij} \]  

(1.75)

The above decompositions and approximations are combined to give the final Favre-averaged LES energy equation,

\[ \frac{\partial}{\partial t} \left[ \bar{p} \bar{e} + \frac{1}{2} \bar{p} \bar{u}_i \bar{u}_i + \dot{k} \right] + \frac{\partial}{\partial x_j} \left[ \bar{p} \bar{u}_j \dot{h} + \frac{1}{2} \bar{p} \bar{u}_j \bar{u}_i + \bar{u}_i \dot{k} \right] = \]  

\[ \frac{\partial}{\partial x_j} \left[ \tilde{u}_i \sigma_{ij} + \tilde{u}_i \sigma_{ij}^{SGS} \right] - \frac{\partial}{\partial x_j} \left[ q_j + q_j^{SGS} \right] \]  

(1.76)

1.3 Scale Resolving Simulations

As discussed in the previous two sections the Navier-Stokes equations (1.11) - (1.13) can be averaged temporally to produce the RANS equations or spatially filtered to obtain the LES equations. In each case additional terms are added to the averaged equations, which at first glance may appear similar. However, they represent very...
different physical effects. To understand these modeling differences the turbulent energy spectrum must be understood.

Originally proposed by Richardson (1922) and expanded upon by Kolmogorov (1941), turbulent energy is transferred from the largest scales to subsequently smaller scales in a cascade like fashion. This is referred to as the energy cascade, and a schematic energy spectrum is shown in Fig. 1.5. The depicted energy spectrum shows the total energy as a function of wavenumber, $\kappa = \frac{1}{L}$. It can be seen in the spectrum that the bulk of the energy is contained at a wavenumber associated with some integral length scale, $L$. The integral length scale often be related to the driving fluid mechanics of the flow, such as the shedding of vortices behind a cylinder. The integral length scale can be computed from the energy spectrum,

$$L = \frac{\int_0^\infty \frac{E(\kappa)}{\kappa} d\kappa}{\int_0^\infty E(\kappa) d\kappa} \quad (1.77)$$

The dissipation range resides at the other end of the spectrum. Here the length scales are on the order of the Kolmogorov scale,

$$\eta_k = \left( \frac{L^3}{\nu} \right)^{1/4} \quad (1.78)$$

where $\nu$ is the molecular viscosity, and $\epsilon$ is the rate of turbulent dissipation. The Kolmogorov scales are the smallest within turbulent flow and represent the range at which turbulent eddies are dissipated to heat. Between the large energy containing
integral scale and the dissipative Kolmogorov scale is the inertial subrange. It is within the inertial subrange that Kolmogorov’s energy cascade is observed. Within the three regions energy is conserved, and the energy introduced at the integral scale is transferred through the inertial subrange and dissipated to heat at a constant rate.

The primary difference between the averaging approaches is related to how each method treats the energy spectrum depicted in Fig. 1.5. In the temporally averaged equations, either Reynolds- or Favre-averaged, the entire energy spectrum must be modeled. The modeled effects are accounted for in the additional terms, such as $\overline{u_i''u_j''}$ in Eq. (1.60), which appear after the averaging process. Wilcox provides an overview of common turbulence models and their details (Wilcox, 2004).

In the case of the filtered equations, the spatial filtering allows for the resolution of the largest length scales (low wavenumbers) of the energy spectrum, where the bulk
of the energy is contained. The amount of the energy spectrum which is resolved can vary greatly with flow conditions, grid resolution, and numerical model. Simulations which allow for the resolution of the energy containing scales are often referred to as Scale Resolving Simulations (SRSs). Generally, SRSs are split into different classes depending on the treatment of the energy spectrum. The three main SRSs classes are, Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), and hybrid RANS-LES methods such as Detached Eddy Simulation (DES). Each of these methods have different computational requirements, strengths and weaknesses.

The most computationally expensive of the SRSs is DNS. DNS allows for the complete resolution of the energy spectrum nearing or including the Kolmogorov scales. With increased Reynolds number the scale of the viscous effects is reduced. It follows then that the Reynolds number is a strong limiting factor in DNS simulations. Sagaut et al. (2013) shows that the number of grid points required for a full DNS computation scales with Reynolds number,

$$N_{xyz} = Re_L^{9/4}$$ (1.79)

With the extensive grid requirements comes very restrictive time step limits as well. These limitations restrict the use of DNS to Reynolds numbers far below those seen in practical aerospace applications. In applications where DNS resolution can be achieved, it does provide data sets with complete statistics related to turbulence and its transport. This can provide significant insight into the fundamental fluid
dynamics. For wall bounded turbulent flows, modern DNS techniques allow for the computation of flows with $Re_\theta$ in the low to mid thousands (i.e. $Re_\theta \in [2000, 4000]$) (Li, Schlatter, Brandt, & Henningson, 2009; Schlatter, Li, Brethouwer, Johansson, & Henningson, 2010; Poggie, Bisek, & Gosse, 2015).

Slightly less computationally expensive than DNS is LES. With LES, only the large energy containing scales are resolved with the small scales are removed from the simulations through the use of a filter. These small scales are referred to as subgrid and are of a size smaller than the computational grid. While DNS must resolve the entire energy spectrum, the requirements for LES are much more ambiguous. At least partial resolution of the inertial subrange is required, though how much is resolved varies from simulation to simulation (Pope, 2004; Sagaut et al., 2013).

As discussed in Section 1.2.3 the separation of the scales occurs through a spatial filtering process, and depends on a filter width, $\Delta$. For a given filter width (or grid resolution), the amount of turbulent spectrum resolved greatly depends on the flow regime of interest (e.g. wall bounded or free shear flows), as well as the numerics utilized. For wall bounded flows, even the largest scales present are physically very small and therefore require high grid resolution (Bradshaw & Perot, 1993). These small scale structures often require prohibitively large computational grids for proper resolution. Vyas et al. (2018) utilized $O(10^8)$ points for a supersonic flat plate LES. This is still relatively inexpensive compared to a true DNS simulation. Poggie et al. (2015) simulated a similar supersonic flat plate, albeit at a lower Reynolds number, with DNS levels of resolution, using up to $33 \times 10^9$ points.
The energy containing scales for free shear flows are generally much larger than those found in wall bounded turbulence, and are less dependent on the Reynolds number (Sagaut et al., 2013). Therefore, grid requirements for free shear flows are not as stringent as the wall bounded flows. Well resolved free jet LES can be achieved at grid levels of $O(3 \times 10^7)$ (Coderoni, Lyrintzis, & Blaisdell, 2018; DeBonis, 2018).

Subgrid models are often used to account for the unresolved turbulent transport. These models vary greatly in complexity, ranging from simple algebraic relations to full transport equations. One of the most common approaches is the Smagorinsky model (Smagorinsky, 1963). This model is an extension of Prandtl’s mixing length hypothesis (originally published in German, referenced here in English) (Prandtl, 1949) to the sub-grid scale. It follows the Boussinesq assumption that the turbulent stress tensor is directly proportionally to the mean (filtered) strain rate, through a sub-grid eddy viscosity,

$$\tau_{ij} = 2 \nu_{SGS} S_{ij}$$  \hspace{1cm} (1.80)

where $\tau_{ij}$ is the turbulent stress tensor, $\nu_t$ is the sub grid viscosity, and $S_{ij}$ is the resolvable strain rate. In the Smagorinsky model the sub-grid viscosity is given by,

$$\mu_{SGS} = \rho (C_s \Delta)^2 \sqrt{S_{ij} S_{ij}}$$  \hspace{1cm} (1.81)
$C_s$ is the Samgorinsky coefficient, which typically varies with the flow physics of interest. With the sub-grid viscosity computed, effective viscosity can be computed,

$$\mu_{eff} = \mu + \mu_{SGS} \quad (1.82)$$

Note, this closure is analogous to that done in Eq. 1.38 for the RANS closure.

Pope (2004) highlights the distinctions between a \textit{physical LES} and a \textit{numerical LES}. In general, the \textit{physical LES} utilizes a numerical scheme with negligible numerical errors, along with a sub-grid “physical” model to account for the transfer of energy from larger to smaller scales in the sub-grid inertial range. In contrast, \textit{numerical LES} is “...is fundamentally linked to the numerical method”, which contains some finite amount of numerical dissipation.

Some researchers choose to rely on the numerical dissipation associated with the differencing schemes to act as the subgrid model. This approach is referred to Implicit Large Eddy Simulation (ILES), and is one of the \textit{numerical LES} approaches described in Pope (2004). In some fundamental flows the amount of numerical dissipation can be estimated (Aspden, Nikiforakis, Dalziel, & Bell, 2008; DeBonis, 2013). However, for more even moderately complex flows (as is the case in this work) evaluating the numerical dissipation added by the scheme quickly becomes very difficult. For this reason ILES is becoming more common in active research communities. DeBonis (2018) used ILES to examine the turbulent heat flux in a heated free jet, while Coderoni et al. (2018) used ILES to investigate the
effects of fluidic injection on jet aeroacoustics. The LES performed in this work is done with implicit filtering (ILES).

Another class of SRSs are hybrid LES/RANS methods of simulation. With hybrid methods the simulation is split into regions of RANS, where the entire turbulent spectrum is modeled, and regions of LES, where the turbulent spectrum is resolved. The method of switching between LES and RANS varies in different implementations, but generally is performed by either sharp zonal boundaries or through blending functions depending on the hybrid model formulation. Spalart (1997) proposed the first Detached Eddy Simulation (DES) which allows for the switching between a RANS turbulent closure near the walls and LES away from the walls in separated flow regions. The original Spalart-Allmaras (SA) one-equation turbulence model (P. R. Spalart & Allmaras, 1992) is used for the RANS closure in attached regions of the flow. The SA model provides a transport equation for a turbulent viscosity term, $\tilde{\nu}$. This transport equation is given by,

$$\frac{\partial \tilde{\nu}}{\partial t} + u_j \frac{\partial \tilde{\nu}}{\partial x_j} = c_{b1} (1 - f_{i2}) S \tilde{\nu} - \left[ c_w f_w - \frac{c_{b1}}{k^2} f_{i2} \right] \left( \frac{\tilde{\nu}}{d} \right)^2 + \frac{1}{\sigma} \left[ \frac{\partial}{\partial x_j} \left( \nu + \tilde{\nu} \right) \frac{\partial \tilde{\nu}}{\partial x_j} + c_{b2} \frac{\partial \tilde{\nu}}{\partial x_i} \frac{\partial \tilde{\nu}}{\partial x_i} \right]$$  \hspace{1cm} (1.83)

The eddy viscosity can be computed from,

$$\mu_t = \rho \tilde{\nu} f_{v1}$$  \hspace{1cm} (1.84)
Table 1.2: Constants for the Spalart-Allmaras Turbulence model

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>$c_{b_1}$</th>
<th>$\sigma$</th>
<th>$c_{b_2}$</th>
<th>$\kappa$</th>
<th>$c_{w_2}$</th>
<th>$c_{w_3}$</th>
<th>$c_{t_3}$</th>
<th>$c_{t_4}$</th>
<th>$c_{w_1}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>0.1355</td>
<td>$2/3$</td>
<td>0.622</td>
<td>0.41</td>
<td>0.3</td>
<td>2</td>
<td>7.1</td>
<td>1.2</td>
<td>0.5</td>
</tr>
</tbody>
</table>

A number of algebraic relations must be defined for use in the transport equation (1.83),

$$f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3}$$  \hspace{1cm} (1.85)

$$\chi = \frac{\tilde{\nu}}{\nu}$$  \hspace{1cm} (1.86)

$$\tilde{S} = \Omega + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2}$$  \hspace{1cm} (1.87)

$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}$$  \hspace{1cm} (1.88)

$$\Omega = \sqrt{2 \Omega_{ij} \Omega_{ij}}$$  \hspace{1cm} (1.89)

$$\Omega_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)$$  \hspace{1cm} (1.90)

$$f_w = g \left[ \frac{1 + c_{w_3}^6}{g^6 + c_{w_3}^6} \right]^{1/6}$$  \hspace{1cm} (1.91)

$$g = r + c_{w_2} (r^6 - r)$$  \hspace{1cm} (1.92)

$$r = \min \left[ \frac{\tilde{\nu}}{S \kappa^2 d^2}, 10 \right]$$  \hspace{1cm} (1.93)

$$f_{t_2} = c_{t_3} e^{-c_{t_4} \chi^2}$$  \hspace{1cm} (1.94)

Constant parameters which are used within the SA model can be found in Table 1.2. The DES model of Spalart modifies the SA turbulence closure by replacing the
distance to the nearest wall, $d$, with a modified distance function $\tilde{d}$,

$$\tilde{d} = \min[d, C_{DES} \Delta]$$ (1.95)

where $C_{DES}$ is a constant coefficient, and $\Delta$ is a length scale associated with the local cell. The constant $C_{DES}$ has a default value of 0.62; however, it can be modified to affect the on/off switching of the turbulent viscosity. This modified distance function allows for the turbulent viscosity, $\mu_t$, to be switched off in regions away from the wall or where the grid is highly refined. Due to this grid dependence care must be taken to generate a computational mesh which maintains LES-like resolution in the “detached” regions, and a RANS-like grid in attached regions (P. Spalart, 2001; Sagaut et al., 2013). A significant reduction in computational costs can still be achieved due to the much reduced grid requirements in wall bounded regions. In recent years modification to the original DES methodology have been made. Spalart et al. (2006) developed the Delayed Detached Eddy Simulation (DDES) model. Here an improved switch detector is found to be more resistant to “ambiguous grid densities”. Menter et al. (2004) extended the DES methodology to a two-equation RANS closure.

**Statistical Covariances and Correlations in Scale Resolving Simulations**

The Reynolds stress ($u_i'u_j'$) and turbulent heat flux ($u_i'T'$) are open terms with approximate closures discussed briefly in section 1.2.2. To better model these terms
it is important to understand both the physical and mathematical meanings of the terms. Both the turbulent heat flux vector and the Reynolds stress tensor are composed of statistical covariances. In statistics the covariance is used to describe the relationship between different signals or processes. A positive value indicates that the quantities exhibit a direct correlation, meaning an increase in quantity 1 statistically results in an increase in quantity 2. Conversely, a negative covariance would mean an inverse correlation where an increase in quantity 1 statistically corresponds to a decrease in quantity 2. These covariances are used to represent the effects the unresolved turbulent motion has on the governing equations. The use of SRSs allows for the direct computation of these unknown covariances. This information can be used to formulate new and improved turbulence closures.

The Reynolds stress tensor, $u''_i u'_j$, can be expanded to,

$$u''_i u'_j = \begin{bmatrix}
    u''u'' & u''v'' & u''w'' \\
    v''u'' & v''v'' & v''w'' \\
    w''u'' & w''v'' & w''w''
\end{bmatrix}$$

shows that there are 6 unique ($u''_i u'_j$ is a symmetric second order tensor) velocity covariances which comprise the tensor. The diagonal terms are the velocity auto-covariances, meaning the covariance of the velocity signals with themselves. These auto-covariances are representative of the normal turbulent stresses. The off-diagonal terms are the covariants of the different velocity components with each
other, and correspond to the turbulent shear stresses. While the turbulent heat flux vector, $\overline{u''T''}$, expands to,

$$\overline{u''T''} = \langle u''T'', v''T'', w''T'' \rangle$$ (1.97)

which contains three velocity-temperature covariances. Both the Reynolds stresses and the turbulent heat fluxes are desired outcomes from the current simulations. There computation can be found in the definition of the Reynolds average. The shear ($u''v''$) component of the Reynolds stress tensor,

$$uv = (\overline{u} + u'') (\overline{v} + v'')$$ (1.98)

$$\overline{uv} = \overline{u} \overline{v} + u''v'' + \overline{u''v''} + \overline{u''v''}$$ (1.99)

$$\overline{u''v''} = \overline{uv} - \overline{u} \overline{v}$$ (1.100)

With this definition, it can be seen that the covariance of two quantities can be obtained with the mean of each of the quantities ($\overline{u}$ and $\overline{v}$) and the mean of the product of the two quantities ($\overline{uv}$). Running averages of these can be stored allowing to the monitoring of the various covariances required.

### 1.3.1 Synthetic Turbulence Generation

As previously discussed the objective of SRSs is to resolve the physical scales, and the associated energy content of turbulent motion. To accomplish this, particularly
in LES, special care must be given to the inflow boundary conditions. In the case of spatially evolving turbulence, the transition from laminar to turbulent flow can be resolved numerically (Sayadi, Hamman, & Moin, 2011). However, this transition initiates in the smallest scales within the boundary layer, and as such requires DNS or near-DNS levels of grid resolution. This is impractical in most research circles, and therefore methods to apply artificial turbulence to the inflow boundary have been developed. Ideally, the methods utilized should be able to provide first and second order statistics for the desired flow conditions.

A wide range of techniques have been developed for the application of unsteady inflow profiles to SRS. Lund et al. (1998) developed a methodology to scale the results from a precursor simulation to the conditions desired for the present simulation. This technique is still commonly used (Johnson & Kapat, 2013). However, the method is limited by the additional computational expense of a secondary simulation, as well as being limited to somewhat simple geometries.

Another common approach is to numerically trip the boundary layer. This is often done through the use of body force terms (Mullenix, Gaitonde, & Visbal, 2013; Poggie et al., 2015). These methods provide a more physically natural approach. However, this approach can be very computationally expensive as long transition lengths can be required, especially for higher Reynolds numbers.

One methodology, the Synthetic Eddy Method (SEM), was developed by Jarrin et al. (2009). Here, inflow turbulence is generated at the inflow boundary plane based
on a prescribed turbulent length scale and Reynolds stress tensor. The SEM approach is computationally inexpensive, adding little overhead to the overall simulation. Sufficient length must be given for the artificial turbulence to transition into realistic turbulence. This transition typically occurs within 10-15 boundary layer thicknesses (Mankbadi, Vyas, DeBonis, & Georgiadis, 2018). Poletto et. al. (2013) extended the SEM methodology to include a divergence free condition at the inflow plane. This divergence free SEM (DF-SEM) is the approach used herein, and is described below.

In the original SEM described by Jarrin et al. (2009) the $i^{th}$ component of the velocity signal is defined as:

$$u_i = U_i + \frac{1}{\sqrt{N}} \sum_{k=1}^{N} a_{ij} \epsilon_j f_{\sigma}^k \left( \vec{x} - \vec{x}_k \right)$$

(1.101)

Here, $U_i$ is the local mean velocity to be perturbed by the synthetic turbulence.

There must be a sufficient number of eddies, $N$, to statically fill the inflow plane bounding box. The location of the $k^{th}$ eddy is centered at $\vec{x}_k$. $\epsilon_j^k$ is a signed intensity of the $k^{th}$ eddy and is an independent random number. The shape of the synthetic eddy is represented by the shape function,

$$f_{\sigma}^k \left( \vec{x} - \vec{x}_k \right) = \sqrt{V_B \sigma^{-3}} f \left( \frac{x - x^k}{\sigma} \right) f \left( \frac{y - y^k}{\sigma} \right) f \left( \frac{z - z^k}{\sigma} \right)$$

(1.102)
Here a basic tent function is used,

\[
f(x) = \begin{cases} 
\sqrt{3/2} (1 - |x|) & x < 1 \\
0 & \text{elsewhere} 
\end{cases}
\] (1.103)

The Cholesky decomposition of the Reynolds stress tensor is represented by the tensor, \(a_{ij}\), and can be expanded as:

\[
a_{ij} = \begin{bmatrix} 
\sqrt{R_{11}} & 0 & 0 \\
R_{21}/a_{11} & \sqrt{R_{22} - a_{21}^2} & 0 \\
R_{31}/a_{11} & (R_{32} - a_{21}a_{31})/a_{22} & \sqrt{R_{33} - a_{31}^2 - a + 32^2} 
\end{bmatrix}
\] (1.104)

Poletto et al. (2013) extended the method to provide a divergence free velocity field. To ensure the synthetic turbulence field is divergence free, the vorticity field due to the original perturbation velocity field is evaluated,

\[
\nabla \times \omega' = \nabla (\nabla \cdot u') - \nabla^2 u'
\] (1.105)

Since the desired result is, by definition, incompressible, the first term on the RHS must be zero, resulting in a Poisson equation.

\[
\nabla \times \omega' = -\nabla^2 u'
\] (1.106)
Using the Biot-Savart kernel Poletto et al. solved for a velocity perturbation field,

\[ u'_i = \sqrt{\frac{1}{N} \sum_{k=1}^{N} q_\sigma \left( \frac{|r^k|}{|r^k|^3} \right) r^k \times a^k_i} \] (1.107)

Here, \( r^k = \frac{x-x^k}{a_k} \), and \( q_\sigma \) is a shape function. The eddy intensities are represented by \( a^k_i \). Notably absent from this formulation of the perturbation field is the Cholesky decomposition of the Reynolds stress tensor, \( a_{ij} \). This absence eliminates the control of the turbulence anisotropy. The reintroduction of this term would violate the desired divergence free condition. To reintroduce the turbulence anisotropy the method of Poletto et al. reformulates the shape function to include the length scale anisotropy. The derivation of the shape function must maintain the divergence free condition for the velocity perturbation, \( \nabla \cdot u' = 0 \). Poletto et al. suggested a simple shape function which satisfies the divergence free condition, \( q_i \),

\[ q_i = \begin{cases} \sigma_i \left[ 1 - (d^k)^2 \right] , & d_k < 1 \\ 0 , & \text{elsewhere} \end{cases} \] (1.108)

with, \( d^k = \sqrt{(r^k_j)^2} \).

The final form of the velocity perturbation field can then be expressed as,

\[ u'_\beta = \sqrt{\frac{1}{N} \sum_{k=1}^{N} a^k_\beta \left[ 1 - (d^k)^2 \right] \epsilon_{\beta ji} r^k_j \alpha^k_i} \] (1.109)
Here $\epsilon_{\beta j i}$ represents the Levi-Civita symbol, $\sigma^k_i$ the turbulent length scale components, and $\alpha^k_i$ the intensities.

Finally, it must be noted that because the eddy intensities are independent and therefore uncorrelated, the modeled shear stresses becomes zero. This is compensated for using an axis transformation. The eddy field is initially generated in the principal axes coordinate system of the Reynolds stress tensor. In this principal coordinate system the Reynolds stress tensor will be diagonal and therefore only contain normal stresses. The perturbations in the principal axis are then transformed in into the global system, reintroducing the cross-correlations (i.e. shear) components of the Reynolds stress tensor, viz.,

$$u^G_i = C_1 R_{im}^{P \rightarrow G} u^P_m$$

(1.110)

where, $R_{im}^{P \rightarrow G}$ is the coordinate transformation between the principal and global coordinate systems, and $u^G_i$ and $u^P_m$ are the perturbation velocity fields in the global and principal axes respectively. The constant, $C_1$ is a required normalization factor,

$$C_1 = \frac{\sqrt{10V_0} \sum_{i=1}^{3} \frac{\sigma_i}{\Xi} \min (\sigma_i)}{\sqrt{N \Pi_{i=1}^{3} \sigma_i}}$$

(1.111)

The original paper of Poletto et al. (2013) gives further details pertaining to the derivation and validation of the DF-SEM method.
Other common methods aim to generate synthetic turbulence at the inflow plane. Klein et al. (2003) developed a method to digitally filter perturbations onto the computations mesh. Spectral methods, which prescribe a predefined distribution of turbulent energy spectrum have also been developed (Lee, Lele, & Moin, 1992). Keating et al. (2004) have provided a comparison of several methods including on the effects which the turbulent length scales have on the simulations accuracy. A recent comprehensive review of synthetic turbulence generation methods has been done by Dhamankar et al. (2018).
2. EXPERIMENTAL AND NUMERICAL SETUP

As discussed previously, the analysis of turbulent heat fluxes remains an active area of research. Understanding the transport of heat through turbulent motion is critical for the advancement of aerothermal system design. The following sections will describe the experimental and numerical setups used herein to examine the turbulent heat fluxes experienced in the jet in a cross flow (JICF) representative of film cooling configurations.

2.1 Experimental Setup

An experiment was conducted in a wind tunnel at NASA Glenn Research Center, ERB test cell SW-6, to acquire turbulent heat flux measurements on a large scale model of film cooling/heating holes. The experiment was conducted, and data collected by Philip Poinsatte and Douglas Thurman of NASA GRC. The raw data was then processed and analyzed by the author. The test facility, shown in Fig. 2.1, consists of an aluminum bellmouth, flow conditioning honeycomb and screens, and a square acrylic section 0.2 m x 0.2 m wide with 1.90 cm thick walls. Ambient air is entrained into the tunnel via the central exhaust system. The central exhaust system at NASA GRC is a lab wide service which provides test cells a
vacuum down to 0.14 Bar. The test section consisted of an interchangeable floor section fabricated from ABSplus thermoplastic in a 3-D printer, with three injection holes for periodic flow and boundary conditions. The hole diameter was 1.905 cm (0.75 inch) and was inclined at 30° from the horizontal surface. The spanwise spacing was set to $Z/D=3$. The coordinate system is right handed, with X aligned in the tunnel axial direction, Y in the wall normal direction, and Z in the tunnel spanwise direction. The origin is located at the leading edge of the cooling hole.

Figure 2.1: SW-6 Facility in Center Wide Exhaust Configuration. From Wernet et. al. 2016

A PC-based data acquisition system was used to acquire data from pressure transducers and thermocouples. The tunnel flow rate was measured from a total pressure probe placed just upstream of the test section and static pressure taps located on the sidewalls at the same plane. Freestream temperature was measured with a thermocouple located upstream of the holes near the total pressure probe. Tunnel flow was nominally set to 9.14 m/s at ambient conditions which
corresponded to a Reynolds number based on hole diameter and freestream velocity of 11,000. Freestream turbulence was measured to be less than 1% at the inlet to the test section, and boundary layer thickness measured around 2 cm upstream of the injection holes.

The injection flow was provided by blowing pressurized air through a flow meter and into a plenum attached to the underside of the test section floor plate. The tubes connecting the plenum to the test section holes had a length of approximately $L/D=20$, which is sufficient to ensure a turbulent velocity profile in the tube. The injection flow conditions were measured with static pressure taps and thermocouples both inside the plenum and in each injection tube. In previous film cooling tests with this tunnel, the injected air was cooled by passing through copper tubing that was coiled inside an ice water tank (Thurman et al., 2016). However for this effort, a larger temperature difference was desired, so the injection air was heated using three 750 W in-line electrical pipe heaters. This provided a temperature difference of approximately 36 °K. Note that this heated jet configuration would not be typical in a turbomachinery application, but was believed to be an appropriate analogy.

Three different probe configurations were used to take velocity and temperature measurements. A 3.8$\mu$m single wire probe was initially used for temperature measurements. Two different 3-wire probes (x-T probes) were then used to make simultaneous velocity and temperature measurements. Both sets of x-T probes consisted of a 1.2$\mu$m wire 0.4 mm in length for temperature measurements (T-wire). Velocity ($\rho V$) measurements were made using two 25$\mu$m wires (x-wires) positioned
perpendicularly to one another. The law of cosines was then used to reconstruct to velocity components from the two perpendicular velocity signals. The T-wire was connected to a CCA system to measure temperature, while the x-wires were operated in CTA mode to obtain velocity. One x-T probe was oriented for U- and V- velocity measurements, while the other probe measured U- and W- velocity components. The U-V and U-W velocity measurements were taken independently of each other. The T-wire in the U-V probe was slightly offset spanwise (W velocity direction), while the T-wire in the U-W probe was vertically offset (V velocity direction). It is found that these wire offsets must be accounted for properly when evaluating turbulence quantities. The x-T probes used here have probe volumes of roughly $0.15D \times 0.2D \times 0.15D$ with the long dimension oriented in the secondary velocity direction (ie. in the V direction for the U-V probe). Figure 2.2 shows the U-V and U-W probes, with the X-wires highlighted in blue and the T-wires highlighted in yellow.

Pressure, temperature, and velocity surveys were taken along the centerline in the streamwise direction and at cross sectional planes $X/D = 2, 3, 4, 6,$ and 10, as measured from the hole leading edge. Figure 2.3 shows an example of a mean temperature measurement with the probe locations overlaid. An actuator system was used to position the probe in the tunnel. The pressure surveys were taken with a pitot-static probe. Data at nominal blowing ratios of 1.2 and 2.4 were acquired. Both velocity and temperature measurements were averaged for 3 seconds at a frequency of 50 kHz. The surveys were completed over several days.
The probes were calibrated in a separate calibration tunnel by varying the velocity and temperature of the flow and recording the voltages for each wire. Since the CTA hot-wire is sensitive to the temperature of the flow, a series of calibration curves at various velocities and temperatures were used to interpolate $\rho V$ from the wire voltage. Static pressure and temperature surveys were used to calculate a mean density to obtain the velocity. Table 2.1 lists the conditions considered for the experimental data collected. However, numerical simulations were only conducted at $BR = 1.2$, therefore only the lower blowing ratio will be discussed herein. The analysis of the higher blowing ratio will be left for future research.
Figure 2.3: Sample of experimental temperature measurement. Black squares represent the probing locations for the experimental surveys.

Table 2.1: Nominal experimental conditions

<table>
<thead>
<tr>
<th>BR</th>
<th>$V_{\text{Tunnel}}$ (m/s)</th>
<th>$\rho_{\text{Tunnel}}$ (kg/m$^3$)</th>
<th>$T_{\text{Tunnel}}$ (K)</th>
<th>$V_{\text{Tube}}$ (m/s)</th>
<th>$\rho_{\text{Tube}}$ (kg/m$^3$)</th>
<th>$T_{\text{Tube}}$ (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1.2</td>
<td>9.14</td>
<td>1.174</td>
<td>13.7</td>
<td>1.038</td>
<td>336</td>
</tr>
<tr>
<td></td>
<td>2.4</td>
<td>9.14</td>
<td>1.169</td>
<td>25.3</td>
<td>1.032</td>
<td>336</td>
</tr>
</tbody>
</table>

2.1.1 CCA frequency compensation

To account for the low-pass filtering inherent to CCA measurements, the numerical compensation technique of Tagawa et al. (2005) was implemented. The technique provides a closed form equation for the frequency response of a given fine-wire thermometer. The model is based solely on physical properties of the probe, and the
local flow properties. It can therefore be easily applied to a wide range of CCA probes without the need for extra characterization experiments.

The goal of the technique is to provide a transfer function that models the frequency response of the CCA probe. The approach treats the individual components of the CCA probe separately, linked together through boundary conditions. The time constant of each component is given by,

$$\tau_i = \frac{1}{2} \pi f_{ci} = \frac{\rho_c c_i d_i^2}{4 Nu \lambda_g}, \quad i = 1, 2, 3$$  \hspace{1cm} (2.1)

The Nusselt number, Nu, can be obtained through empirical relations as a function of the wire diameter based Reynolds number. The Collis-Williams relations (Collis
& Williams, 1959) are used in this work as suggested in the original Tagawa et al. paper.

\[
Nu(T_f/T_g)^{-0.17} = 0.25 + 0.56Re^{0.45} \tag{2.2}
\]

where \(T_g\) is the gas temperature, and \(T_f = 0.5(T_s + T_g)\) is the film temperature.

Due to the assumptions inherent to CCA, namely that the sensor wire is the same temperature as the gas, it is a reasonable assumption to say the surface temperature, \(T_s\), is equal to the gas temperature. Therefore,

\[
Nu \approx 0.25 + 0.56Re^{0.45} \tag{2.3}
\]

In the case of low Reynolds number flow (\(Re < 0.5\)) a different empirical relation must be used to account for Knudsen number effects (Collis & Williams, 1959).

\[
Nu = \frac{1}{1.18 + 2Kn - 1.1log_{10}(Re)} \tag{2.4}
\]

\[
(2.5)
\]

while the Knudsen number expressed as a function of Mach and Reynolds number is,

\[
Kn = \frac{Ma}{Re} \sqrt{\frac{\gamma \pi}{2}} \tag{2.6}
\]
A common parameter appearing for each of the wire components has been defined,

\[ \Omega_i = \sqrt{1 + \frac{j \omega \tau_i}{a_i \tau_i}}, \quad i = 1, 2 \]  \hspace{1cm} (2.7)

The subscripts above represent the different components of the CCA probe. Figure 2.4 depicts these components. The first, \( i = 1 \) is the sensor itself. Next, is the support stubs \( i = 2 \). The prongs are represented by \( i = 3 \). Here, \( \rho_i, c_i, a_i \) are the density, specific heat, and thermal diffusivity respectively of the \( i^{th} \) CCA component. The gas thermal conductivity is represented by, \( \lambda_g \). The Nusselt number, \( \text{Nu} \), can be evaluated using empirical relations for flow around a cylinder.

The nondimensional time constant for the prongs, \( \tau_3 \) is assumed to be unity. The final transfer function averaged over the length of the sensor is given by,

\[ H(\omega) = \frac{1}{1 + j \omega \tau_1} - \frac{1}{(1 + j \omega \tau_2)} - \frac{1}{(1 + j \omega \tau_3)} + \frac{1}{1 + j \omega \tau_1} - \frac{1}{1 + j \omega \tau_2} \cosh\left(\frac{\Omega_1(L-l)}{2}\right) \hspace{1cm} (2.8) \]

The transfer function is applied to the temperature signal in the frequency domain. The Fast Fourier Transform (FFT) of the time series temperature data is computed, and the compensation transfer function applied,

\[ Y(\omega) = \text{FFT}(T(t)) \]  \hspace{1cm} (2.9)

\[ Y_c(\omega) = \frac{Y(\omega)}{H(\omega)} \]  \hspace{1cm} (2.10)
The compensated temperature spectrum is then returned to the time domain through the application of an inverse-FFT (IFFT),

\[ T_c(t) = IFFT(Y_c(\omega)) \] (2.11)

\( T_c(t) \) is the compensated temperature signal as a function of time. The new estimated temperature turbulent statistics may then be computed. For the full derivation and a rigorous analysis of the boundary conditions applied within, see the original paper of Tagaw et al. (2005).

2.2 Numerical Setup

Three methods of numerical simulation were used to evaluate the film cooling JICF problem described above. Conventional RANS turbulent closures are examined as a baseline set of simulations. Two different SRSs approaches were examined, to gain a more fundamental understanding of the underlying flow physics involved in the turbulent transport of heat. The first approach is a DES model, and the second approach is a fully wall resolved series of LES cases. The LES cases utilized the Divergence Free Synthetic Eddy of Method (DF-SEM) of Poletto et al. (2013) to emulate realistic inflow turbulence. The following sections will describe the numerical methods used for each of these simulation sets.
2.2.1 RANS Simulations

RANS simulations were conducted to provide a baseline of the JICF configuration examined within the experimental setup, as well as provide a data set for the examination of the gradient diffusion hypothesis. A series of simulations were conducted with different turbulent closures and grid resolutions. The specifics of these simulations will be discussed in the following subsections.

Grid and Boundary Conditions

Two different grid levels were used for the RANS simulations presented here. The grid levels, referred to as “fine” and “very fine” use identical boundary conditions, and grid topologies, with differing grid counts and spacing. The very fine grid is identical to the LES grid described later, and contains approximately 68 million structured hexahedral cells, while the fine grid contains approximately 34 million hexahedra. Both grid levels are wall resolved, with a wall spacing set to 25.4 $\mu$m, along the channel walls to obtain a target of $y^+ \leq 1$, based on a tunnel velocity of 9.14 m/s.

In the fine grid the wall normal spacing stretched to $\Delta y^+ \approx 42$ at $Y/D = 1$ and $\Delta y^+ \approx 50$ at $Y/D = 2$. This doubles the wall normal stretching seen in the very fine grid which saw stretching to $\Delta y^+ \approx 21$ at $Y/D = 1$ and $\Delta y^+ \approx 25$ at $Y/D = 2$. The streamwise spacing for each grid was roughly constant, except for the region near the hole exit where the viscous wall spacing within the tube needed to be matched.
The fine grid spacing was $\Delta x^+ \approx 80$, while the very fine grid had uniform spacing of $\Delta x^+ = 15$. Far downstream of the cooling hole the streamwise spacing did stretch near the domain exit, however this was outside of the examined region of the domain. The spanwise spacing for the two grid levels remains constant, with an average spacing of $\Delta z^+ \approx 9$, with a maximum value at the symmetry boundary conditions of $\Delta z^+ = 15$. The sidewalls are treated as symmetry conditions, as is appropriate for steady RANS simulations. The grid within the cooling tube is of an O-H topology comprised of 5 blocks, as seen in Fig. 2.5. This topology is extruded from the tunnel floor to the top of the computational domain.

A summary of the boundary conditions is:

- **Tunnel Inflow**: Velocity profile fixed, while pressure is allowed to float.

- **Outflow**: A pressure outflow condition is used. This extrapolates the velocity while fixing the pressure.

- **Cooling Holes Inflow**: Velocity profile fixed.

- **Viscous Walls**: Walls are treated as viscous and adiabatic.

- **Top**: Inflow-outflow condition. Allows the boundary to act as either a velocity inlet or pressure outlet depending on the local velocity vector.
Figure 2.5: O-H grid topology of the cooling tube exit plane. Edge of the cooling tube exit outlined in red. Every fifth grid line shown.

**Numerical Procedure**

The OpenFOAM software package is used to preform RANS simulations on the jet in crossflow cases examined herein. The pressure-based rhoSimpleFOAM solver is utilized to solve the Navier-Stokes and energy equations. This solver uses the pressure-velocity coupling of the SIMPLE(C) algorithm. The energy equation is formulated with the calorically-perfect, sensible internal energy, and the ideal gas law is used for the equation of state. Sutherland’s law is used for the gas viscosity and thermal conductivity. Second order upwinded schemes are used for the spatial derivatives. The velocity profiles mapped to the velocity inlet boundary conditions were obtained from time averaged LES generated from the SEM boundary conditions, to be discussed in section 2.2.3. The solutions were initialized by
mapping constant conditions to the tube and tunnel regions. The tube was initialized to a velocity of 12.6 m/s at 336 K, aligned along the tube axial direction. The tunnel was initialized in the axial direction to a velocity of 9.14 m/s at 300 K, giving the initial condition an effective blowing ratio of BR=1.2. The solutions were initially converged with first order derivatives to a residual level of 1x10^-7. After the initial convergence the schemes were increase to second order, and convergence was achieved again.

2.2.2 Detached Eddy Simulations

A Detached Edddy Simulation (DES) was performed on the JICF film cooling problem to compare with the experimental setup described above. The “detached” nature of the jet seen at the BR of interest here suggested the hybrid RANS/LES approach would be an efficient method of modeling the interaction area of interest. The details of the simulation is described in the following sections.

Grid and Boundary Conditions

The computational domain was set to match the inflow lengths observed in the SW-6 test facility described in Section 2.1, with full tunnel and tube lengths. The grid was packed to the walls with the first point well in the expected viscous sublayer, with a wall spacing set to 25.4 µm, along the channel walls to obtain a target of y^+ ≤ 1, based on a tunnel velocity of 9.14 m/s. Given the time resolved
nature of DES simulations, periodic conditions are used for the side walls. Approximately 45 million structured hexahedral cells were used. Figure 2.6 shows the computational domain used, with the cooling hole grid topology shown. An O-H grid topology similar to the one used in the RANS simulations was used here. The initial \( y^+ = 1 \) was not allowed to stretch to more than a \( \Delta y^+ = 24 \) until a height of \( y/D = 4 \) above the tunnel floor. The spanwise spacing is nearly constant (away from the cooling hole exit) at \( \Delta z^+ \approx 15 \). Streamwise grid spacing differs depending on the region of the flow domain. The cooling tube and tunnel inflow have grid spacing which is appropriate for RANS simulations, \( \Delta x^+ \approx 500 \). While the region downstream of the cooling hole is expected to “detached” and is therefore adheres to LES grid requirements of \( \Delta x^+ \approx 15 \). Downstream of the jet exit \( (X/D > 6) \), the axial spacing begins to stretch, returning to a spacing more appropriate for RANS simulations. The boundary conditions are intended to match the physical properties of the wind tunnel,

- **Tunnel Inflow**: Stagnation conditions are held constant, to match the conditions in the SW-6 test cell at the time of the experiment.

- **Outflow**: Static pressure is imposed at the outflow domain, such that the tunnel massflow rate matches the experiment.

- **Cooling Holes Inflow**: Stagnation conditions are held constant at the cooling tube inflow domain, to roughly match the plenum conditions. Since exact
experimental measurements are not available, this boundary was adjusted to match the desired blowing ratio.

- *Viscous Walls*: Walls are treated as viscous and adiabatic.

![Computational domain used with the DES simulations. Insert: Topology of the tunnel-tube grid interface. Every third grid line is shown to better show the grid details.](image)

**Numerical Procedure**

The Spalart-Allmaras based Detached Eddy Simulation (DES) (P. R. Spalart et al., 1997) implemented in Ansys Fluent was used for this simulation. This approach is a hybrid RANS/LES approach in which the distance function within the RANS-SA
model is compared to a local grid length scale. Specifically, the distance function within the RANS-SA model is replaced with,

\[ \tilde{d} = \min[d, C_{DES} \Delta] \]  \hspace{1cm} (2.12)

where \( d \) is the distance to the nearest wall, \( C_{DES} \) is a constant coefficient, and \( \Delta \) is a length scale associated with the local cell. The DES model behaves like the standard RANS model when near surfaces, and switches to an LES calculation with Smagorinsky-like subgrid scale modeling when away from the walls. A second order implicit time marching scheme is used for the DES presented here. A timestep of \( \Delta t \approx 0.01 \tau (\tau = \frac{D}{u_\infty}) \) was used for the simulations, and sampled for a minimum period of \( T = 72 \tau \) after sufficient start-up time has passed. Assuming a Strouhal number, \( St = 0.2 \), this allows for roughly 15 cycles to be sampled at 500 samples per cycle. Fluent’s bounded central differencing (BCD) scheme was used for the DES.

\subsection*{2.2.3 Large Eddy Simulations with Synthetic Turbulence}

\textbf{Numerical Scheme}

The OpenFOAM software package is used to perform large eddy simulations (LES) on the jet in crossflow cases examined herein. The pressure-based rhoPimpleFOAM solver is utilized to solve the Navier-Stokes and energy equations. This solver combines a PISO-like time marching scheme with the pressure-velocity coupling of
the SIMPLE algorithm. The energy equation is formulated with the
calorically-perfect, sensible internal energy, and the ideal gas law is used for the
equation of state. Sutherland’s law is used for the gas viscosity and thermal
conductivity. Bounded second order schemes are used for the spatial derivatives,
and a Crank-Nicolson scheme is used for time advancement. The Crank-Nicolson
scheme includes an off-centering coefficient which can be used to increase stability
at the cost of overall scheme order. A coefficient of 0.25 was required for stability
across all of the simulations. The divergence-free synthetic eddy method (DF-SEM)
of Poletto et al. (2013) was used for the generation of turbulence at the domain
inlets. While not without limitations, the DF-SEM approach does a good job of
quickly providing a realistic incoming boundary layer. Stability was most limiting
with SEM applied within the tube.

Grid and Boundary Conditions

Two grid levels are examined for a grid sensitivity study. Both grids are packed to
the walls with the first point well in the expected viscous sublayer, with a wall
spacing set to 25.4 µm, along the channel walls to obtain a target of \( y^+ \leq 1 \), based
on a tunnel velocity of 9.14 m/s. The side walls are cyclic to simulate a infinite span
of injectors. Both grids are constructed in a similar topology, with the injector
being comprised of 5 structured zones in an O-H formation. These zones then
extend vertically through the tunnel domain. The fine grid domain extends
$X \in [-3.3D, 14D], Y \in [0, 5.5D], Z \in [-1.5D]$ with the tube extending $8.67D$
upstream of the jet exit. The spanwise spacing is capped at $\Delta Z^+ \leq 15$ and reduces in regions near the hole exit. Similarly, the streamwise spacing is capped at $\Delta X^+ \leq 30$ but is generally around $\Delta X^+ = 15$ in area of interest downstream of the hole. The wall spacing is allowed to stretch up to a maximum value of $\Delta Y^+ = 75$
two hole diameters above the tunnel floor. The fine grid is comprised of roughly 35 million cells.

The very fine grid has an extended tunnel inflow ($+7D$) to allow for SEM transition, bringing the full domain extent to $X \in [-10.33D, 14D]$. The Y and Z extents remain the same as the fine grid level. In the fine grid the spanwise spacing is mostly constant and capped at $\Delta Z^+ \approx 9$, but does reduce in regions near the hole exit. The streamwise spacing is capped at $\Delta X^+ = 20$, the grid is allowed to stretch slightly far downstream of the hole ($\frac{X}{D} \approx 12$). The wall normal spacing is significantly refined stretching to a maximum of $\Delta Y^+ = 25$, two tube diameters above the tunnel floor. The circumferential spacing is decreased slightly within the tube on the fine grid as well. The very fine grid is comprised of roughly 68 million cells. Figure 2.7 shows the extent of the computational domain used in the computations.

The boundary conditions are identical for each grid,

- **Tunnel Inflow**: A fully developed velocity profile and Reynolds stress profile (for SEM) are applied. The fine grid inflow is split into two regions to limit
Figure 2.7: Schematic and isometric view of the computational domain. Isosurface of Q-criterion colored by temperature fluctuations.

(a) Schematic of the computational domain

(b) Isometric view of computational domain with isosurface of Q-criterion

the extent of the SEM to the boundary layer only. The profiles are obtained from a $k - \omega$ SST solution.

- **Outflow**: Static pressure is imposed at the outflow domain, and the velocity is held to a zero-gradient normal to the exit.
• **Cooling Holes Inflow:** Similar to the tunnel inflow, a fully developed velocity profile and Reynolds stress profile (for SEM) is applied. The profiles are obtained from a $k - \omega$ SST solution.

• **Viscous Walls:** Walls are treated as viscous and adiabatic.

• **Cyclic:** The solution is periodic in the spanwise direction.

• **Freestream:** The freestream boundary (i.e. parallel to the tunnel floor) is treated with an inlet-outlet type condition. The inlet-outlet boundary acts as velocity inlet and pressure outlet based on the local velocity vector.

A flat plate grid, was also used for the validation cases of the SEM. This grid is identical to the SEM inflow region of the film cooling grid described above. The grid was approximately $[10\delta x 6\delta x 3\delta]$ in the x, y, and z directions respectively. The grid spacing was fixed to $\Delta X^+ = 15$ and $\Delta Z^+ = 8$ in the stream- and spanwise directions. Wall normal spacing was $\Delta Y^+ \leq 1$ at the wall and allowed to stretch to no more than $\Delta Y^+ = 25$ at the edge of the boundary layer. Roughly 14.5 million cells are used for the entire domain. The inlet and outlet boundary conditions match those used in the film cooling simulations.

**Numerical Procedure**

The unsteady LES simulations are initialized in an efficient manner. First, the entire domain is computed using a steady RANS approach. Then, the tube and
duct transition regions (i.e., prior to the crossflow region) are each independently computed using unsteady LES with the divergence-free SEM applied at the inflow boundaries (with steady RANS solution used to initialize the LES). For cases in which SEM is not applied to either the tube or duct inflow boundary, the velocity profile from the steady RANS solution is applied. Once a proper start-up time period has been computed, these intermediate tube and duct transition solutions are mapped back onto the full tube-duct domain, and the steady RANS solution remains intact elsewhere in the domain. It should be noted that to maintain a consistent boundary layer and momentum thickness at the inflow boundaries for cases in which SEM is turned off at the tube or duct inflow boundary, it was necessary to replace the RANS-based inlet velocity profiles with time-averaged inlet velocity profiles generated by the full SEM case (i.e., the baseline case in which SEM is applied to both tube and duct).

Substantial computational effort is required to obtain final turbulence statistics. The flow through time from the jet exit to domain exit is roughly $\tau = 0.01667$ seconds. The solution is computed for $6\tau$ before averaging and time-statistics start to be collected. This allows the solution to reach a quasi steady state, and for transient interactions between the tunnel and jet flows to be propagated and dissipated through the domain. Time statistics and averaging is then conducted over another $40\tau$. This provided sufficient time for statistical quantities to converge. A timestep of $1\times10^{-6}$ seconds ($\Delta \tau = 5\times10^{-5}$) is used for both the initial FTTs and the averaging period.
3. ANALYSIS OF EXPERIMENTAL RESULTS

In this chapter, the analysis of the experimental results from the constant current and constant temperature anemometry measurements will be presented. The effects of numerical compensation of constant current anemometry is also explored.

3.1 Effects of Numerical Temperature Compensation

Due to the well documented (Smits et al., 1978; Childs et al., 2000) difficulties associated with the use of constant current anemometry for unsteady temperature measurements the numerical compensation procedure of Tagawa et al. (2005) was implemented. Initial effort was focused on the evaluation of the sensor diameter effects on unsteady temperature measurements, specifically with the measurement of heat fluxes in mind.

Using the method of Tagawa et al. (2005) described in section 2.1.1, the cutoff frequency for both fine wire thermometers are estimated analytically. Figure 3.1 plots the cutoff frequency (-3 dB) as a function of the local velocity and temperature over a range of velocities and temperatures representative of the conditions observed in the SW-6 facility. The 3.8\(\mu m\) wire is seen to have a cutoff frequency roughly an order of magnitude lower than that of the 1.2\(\mu m\) wire. The kink in the curves is due
to a change in empirical Nusselt number relations to account for Reynolds numbers less than 0.5. Both velocity and temperature have an effect of the cutoff frequency, however the velocity effects are much stronger here. Due to the relatively small temperature range in the experiment, only a 15% change in the density is seen throughout the flow field, limiting its influence on the cutoff frequency.

Figure 3.1: -3 dB cutoff frequency as a function of velocity and temperature for the 1.2µm and 3.8µm CCA wires examined.

In Fig. 3.2 the cutoff frequency estimation is applied to the center plane measurements made by both CCA probes. Due to the strong dependence of cutoff frequency on velocity, the boundary layer and wake recirculation regions exhibit the lowest cutoff frequencies in the flow field, while the jet experiences the highest frequency response. The reduction of frequency response upstream of the leading edge of the injection hole is of minimal impact overall, as the injected flow does not propagate far upstream.
Figure 3.2: Estimation of the -3 dB cutoff frequency along the centerplane of the heated jet in crossflow, for the 1.2μm and 3.8μm CCA wires examined.

In figure 3.3(a) the frequency response of both the 1.2 μm and 3.8 μm wires are examined at the representative flow conditions of \( T = 300K \) and \( U = 10m/s \) using Eq. 2.8. While the finer 1.2μm wire clearly has a higher cutoff frequency compared to the thicker 3.8μm wire, it has a slightly lower frequency response in the low frequency (1-100 Hz) range. This is related to the length to diameter ratio, \( \frac{l}{d_1} \), and by prong end effects. These end effects are studied in greater detail in the works of Tagawa et al. (2005) and Arwatz et al. (2013). In the current work the method of
Tagawa is used to generate the compensation function. Representative samples of the 1.2 µm and 3.8 µm wires compensation functions are shown in Fig. 3.3(b).

![Frequency Response and Compensation Function](image)

(a) Frequency Response  
(b) Compensation Function

Figure 3.3: Transfer function computed from Eq. 2.8 representing the frequency response and compensation function at 10m/s and 300 K for both the 1.2 µm and 3.8 µm wires. -3 dB cutoff frequency labeled for reference.

Sample temperature spectra for both the 1.2µm and 3.8µm wires (taken from the same location, $\vec{X} = (4, 0.15, 0) D$), including the effects of compensation, are shown in Fig. 3.4. The applied compensation function and $-5/3$ spectrum are shown for reference. For both sensors, an inertial and dissipation range is discernible in the uncompensated spectra. The uncompensated inertial range for the 1.2µm wire extends much further than in the 3.8µm spectra. The uncompensated inertial range appears to extend to $\approx 2$kHz in the 1.2µm spectra, whereas in the 3.8µm spectra the dissipation range appears to start well before 0.75kHz. The compensation has a more drastic effect on the spectra of the 3.8µm signal, than it does on the 1.2µm
signal. The slope of the compensated 3.8µm inertial and dissipation ranges are increased significantly over the uncompensated ones. As intended, this delays the effective start of the dissipation range to provide a higher effective cutoff frequency. However, the compensation is unable to delay the start of the the 3.8µm dissipation range to that of the uncompensated 1.2µm spectra. The compensation function applied to the 1.2µm spectrum provides a more subtle effect, slightly changing the slope of the dissipation range. One of the primary differences between the two compensated spectra is seen at the highest frequency content (F > 10kHz). Here the compensated 3.8µm spectra magnitude is elevated higher than the compensated 1.2µm spectra. This significant increase in high frequency content will be clearly seen in the recomputed time signal.
A sample of the compensated time signals are shown in Fig. 3.5, with the original time signal provided for reference. The low pass filtering which is inherent to the CCA technique is clearly visible in the uncompensated signals. As expected, the 3.8 µm uncompensated signal is significantly smoother than the 1.2 µm signal. The compensated signal appears to capture the peak amplitudes also observed in the 1.2 µm signal. The high frequency content added to the 3.8 µm signal far surpasses what is present in either (uncompensated or compensated) 1.2 µm signals. This is to be expected based on the significant increase of high frequency content in the compensated 3.8 µm spectra, as shown in Fig. 3.4(a). The compensated 1.2 µm time signal has slightly elevated peaks, however the increase in higher frequency content is minimal.

The final overall effects of the temperature compensation technique is applied to the full survey for both 1.2 µm and 3.8 µm CCA wires in Fig. 3.6. Before compensation the 3.8 µm wire shows a maximum \( \frac{T_{\text{rms}}}{\Delta T} \) of approximately 17% in the upper shear layer. The uncompensated 1.2 µm wire measures a maximum of 25% \( \frac{T_{\text{rms}}}{\Delta T} \), also in the upper shear layer. The temperature compensation technique raises the \( \frac{T_{\text{rms}}}{\Delta T} \) to 23% and 30% for the 3.8 µm and 1.2 µm wires, respectively. There is a finite limit to the amount of signal attenuation which can be overcome with the present compensation technique. This is highlighted by the compensated 3.8 µm signals under-prediction of the thermal variance compared to the 1.2 µm wire.
Figure 3.5: Sample CCA temperature signal from 3.8\(\mu m\) (left) and 1.2\(\mu m\) (right) CCA measurements with and without compensation.
Figure 3.6: Centerline measurements of normalized temperature r.m.s. fluctuations ($T_{rms}$) with 3.8$\mu$m (top) and 1.2$\mu$m (bottom) CCA wires with and without effects of compensation.
Figure 3.7 shows the percentage increase in r.m.s temperature fluctuations after the compensation technique is applied to both the 3.8µm and 1.2µm measurements.

The 3.8µm wire sees roughly double the temperature variance increase compared to the 1.2µm wire, varying between 20% – 50% and 10% – 25%, respectively. The distribution of $T_{rms}$ increase differs greatly between the two different CCA sensors.

The largest increases in the 1.2µm measurements are seen in the wake region, where as the aft end of the jet core has the strongest amplification in the 3.8µm measurements.

![Diagram of percentage increase in T_rms](image1)

(a) 3.8µm

![Diagram of percentage increase in T_rms](image2)

(b) 1.2µm

Figure 3.7: Percent increase in $\frac{T_{rms}}{\Delta T}$ with 1.2µm and 3.8µm CCA surveys

The compensation technique of Tagawa (2005) applied here, provides an estimated reconstruction of the implicitly filtered (low-pass) high frequency content of the
CCA measurements. While some limitations still exist, it provides a closed form estimate of the physical impact of the implicit filtering seen in CCA measurements, and is therefore well suited for this experimental and numerical work. For the application to heat flux measurements, a method which utilizes the available velocity spectra as a parameter in the temperature compensation would be ideal. From this point forward, the experimental thermal measurements (temperatures and heat fluxes) will be the compensated 1.2\textmu m CCA measurements.

### 3.2 Sensitivity of Anemometry Measurements to Probe Configuration

In this section the thermal anemometry (CCA and CTA) results will be presented. As previously mentioned in Section 2.1, two xT-probes were used to obtain the full array of velocity, temperature, and heat flux measurements. Both the streamwise velocity and temperature are resolved with both the UV- and UW- probes, and provides an initial point of reference for both velocity and heat flux measurements.

The sensitivity of temperature measurements to the orientation of the xT-probes (i.e., the UV and UW probes) is evaluated in Fig. 3.8. Good agreement between the two probes (UV and UW) for both mean normalized temperature and normalized temperature variance. Here the vertical shift in the UW- probe has been accounted for during post-processing. It should be noted that centerline and spanwise measurements are typically taken on different days, and so, the variation between
centerline and spanwise measurements is possibly attributed to day-to-day experimental variations.

The sensitivity of streamwise velocity between the two xT-probes is also presented in Fig. 3.9. The measured freestream normalization velocity, $U_\infty$, has been increased to account for the slightly higher freestream velocities recorded with the UW-probe, while $U_\infty$ remains unchanged for the UV probe. The streamwise velocity agreement is not as good between probes as the temperature. The mean core-jet U-velocity is roughly 13% higher when measured by the UW-probe. There is also a difference in the spanwise distribution mean streamwise velocity profiles. At $X/D = 2.00$ a significant bias is to the positive $Z/D$ direction and is seen with the UV-probe, whereas the UW-probes exhibits a more symmetric profile. This asymmetry is unlikely to be attributed to the flow field, as both temperature measurement sets shows very good symmetry.

In Fig. 3.10 the second velocity component measured by each of the x-T probes is examined. The mean vertical velocity measured by the U-V probe exhibits reasonable symmetry in the measurements. There is a slight shift in the positive $Z/D$ direction, though this is likely related to offsets which occurred in the tunnel set up process rather than probe biasing. The mean spanwise velocity is shown in Fig. 3.10(b). Along the center plane ($Z/D = 0.0$) there appears to be a large spanwise component of the flow, roughly 10% of the freestream velocity. Given the absence of the this crossflow from the cross-cut planes ($X/D = [2,4,6]$), this effect is likely caused by light miss alignments of the probe on the day that dataset was
taken. The vertical and spanwise velocity fluctuations are shown in Figs 3.10(c) and 3.10(d) respectively. The peak in vertical velocity fluctuations are seen downstream of the jet at an axial location starting around $X/D=4.0$, slightly elevated compared to the secondary peak in mean vertical velocity. The secondary peak in mean vertical velocity occurs as upwash resulting from the CRVPs interactions along the centerplane of the jet. The peak in $V_{rms}$ however is related to the flapping and roll-up of shear layer vortices at the tail of the jet. The spanwise fluctuations, $W_{rms}$, are lower in magnitude ranging between 10% to 18%. These levels are similar to the streamwise fluctuations measured by the U-W probe.

Figure 3.11 shows the measured turbulent heat fluxes from both the UV- and UW-probes. Despite the large differences in the U-V and U-W streamwise fluctuations seen earlier, the streamwise heat fluxes agree very well overall. Along the centerplane, very few, minor differences exist. With the cross-cut planes probe biasing is seen with the U-V probe that is not present with the U-W probe. For the U-W probe the vertical axis is plotted as the temperature shifted Y coordinate. With both probes, a region of relatively strong streamwise heat flux is seen in the wake region of the jet. The vertical turbulent heat flux, $\frac{\overline{V^\prime T^\prime}}{U^\prime \Delta T}$, shows similar biasing to the streamwise flux especially off-center of the jet. The strong streamwise and vertical fluxes seen in the top shear layer of the jet indicate a forward leaning heat flux vector in this shear layer region. Along the lower shear layer a negative value of $\frac{\overline{V^\prime T^\prime}}{U^\prime \Delta T}$ is seen, indicating the transport of heat downwards away from the core of the jet. A positive vertical component is seen in the recirculation region just after the
jet exit. This would seem to be counter to the anti-mean temperature gradient typically associated with heat transfer. For the spanwise heat flux taken with the U-W probe, the biasing once again is seen off-center of the jet. The centerplane measurements of the spanwise turbulent heat flux show less asymmetry/offset than the mean spanwise velocity does.
Figure 3.8: Normalized temperature measurements, Mean and variance, with UV- and UW- xT probes at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00
Figure 3.9: Normalized velocity measurements, Mean and variance, with UV- and UW- xT probes at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00
Figure 3.10: Normalized velocity measurements, Mean and variance, with UV- and UW- xT probes at \( Z/D = 0.00 \), \( X/D = 2.00 \), \( X/D = 4.00 \), \( X/D = 6.00 \)
Figure 3.11: Normalized heat fluxes, with UV- and UW- xT probes at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00
The aforementioned probe biasing likely results from bias caused by non-resolvable velocity components influencing the two x-wires unevenly. The UV-probe expects no spanwise velocity, whereas the UW-probe assumes no vertical velocity. Using the mean flow field results from the Full-SEM LES, whose results will be detailed in following chapters, the local flow angles were computed and are shown in Fig. 3.12 to support this argument. The local flow angles can be computed,

\[
\Theta_u = \tan^{-1} \left( \frac{V_{\text{mean}}}{U_{\text{mean}}} \right)
\]

\[
\Theta_w = \tan^{-1} \left( \frac{W_{\text{mean}}}{U_{\text{mean}}} \right)
\]

Along the centerline most of the flow has a relative flow angle upwards of 20°, far from ideal for the UW-probe. Off-center the spanwise flow angle of found to be in the ±15° range in the regions of highest bias for the UV-probe. These unwanted velocity components, and related flow angle changes, appear to greatly increase the uncertainty of the probe measurements in these respective regions. Consideration of the local flow field should be taken when selecting the probe data for comparison. Each probe gives a high confidence in regions where the flow matches the probe’s assumptions. Along the centerline of the jet the U-V probe is preferred, where as the U-W probe is preferred on the outskirts of the jet. Far downstream of the hole, \(X/D > 8\) the flow begins to realign with the axial direction where the conditions are favorable for both probes. The centerline of the jet is a main region of focus for this
research, as such the U-V probe is the primary dataset used for comparison herein, unless noted otherwise.

Figure 3.12: Local flow angles $\Theta_u$ and $\Theta_w$ computed from the Full-SEM LES
4. RESULTS OF NUMERICAL SIMULATIONS

4.1 RANS Simulations

It has been well documented that standard RANS closures are unable to accurately capture the complex nature of the JICF (Muldoon & Acharya, 2000). However, much of the research is focused on the fluid dynamics of the flow rather than the modeling of turbulent thermodynamics and heat transfer which are critical to aerothermal analyses. Initial RANS simulations were conducted herein to evaluate RANS methods ability to predict the physics of the JICF, with particular focus on the turbulent heat flux. For completeness, the full results of the RANS closures will be presented here, along with experimental data for reference. The standard $\kappa - \epsilon$ turbulent closure (Jones & Launder, 1972) is utilized here.

4.1.1 RANS Grid Sensitivity

A sensitivity study on the effect of grid refinement on the standard $\kappa - \epsilon$ turbulence closure was performed. Identical boundary conditions were used at both grid levels, as described in Section 2.2.1. Both computational domains used for the RANS simulations are wall resolved, and contain 35 and 68 million cells in the fine and very fine grid levels, respectively.
Figure 4.1 shows the normalized, streamwise and vertical velocity components along the centerline of the jet at various downstream locations. Both grid levels show nearly identical velocity profiles at the cooling hole exit. In all three locations examined, excellent agreement is seen within the wall bounded region of the jet (i.e. $X/D < 0.5$) for both velocity components. At the downstream locations $X/D \in [4.0, 6.0]$ the very fine grid level shows slightly elevated peak streamwise velocities. To a much lesser extent, this can also be seen in the vertical velocity component.

Figure 4.2 compares the normalized temperature and turbulent kinetic energy profiles. Like the previously examined velocity profiles, the temperature profiles show similar levels of agreement. The film effectiveness (i.e. temperature at the
wall) varies minimally between the two grid levels. The TKE profiles contain larger differences than seen with the velocity or temperature profiles, however they are still relatively modest. The near wall region of the TKE profiles agree very well.

The effect of grid refinement on the modeled turbulent heat fluxes is examined Fig. 4.3. Both streamwise and vertical turbulent heat fluxes show moderate differences between grid levels. The very fine grid again shows elevated levels compared to the fine grid. However, the overall trends are identical. The larger vertical spacing used in the upper regions of the jet with the "fine" grid level likely have a dissipative effect on RANS predicted temperature gradients. Overall, sufficient agreement between the two grid levels is seen to give confidence to the “fine” grid level.
4.1.2 $\kappa - \epsilon$ RANS Comparison to Data

Figure 4.4 shows the comparison between the $\kappa - \epsilon$ RANS simulations with the experimental data taken with the U-V x-T probe. The upper shear layer appears to be well represented in the RANS simulations. However, the RANS closure predictably over estimates the jet penetration, with the $\theta = 0.7$ contour extending to $X/D \approx 5$, compared to $X/D \approx 3.5$ in the $\kappa - \epsilon$ results. The largest region of discrepancy is the wake behind the jet, where the RANS cannot match the jet spreading and reattachment downstream near $X/D \approx 6$.

In Fig. 4.5 the mean velocity is compared for the $\kappa - \epsilon$ RANS simulations and the U-V probe data. Again, the $\kappa - \epsilon$ closure seems to over predict the jet penetration.
At the trailing edge of the cooling hole the RANS predicted velocity profile does not match well with the experimental data. However, this could be related to probe biasing. It should as be noted that the x-wire used to measure the velocity components cannot detect reversed flows. This helps account for the discrepancy in the jet wake behind the cooling hole. The mean vertical velocity is compared in Fig. 4.6. As a whole, the RANS model captures the general trends of the JICF. However, elevated peak jet conditions lead to over penetration of the jet core. The streamwise and wall-normal turbulent heat fluxes are shown in Figs 4.7 and 4.8, respectively. A stark difference is seen in the streamwise turbulent heat fluxes with large differences in both magnitude and direction. In general the RANS modeled heat flux is roughly an order of magnitude lower than the experimental results. The most obvious difference in the directionality of the streamwise flux in the upper shear layer. While the experimental data shows a very slight reversed heat flux on top of the shear layer, the entire modeled heat flux in the upper shear layer is revered (i.e. less than zero). The prediction of the vertical turbulent heat flux does not suffer the same deficiencies as the streamwise flux. The magnitudes of the measured and modeled heat fluxes are of the same order. In general the modeled heat flux does not match with that experimentally measured. The limitations of two-equation RANS models, and the Boussinesq approximation are well documented. The vast majority of two-equation models have an inherent assumption of equilibrium turbulence, which is not guaranteed for separated and shear flows (Wilcox, 2004). The JICF associated with discrete film cooling
Figure 4.4: Normalized mean temperature for $\kappa - \epsilon$ RANS and U-V x-T Probe
Figure 4.5: Normalized mean streamwise velocity for $\kappa - \epsilon$ RANS and U-V x-T Probe
Figure 4.6: Normalized mean vertical velocity for $\kappa - \epsilon$ RANS and U-V x-T Probe
Figure 4.7: Normalized streamwise turbulent heat flux, $\frac{u'T}{\bar{u}_x \Delta T}$, for $\kappa - \epsilon$ RANS and U-V x-T Probe
Figure 4.8: Normalized vertical turbulent heat flux, $\frac{\nu T}{U_\infty \Delta T}$, for $\kappa - \epsilon$ RANS and U-V x-T Probe
configurations is comprised of both of these flow types. This therefore poses significant challenges for two-equation turbulence closures applied to the JICF problem. The RANS simulations presented here aim to provide a initial look at the general limitation of the Boussinesq and gradient diffusion hypotheses. Combined with the accompanying Scale Resolving Simulations (SRSs) to be presented in the following sections, a fundamental understanding into the underlying flow physics can be gained (see Ch. 5). These insights will provide future model developers a starting point to develop new turbulent closures.

4.2 Detached Eddy Simulations

The Detached Eddy Simulation (DES) of Spalart (1997) was the first SRS approach applied to the film cooling configuration described in Section 2.1. It was chosen to match the “detached” nature of film cooling JICFs at the moderate blowing ratio (i.e. BR=1.2) examined here. The results of this simulation are reported here with the U-V xT-probe experimental data repeated again for reference.

Examination of the mean temperature contours seen in Fig. 4.9 highlights major deficiencies in the DES results. The leading edge shear layer is very thin with very steep mean temperature gradients. At the trailing edge of the cooling hole minimal mixing is observed with the vast majority of the jet remaining a near constant temperature. The lack of mixing causes the peak jet temperatures to penetrate far further downstream than seen in the experimental measurements. Additionally, at
the X/D = 4.0 location, the peak jet temperature extends far further circumferentially than suggested by the experimental measurements. The “uncooled” region (i.e. $\Theta = 0$) seen in the wake of the jet is massively over predicted by the DES compared to the experimental measurements. It is likely that without mixing occurring along the initial shear layer of the flow exiting the jet, heat can not be entrained into this wake region. Similar effects are seen in the streamwise velocity contours shown in Fig. 4.10. Again, a lack of mixing in the initial shear layer leads to peak jet velocities sustained further downstream.

The room-mean-squared (rms) flow quantities can be used to examine the amount of mixing which is occurring in the flow. Figure 4.11 shows the normalized temperature fluctuations from the DES. The lack of fluctuations along the entire leading edge of the jet shows an extreme delay in mixing is occurring. The peak fluctuation levels occur near the tail end of the jet core and reach levels of 45%. This is due to the binary like (i.e. hot-cold) mixing which is driven by the flapping of the jet. In the wake of the jet a large region of steady temperature (i.e. $T_{rms} \approx 0$) flow is seen. While mixing is occurring along the bottom shear layer, the flow which is entrained around the jet is unaffected by the jet shear layer and remains at the freestream temperature. The streamwise velocity fluctuations shown in Fig. 4.12 show the same deficiencies along the entire leading edge of the jet that are seen in the temperature fluctuations. This reinforces the idea that the initial shear layer and interaction region is behaving in a completely steady fashion.
Turbulent viscosity will dampen fluctuations in the flow. The turbulent (eddy) viscosity ratio, $\frac{\mu_t}{\mu}$, can be used to ensure the steady behavior of the shear layer is not due to a delayed switching from the RANS to LES models. Contours of the eddy viscosity are shown in Fig. 4.13. The turbulent viscosity contours indicate that the RANS contribution to the solution are fully switched off by the leading edge of the cooling hole. In the tunnel the RANS contributions are switched off approximately two diameters upstream of the hole leading edge, while the cooling tube sees the turbulent viscosity shut off roughly a half diameter before the cooling hole exit. Therefore, the steady behavior of the windward side of the jet is not directly damped by the presence of turbulent viscosity from the RANS model. DES models in general lack a method of converting the modeled turbulence, which is represented through the turbulent viscosity $\mu_t$, into the physical structures needed for the scale resolving region of the simulation. As a result of these findings it is determined that a full SRS with turbulent inflow treatment is required to appropriately resolve all of the key features of film cooling configurations examined here.
Figure 4.9: Normalized mean temperature for SA-DES and U-V x-T Probe. Slices at $Z/D = 0.0$ $X/D = [2,4,6]$
Figure 4.10: Normalized mean streamwise velocity for SA-DES and U-V x-T Probe. Slices at Z/D = 0.0 X/D = [2,4,6]
Figure 4.11: Normalized mean temperature fluctuations for SA-DES and U-V x-T Probe. Slices at $Z/D = 0.0$ $X/D = [2,4,6]$
Figure 4.12: Normalized mean streamwise velocity fluctuations for SA-DES and U-V x-T Probe. Slices at $Z/D = 0.0$ $X/D = [2,4,6]$
Figure 4.13: Turbulent viscosity ratio, $\mu_t/\mu$, for the SA-DES case
4.3 Large Eddy Simulations with Synthetic Turbulence

The effect of physical, resolved turbulence in LES simulations of the film cooling problem was studied with a matrix of test cases. The cases were composed of: Full-SEM, Tube-SEM, and Tunnel-SEM. The Full-SEM case had synthetic turbulence generated at both the tunnel and tube inflow boundaries, while the Tube-SEM and Tunnel-SEM only generated inflow turbulence at the tube and tunnel boundaries, respectively.

4.3.1 Validation of Numerical Model

A validation of the SEM was performed on a flat plate case with a similar Reynolds number based on momentum thickness \((Re_\theta)\) to the film cooling case of interest. A 2D RANS simulation \((k-\omega\text{ SST})\) was used to obtain inputs for the SEM boundary. Mean velocity and isotropic Reynolds stress profiles at \(Re_\theta \approx 1300\) and a freestream velocity of 9.14 m/s were mapped to the SEM inlet boundary. Figure 4.15 shows the span-averaged friction coefficient development downstream. Figure 4.14 shows Q-criterion colored by the normalized axial Reynolds stress. The initial freestream turbulence provided from the RANS profile quickly dissipates within the first three boundary layer thicknesses downstream of the inlet. The small structures associated with the inlet synthetic turbulence start to form larger structures roughly half way through the domain. Figure 4.14 shows the axial stress profile initially growing before normalizing around \(X/\delta = 6\). This is due to the transition process required to
transform the synthetic unnatural turbulence at the inflow to the realistic structures seen downstream. This transition process is sensitive to the number and size of eddies used to populate the inflow domain. The range of $C_f \in [0.0032 - 0.0033]$ after $Re_\theta = 1500$ is approximately 14% lower than that predicted by DNS from Schlatter (2009).

Turbulent boundary layers can be characterized through the use of several thickness parameters. The most basic of these parameters is the boundary layer thickness, $\delta$, which is simply the height off the wall where the velocity reaches 99% of the freestream value. This is sometime referred as the 99% boundary layer thickness, $\delta_{99}$. The next key parameter is the displacement thickness, $\delta^*$, which represents the offset which would be required for an inviscid flow (free-slip) to have the same mass flow rate as the real boundary layer. The displacement thickness is integrated through the boundary layer,

$$\delta^* = \int_0^\infty \left(1 - \frac{\rho(y)u(y)}{\rho_0u_0}\right) dy \quad (4.1)$$

Similarly, the momentum thickness, $\theta$, can be defined as how much a surface must be offset for an inviscid layer to contain the same momentum as the real viscous boundary layer. Again integrating through the boundary layer,

$$\Theta = \int_0^\infty \frac{\rho(y)u(y)}{\rho_0u_0} \left(1 - \frac{\rho(y)u(y)}{\rho_0u_0}\right) dy \quad (4.2)$$
Figure 4.14: Iso-contour of Q-Criterion (Q=50,000) colored by the normalized axial Reynolds stress.

With these thicknesses the shape factor, $H$, is defined as:

$$H = \frac{\delta^*}{\theta} \quad (4.3)$$

The $C_f$ derived from the LES results agree with empirical relations from White (1974) using a shape factor of $H = \frac{\delta^*}{\delta_{\theta}} = 1.44$. Note that a shape factor of $H \in [1.3 - 1.4]$ is typical for turbulent flow over a flat plate with zero pressure gradient.

Figure 4.15: Span averaged friction coefficient as a function of momentum thickness Reynolds number, $Re_{\theta}$
In Fig. 4.16 the Reynolds stress profile at various downstream locations of the SEM inlet are compared to the corresponding profiles from DNS. The local momentum thickness Reynolds number is provided for reference at each location. The axial stress is reasonably well represented at the $X_\delta = 8.0$ location. The first two stations ($X_\delta = 2.5, 5.3$) however, significantly underpredict the axial stress. This is related to the transition process described above. All locations underpredict the spanwise and wall-normal stresses, $\overline{w'w'}$ and $\overline{v'v'}$ respectively. The shear stress remains roughly constant across the range of Reynolds numbers seen here. Across all of the stresses, the outer wake region ($y^+ \geq 150$) is thinner than the DNS data suggests. Moderate grid stretching could be a factor in the wake region discrepancy, although is not considered to be a major factor here due to the grid resolution used. Based on these acceptable results, this SEM flat plate case was also used to refine SEM model.
settings, set the required grid fineness for SEM transition, and obtain an initial duct transition solution for the film cooling simulations.

4.3.2 Effects of Synthetic Turbulence on Inflow

The effect of physical, resolved turbulence in LES simulations of the film cooling problem was studied with a matrix of test cases. The cases were composed of: Full-SEM, Tube-SEM, and Tunnel-SEM. The Full-SEM case had synthetic turbulence generated at both the tunnel and tube inflow boundaries, while the Tube-SEM and Tunnel-SEM only generated inflow turbulence at the tube and tunnel boundaries respectively.

It should be noted that a no-SEM “very fine” grid level simulation as well as a “Coarse” grid level Tube-SEM simulation were also performed. Figure 4.17 shows the temperature fluctuation levels for both the No-SEM simulation as well as the coarse tube-SEM simulations. The very fine tube-SEM simulation and experimental results are provided for reference. In the no-SEM and coarse grid simulations substantial limitations were experienced, as described below. As a result of these limitations, the full data sets of these additional cases will not be reported in full herein. While matching the general trend of the finer tube-SEM case and experiment, the coarse grid simulation shown in Fig.4.17(a) shows modest differences from the finer grid level tube-SEM simulation seen in Fig. 4.17(c). The leading shear layer is shown to be thicker than in the “very fine” grid level.
Additionally, the temperature fluctuations in the near wall wake region are reduced in the coarse grid simulation. Both of these are to be expected as the coarser grid resolution will filter more of the smaller scale energy content from the simulation, locking more energy into the larger scale structures.

The “very fine” no-SEM simulation shows dramatic differences from both the experiment and the tube-SEM results. Even after an initial transition period twice that of the other SEM cases, as well as an averaging period which was 50% longer, the no-SEM simulation was still unable to reach a quasi steady state. The solution oscillates between a laminar like shear layer, similar to that seen in previous DES results, and a K-H instability state. This oscillation has led to the significant increase in peak temperature r.m.s seen in the no-SEM simulation. This supports the current research effort in providing synthetic turbulence at the domain inflows in SRSs.
Figure 4.17: Root-mean-squared temperature fluctuations normalized by temperature difference at $Z/D = 0.00$, $X/D = [2,4,6]$ for a coarse tube-SEM simulation, and a very fine no-SEM simulation. Very fine tube-SEM and U-V probe experimental data provided for reference.
Figure 4.18 depicts the normalized mean temperature contours for the SEM sensitivity cases as well as UV-probe CCA data (lower right image). The differences between the three SEM treatment cases are subtle. The full-SEM simulation shows the highest film effectiveness ($\Theta = \frac{T - T_\infty}{\Delta T}$) with the $\Theta = 0.2$ contour reaching the wall just after the $X/D= 4.00$ location. The tube-SEM and tunnel-SEM cases also show reattachment with the $\Theta = 0.2$ contour reaching the wall at around 6 and 7 diameters downstream, respectively. The tunnel-SEM case also exhibits a a slightly more noticeable cold-spot after the jet exit. The shear layer growth of both the full- and tube-SEM cases show excellent agreement, while the tunnel-SEM shear layer’s growth is delayed until $X/D = 1.00$.

Figure 4.19 depicts normalized temperature variance contours. The full- and tube-SEM simulations are very similar with only subtle differences, such as the slightly longer extent of the underside portion of the wake. The distribution of thermal variance in the tunnel-SEM configuration is significantly different from the other SEM cases. Due to the reduced shear layer growth, temperature fluctuations of up to 40% are seen in the shear layer until $X/D = 3.00$. Additionally, both the extent and magnitude of the underside wake is increased in comparison to the other SEM cases. The circumferential extent of $T_{rms}/\Delta T$ and radial gradients are also larger in the tunnel-SEM simulation. Without the turbulence present within the tube to enhance eddy transport approaching the mixing region, the resulting mixing of the hot and cold gases is less vigorous, leading to higher temperature gradients and variance.
Figures 4.20 and 4.21 contain mean streamwise and vertical velocity contours, respectively. Again, the full- and tube-SEM simulations share many similarities with subtle differences. The jet exit velocity for the tunnel-SEM case is elevated compared to the simulations which include tube turbulence. The applied velocity boundary condition profile in the tunnel-SEM is a time-average of the full-SEM tube three diameters upstream of the hole exit. The presence of turbulence within the tube causes the velocity profile to widen and become flatter, leading to the lower peak velocities in the full- and tube-SEM cases. Similarly, vertical velocity profiles for the tunnel-SEM penetrate further into the crossflow and exhibit a larger secondary up-wash.

Fig. 4.22 illustrates the root-mean-squared streamwise velocity contours. In the LES simulations, a region of strong streamwise velocity fluctuation extends from the trailing edge of the hole to the end of the jet penetration ($X/D \approx 4.5$). However, the hot wire data contains a region of less strong fluctuations which appears to stem more from the upper shear layer and the hole leading edge. The larger probe height ($\approx 0.25D$) and fairly coarse probe resolution could lead to smaller flow features, like the strong fluctuations noted in the LES, to become vertically smeared. It should be noted that the vertical thickness of the underside shear layer is typically smaller than the vertical probe length. At $X/D = 4.00$ the peak $U_{rms}/U_\infty$ region, from the full-SEM simulation, extends a vertical distance of roughly $0.12D$, or half of the probe length.
Fig. 4.23 shows the most significant difference amongst the three SEM cases. While the full- and tube-SEM simulations again show only minor differences, the tunnel-SEM simulation contains significantly higher vertical fluctuations at the end of the jet penetration ($X/D \approx 4.5$). This region roughly correlates with the secondary up-wash seen in the mean vertical velocity. The presence of tube turbulence significantly breaks down the strong coherent K-H type structures into smaller more three-dimensional structures, redistributing the turbulent energy elsewhere. The UV probe shows slightly weaker peak $V_{rms}/U_{\infty}$ magnitudes and the extent of the peak region is slightly more limited. The probe volume and resolution could impact the results here; however the larger size of the feature of interest suggests this may not be the case. The local flow angle, as shown in Fig. 3.12(a), suggests that the local flow conditions are on the edge of the acceptable flow angles for the UV probe. Spanwise velocity fluctuations are plotted in Fig. 4.24. Similar trends are seen here as with the other velocity and temperature fluctuations. The initial shear layer growth is limited in the tunnel-SEM simulations compared to the other LES simulations, and a slightly more energetic underside wake region is produced.

Figure 4.25 is related to the streamwise turbulent heat flux. It contains the product of $U_{rms}$ and $T_{rms}$, which gives an estimate of the heat flux without the effect of signal correlation. Very good agreement is seen between experiment and simulation. As root-mean-squared velocity and temperature contours are both in good agreement with experimental profiles it follows that the uncorrelated heat flux
should also be in good agreement. The primary disagreement between LES and experimental comes from the underside shear layer region, where the LES contains elevated $U_{rms}$ profiles. This is a carry-over effect from the $U_{rms}$ profiles.

The sensitivity of Large Eddy Simulations of film cooling JICF’s to the turbulent inflow treatment was examined. The tunnel crossflow and cooling tube inlet boundary conditions are toggled between a steady inlet and an inlet with synthetic generated via the DF-SEM. Modest differences are seen among the different SEM inflow treatments. The most significant differences are seen in the tunnel-SEM configuration. This suggests the near field characteristics are more strongly dependent on the conditions within the tube. In the experimental literature there is a wide range of reported turbulent quantities for similar flow conditions. Peak values range from as low as approximately 16% (Thole, Gritsch, Schulz, & Wittig, 1998) to as high as nearly 40% in some works (Wright et al., 2011b).

Unfortunately, as is the case in the present experimental work, it is often difficult to measure the conditions within the cooling tube. Optical access is often unavailable, or the presence of the probe would cause significant blockage effects. As a result the tube conditions remain a significant hurdle for CFD validation datasets. For the present study, care was taken in the experiment to provide a fully developed pipe flow to reduce some of the experimental uncertainty.
Figure 4.18: Normalized mean temperature profiles from LES and U-V x-y wire probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00.
Figure 4.19: Root-mean-squared temperature fluctuations normalized by temperature difference at $Z/D = 0.00$, $X/D = 2.00$, $X/D = 4.00$, $X/D = 6.00$. 

(a) Full-SEM
(b) Tunnel-SEM
(c) Tube-SEM
(d) U-V Probe
Figure 4.20: Mean streamwise velocity contours from LES and U-V xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00.

(a) Full-SEM
(b) Tunnel-SEM
(c) Tube-SEM
(d) U-V Probe
Figure 4.21: Mean vertical velocity contours from LES and U-V xT-wire Probe at $Z/D = 0.00$, $X/D = 2.00$, $X/D = 4.00$, $X/D = 6.00$
Figure 4.22: Root-mean-squared streamwise velocity fluctuations contours from LES and U-V xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00.
Figure 4.23: Root-mean-squared vertical velocity fluctuations contours from LES and U-V xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00.
Figure 4.24: Root-mean-squared spanwise velocity fluctuations contours from LES and U-W x-T-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00.
Figure 4.25: Uncorrelated streamwise heat flux ($U_{rms} T_{rms} / U T$) contours from LES and U-V xT-wire Probe at $Z/D = 0.00$, $X/D = 2.00$, $X/D = 4.00$, $X/D = 6.00$.
5. COMPARISON OF EXPERIMENTAL AND NUMERICAL RESULTS

In the previous chapter, the sensitivity of the LES model to the input of synthetic turbulence was studied. In this chapter, the full-SEM simulation results are compared to the UV-probe measurements, and the gradient diffusion hypothesis (GDH) is evaluated.

5.1 UV-Probe and Full-SEM

Figure 5.1 compares the Full-SEM with the experimental data taken with the UV-probe. Excellent agreement is seen between the Full-SEM and hot-wire for mean temperature measurements (upper left image), with overall thickness and gradients being well represented. Good agreement is also obtained for thermal standard deviation (lower left image), with peak values slightly over predicted by the LES. Temperature measurements made at the trailing edge of the jet exit (X/D = 2.00) suggest a modest thermal boundary layer exists within the jet, based on the small reduction of mean temperature and the presence of a small thermal variance near the wall (Y/D = 0.00). This suggests the adiabatic walls used in the LES are not strictly correct. However, the nonadiabaticity of the tube has little impact elsewhere.
in the flow domain. The over prediction could be exaggerated slightly by the lower resolution of the CCA data not fully resolving peaks in the profile close to the wall.

Figure 5.1: Full-SEM LES (solid line) compared with CCA (temperature) and CTA (velocity) data taken with the UV-probe (symbols)

Figure 5.1 also shows reasonable agreement is obtained for mean and root-mean-squared streamline velocity (see middle images). The peak streamwise velocity at the hole exit is elevated slightly compared to the CTA measurements. Streamwise velocity fluctuations agree well at the jet exit as well as downstream at \(X/D = 6.00\). In the mid region (\(X/D = 4.00\)), the magnitude matches well; however, LES results show a series of peaks which are not resolved by the CTA data. Since the x-wires of the UV-probe are oriented in the vertical direction, this orientation stretches the probe volume vertically and can cause smaller flow features
to be smeared. This is especially true when gradients are strong in the stretched
direction, as is the case here. Finer measurement resolution as well as post
processing techniques could mitigate this resolution issue. The mean vertical
velocity profiles show good qualitative agreement between CTA data and LES, with
LES peak velocities slightly elevated near the jet exit.

Figure 5.2 depicts the normalized mean and fluctuating temperature contours for
the Full-SEM LES case as well as UV-probe CCA data. In Figures 5.2(a) and 5.2(b)
normalized mean temperature contours show excellent agreement between LES and
experiment. The leading edge shear layers grow at similar rates for both the LES
and experiment. The recirculation region directly behind the hole exit appear to be
similar in size and extent as does the overall jet penetration. Below in figures 5.2(c)
and 5.2(d) room-mean-squared fluctuations of temperature are shown. Similar
excellent agreement is seen with the thermal fluctuations as with the mean
temperatures. The distribution of thermal variance in the recirculating wake region
under the jet is well represented in the LES simulation. Although the experimental
results along the center line of the jet show slightly lower $T_{rms}$ values than the LES,
this is likely caused by the experimental resolution missing the peaks, this is
supported by similar fluctuation values seen in the cross cut planes in both the LES
and experimental data.
Figure 5.2: Normalized Mean (top) and fluctuating (bottom) temperature profiles from LES (left) and U-V xT-wire probe (right) at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00
Figure 5.3 contains mean and fluctuating streamwise velocity contours. Mean streamwise velocity contours in Figures 5.3(a) and 5.3(b) continue to show good agreement between the experimental U-V probe and full-SEM simulation. The hot-wire measurements are not able to indicate reversed flow in the streamwise direction. The experimental data also exhibits a moderate amount of spanwise biasing. This is due to both the physical separation of the two "X" wires used for the two-component velocity measurements, as well as the spanwise component of the jet velocity which exists largely off center of the jet. The lack of asymmetry present in the temperature contours of Fig. 5.2 support the idea that no physical flow bias exists but is purely measurement bias. The streamwise velocity fluctuations shown in Figures 5.3(c) and 5.3(d) show slight differences in stress distribution. The LES results show a strong streamwise normal-stress emanating from the trailing edge of the jet extending along the lower shear layer towards the end of the jet core. The experimental measurements show similar peak $U_{rms}$ values, however they are shifted towards the upper shear layer, with a lack of a distinct lower shear layer. The thicker region of $U_{rms}$ at the tip of the jet core is potentially stretched vertically by the "X-wires" configuration and orientation.
Figure 5.3: Mean (top) and fluctuating (bottom) streamwise velocity contours from LES (left) and U-V xT-wire Probe (right) at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00.
Figure 5.4: Mean (top) and fluctuating (bottom) vertical velocity contours from LES (left) and U-V xT-wire Probe (right) at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00.
The mean vertical velocity contours are shown in figures 5.4(a) and 5.4(b). The experimental vertical velocity measurements show less spanwise biasing compared to the streamwise velocity component. This difference is likely related to the law of cosines averaging used to extract individual velocity components from the X-wires. Again, excellent agreement is seen among the LES and experimental results. The secondary upwash region (between $X/D = 3.00$ and $X/D = 5.00$), which results from the CRVP interactions are well represented in both datasets. Vertical velocity fluctuations, shown in figures 5.4(c) and 5.4(d), continue the good agreement between LES and experiment. The overall structure of the vertical stress field is well represented with the peak values existing just downstream of the jet core. The extent of the maximum fluctuations is slightly taller in the LES than it is in the experiment, though the overall trend matches very well. The experimental contours show a core of lower turbulence at the jet exit. Reexamination of Fig. 5.1 shows the actual difference in LES and experiment to be relatively small.

The turbulent heat flux vector, $\langle u'T', v'T', w'T' \rangle$, provides the direction and magnitude which turbulence transports heat within the flow field. Figure 5.5 contains the contour flood of streamwise turbulent heat flux contours, $\overline{u'T'}/U_\infty \Delta T$ and contour lines of mean temperature, for both LES and experiment. In both the experimental and LES results the heat flux contours transport heat away from the core of the jet as expected (that is against the mean temperature gradient). Recall, here the temperature gradients have been reversed from the normal film cooling configuration. Downstream around $X/D > 4.5$ a strong streamwise component of
the heat flux dominates the vector transporting the heat further downstream. The experimental heat flux results display an asymmetry not seen in the LES simulations. The cause of this asymmetry is related to the spanwise offset of the temperature measurement in the UV-Probe used here. In the upper shear layer there is a thin region of reversed heat flux (i.e. transport of heat in the upstream direction) in the LES results. This region of reversed heat transfer can also be seen in the experimental data; however, its presence is muted due to the larger probe volume of the U-V probe. This reversed heat transfer corresponds to a reduction of streamwise velocity with a corresponding increase in temperature. At first this may appear counter intuitive given the elevated velocity and temperature of the jet, however near edge of the shear layer there is a thin region where the axial component of velocity is reduced helping account for the reversed direction of heat transfer.

The vertical component of the turbulent heat flux vector is shown by the contour floods in Fig. 5.6, while the contour lines represent the average temperature. Good agreement is seen between the experimental U-V probe measurements and the full-SEM LES results. Both the experimental data as well as the simulation agree on the direction of the vertical heat flux in most regions of the flow. The upper shear layer results in heat transported away from the heated jet, while the lower shear layer transport heat downwards towards the wake. Along the centerline of the wake both the experiment and simulation show a vertical transport of heat ($X/D = 3.5$ to $X/D = 5$). The peak values of heat are slightly elevated in the numerical
results compared to the measured heat fluxes. The experimental results also show an asymmetry which is not seen in the numerical results. This asymmetry is due to the nature of the xT-probe and will be discussed in more detail in the next section. The split between positive (upwards) and negative (downwards) follows the trajectory of the mean temperature contours in both the numerical simulations as well as the experimental measurements.

The spanwise turbulent heat flux is given by contour floods in Fig. 5.7, with contour lines showing mean temperature. The spanwise heat flux goes to zero along the center plane of the jet. It should be noted that the experimental measurements indicate peak heat fluxes along the centerline of the jet which are roughly 4 times smaller than the numerics suggest. This is unsurprising given the previously described limitations with the U-W probe in the near wake region of the jet (especially along the centerline). With these limitations in mind, the experimental measurements support the trend observed in the LES results. At the $X/D = 4.0$ location two primary flow features are seen. Heat is transported outwards away from the jet-core in the outer region of the jet. In the inner wake region ($Y/D = 0.5$ and $Z/D = \pm0.2$) the turbulent heat flux is directed towards the centerline of the jet. This inward heat transport corresponds with the region of positive (upward) heat flux as shown in the wake behind the jet in Fig. 5.6.
Figure 5.5: Streamwise turbulent heat fluxes from LES and U-V xT-wire Probe at \( Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00 \)
Figure 5.6: Vertical turbulent heat fluxes from LES and U-V xT-wire Probe at Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00
Figure 5.7: Spanwise turbulent heat fluxes from LES and U-V xT-wire Probe at \( Z/D = 0.00, X/D = 2.00, X/D = 4.00, X/D = 6.00 \). Note the difference in scale.
5.2 Two-Point Heat Flux Estimations

In the previous section, the experimental heat flux measurements were shown to be asymmetric in nature. This is caused by the larger probe volume combined with the orientation of the xT-probe. To support the explanation of heat flux asymmetry a two-point heat flux study was conducted utilizing the full-SEM LES results. Figure 5.8 shows a schematic depiction of the sensor wires configuration from the U-V probe. The velocity-temperature offset, $\delta_{v-t}$, is on the order of 10\% of the cooling hole diameter (i.e. $\delta_{v-t} \approx 0.1D$). To examine the effects this offset has on heat flux measurements the full-SEM LES results were reprocessed accounting for the velocity-temperature offset. At the $X/D = 3.0$ location the computational heat flux was modified by increasing the temperature offset incrementally up to a $\delta_{v-t} = 0.1D$ in the positive $Z/D$ direction, while leaving the velocity measurement at the original location.

The results of the velocity-temperature offset study are shown in Fig. 5.9. In the top left of Fig. 5.9 the experimental measurements are presented, next to that in the top right the full resolution streamwise heat flux from the LES simulation is shown. The lower four quadrants highlight the effects of velocity - temperature offsets in heat flux measurements. The middle left shows the LES result down sampled to the experimental resolution. This down sampling causes reductions in the peak heat flux levels observed, as well as a stretching of some flow features such as the lower shear layer. The middle right of the figure shows a 5\% diameter offset,
while the bottom of the figure displays the 7.5% and 10% diameter offsets. The inclusion of the temperature offset drastically changes the distribution of the heat flux. The large asymmetry in the primary shear layer is well reproduced with the peak heat flux now seen at $+Z/D = 0.5$. The lower shear layer, a dominate feature in the full resolution LES, completely disappears from the offset result.

With the significance of the velocity-temperature offset now understood, the entire full-SEM LES results can be recomputed accounting for a 7.5% diameter temperature offset. The recomputed streamwise and vertical heat fluxes can be seen in Figs. 5.10 and 5.11, respectively. The full resolution LES derived heat fluxes are included for reference. In Fig. 5.10 excellent agreement is seen with the experimental and offset enabled streamwise heat fluxes. The lower shear layer, which dominates the full resolution LES results, is significantly weakened along the
Figure 5.9: Results of two-point heat flux study. Streamwise heat-fluxes at $X/D = 3.00$ (top left) Full-resolution LES, (top right) UV-Probe. The lower four images show the effects caused by increase probe offset. Black squares represent the experimental probe resolution.

centerline of the jet. In the longitudinal cuts (i.e. $X = \text{constant}$) the asymmetries seen in the experimental data set are closely reproduced at all downstream locations by the offset enabled LES results.

The excellent agreement is also seen in the vertical heat fluxes shown in Fig. 5.11.

A clear and well defined peak in vertical heat flux exists along the upper shear layer
of the full resolution LES results which is missing from the experimental data. The inclusion of the temperature offset significantly reduces the strength of the heat flux in the upper shear layer, closely matching the experimental data. The excellent agreement between the experimental measurements and two-point LES results is continued in the longitudinal cuts of vertical turbulent heat flux.
Figure 5.10: Streamwise heat flux with effects of spanwise velocity-temperature offset included. Temperature offset of $\delta_{c-t} = 0.075D$. 
Figure 5.11: Vertical heat flux with effects of spanwise velocity-temperature offset included. Temperature offset of $\delta_{v-t} = 0.075D$. 

(a) $\frac{\nu'T'}{U_\infty \Delta T}$: Full-SEM

(b) $\frac{\nu'T'}{U_\infty \Delta T}$: Two-Point LES

(c) $\frac{\nu'T'}{U_\infty \Delta T}$: U-V Probe
The inclusion of the spanwise temperature offset seen in the U-V xT-probe has accounted for the asymmetries seen in the streamwise and vertical turbulent heat flux measurements. The ability to reproduce these asymmetries from the full resolution LES results further extends the confidence in the accuracy of the present full-SEM simulations. These effects also highlight the need to fully understand the abilities and limitations associated with any experimental or numerical technique. Furthermore, these results highlight the ability for numerical analysis to complement experimental findings. The use of LES in this case was critical as a RANS closure would be unable to recreate the experimental results as demonstrated here. Future research should focus on correction techniques to overcome these experimental biases. Currently, the combination of CCA and CTA measurements remains one of the few ways to directly measure the turbulent heat fluxes.
5.3 Gradient Diffusion Hypothesis Evaluated

To examine the validity of the GDH used in conventional RANS simulations the magnitude and directionality of the modeled (RANS) and resolved (LES) heat fluxes need to be examined. In Fig. 5.12 the streamwise heat flux contours are shown with contour lines of mean temperature for both the LES results and for a conventional RANS ($\kappa - \epsilon$) solution. Significant differences exist between the LES resolved heat fluxes and the GDH modeled fluxes from RANS. The RANS streamwise heat fluxes are roughly an order of magnitude smaller than what is resolved in the LES simulation. In the RANS GDH model the leading shear layer is entirely in the minus X direction (i.e. $u'$ and $T'$ are negatively correlated). This differs greatly from the LES resolved results which are predominately in the positive direction, except for a thin region on the edge of the shear layer. Additionally, the RANS GDH model does not capture the elevated streamwise flux in the wake of the jet.

The vertical turbulent heat fluxes from the GDH model and the full-SEM LES are compared in Fig. 5.13. The modeled vertical heat flux does a reasonable job of matching the signs of the resolved LES fluxes. The GDH correctly predicts that heat will be transported away from the the heated jet, with a positive (upwards) heat flux in the upper shear layer and a negative (downwards) flux in the lower shear layer. The upwards heat flux in the wake of the jet shown in the LES results is not captured by the GDH model. The magnitude of the modeled vertical heat flux is roughly a factor of three or four smaller than the resolved flux from the LES.
At the trailing edge of the hole the small region off center of the jet with positive (upward) heat fluxes are not replicated by the GDH.

The directionality of the heat flux is further highlighted in figures 5.14 and 5.15 by plotting streamlines defined by the turbulent heat flux vector over mean temperature contours. Per the GDH, in the RANS simulation the turbulent heat flux vector is against the mean temperature gradient \((-\frac{\partial T}{\partial x_j})\). The LES heat flux is the directly computed heat flux. In both approaches a similar heat flux vector is formed along the mean jet path where the turbulent motion transports the heat downstream. Above this branch moderate differences may be observed with the LES heat fluxes angled slightly towards the streamwise direction. This forward orientation agrees well with the recent work of DeBonis in heated free jets (DeBonis, 2018). However, the underside of the jet exhibits significant difference. In the LES solution, a slightly forward leaning heat flux is seen between \(Y/D = [0.5, 1.0]\) but below \(Y/D = 0.5\) a strong streamwise orientation is seen. There are also regions where the heat flux is in the direction of mean temperature gradient contrary to the GDH’s definition. This appears to be caused by heat from the jet being turbulently conducted into the region of cooler temperatures behind the jet \((X/D \in [2, 4])\), before quickly being convected vertically and downstream by cool air which is entrained from the sides of the jet.

Contours of mean temperature with heat lines overlaid at a constant wall normal location of \(Y/D = 0.5\) are shown in Fig. 5.15. Here the peak temperature extends further downstream on the edge of the jet (near \(Z/D \approx \pm0.5\)). In a similar fashion
to the centerline plane two main branches form to transport heat downstream along the peak in the temperature contours. Both the GDH and LES results show these branches, however the RANS solution shows these branches to be parallel where the LES results show the branches pinching closer together around $X/D = 5.0$. The forward inclination of the heat flux vectors is observed both inside and out of these spanwise branches in the LES results. The GDH based RANS simulation is unable to resolve this axial component of the heat flux.

Recall from equation (1.34), that the turbulent heat flux term appears within a gradient operator. DeBonis (DeBonis, 2018) found that accounting for the unresolved axial component of the turbulent heat flux did little to address the concerns in the RANS simulations. This was due to the axial component of the heat flux gradient being of significantly small magnitude compared to the other components. By examining the relative magnitude of the heat flux gradient terms, which appear in the energy equation, the significance of each of the heat flux components can be evaluated. This comparison is seen in Fig. 5.16, where the streamwise component of the heat flux gradient is compared to the vertical and spanwise components, for both the full-SEM LES and GDH based RANS results. The heat-lines are included in Fig. 5.16 for reference as well. In the wake region behind the jet, the axial component of the heat flux gradient is a significant contribution to the energy equation in the LES results. The RANS results show near zero contribution to the energy equation for the wake region. At the $Y/D = 0.5$ plane, the streamwise and spanwise components are compared. Again, the LES
results show a significant contribution to the energy equation, where the GDH
based RANS does not. Clearly, the highly three-dimensional nature of the film
cooling JICF flows contain axial gradients which cannot be ignored to achieve
proper modeling.

Similar to the transport of TKE, the transport of the turbulent heat flux occurs due
to turbulent eddy convection, diffusion, production, and dissipation. A significant
limitation of Boussinesq type approximations (momentum or heat) is the
combination of multiple modes of transport into a single lump term. Models which
are not based on the Boussinesq approximation, such as Reynolds Stress Models
(RSM) (Launder, Reece, & Rodi, 1975) or Algebraic Stress Models (ASM) (Gatski
& Speziale, 1993), provide room for future improvements to the modeling of JICF.
Future models development should aim at identifying and defining terms these
different modes of transport for the difference mechanisms of turbulent heat transfer.
Figure 5.12: Streamwise turbulent heat fluxes from LES and $k - \varepsilon$ RANS at $Z/D = 0.00$, $X/D = 2.00$, $X/D = 4.00$, $X/D = 6.00$
Figure 5.13: Vertical turbulent heat fluxes from LES and $k-\epsilon$ RANS at $Z/D = 0.00$, $X/D = 2.00$, $X/D = 4.00$, $X/D = 6.00$
Figure 5.14: Mean temperature contours with streamlines defined by the turbulent heat flux vector at $Z/D = 0.00$, $X/D = 2.00$, $X/D = 4.00$, $X/D = 6.00$
Figure 5.15: Mean temperature contours with streamlines defined by the turbulent heat flux vector at $Z/D = 0.00$, $Y/D = 0.5$
Figure 5.16: Evaluation of the turbulent heat flux vector gradient and its contribution to the energy equation. Heat lines defined by the turbulent heat flux vector included for reference. Slices at $Z/D = 0.00$, $Y/D = 0.5$. 

(a) Heat Lines: Full-SEM

(b) Heat Lines: $k - \epsilon$ RANS
6. CONCLUSIONS

An experimental and numerical investigation into turbulent mixing of a heated jet in a crossflow was conducted at NASA Glenn Research Center, ERB test cell SW-6, and via a set of high-fidelity Scale Resolving Simulations (SRS). The Detached Eddy Simulation (DES) of Spalart (1992) as well as implicit Large Eddy Simulations (LES) were used herein. The Divergence-Free Synthetic Eddy Method (DF-SEM) of Poletto et al. (2013) was used for the inflow boundary conditions in the LES cases. The DES results did not do an acceptable job of matching the experimental measurements. The DES model exhibited a significant delay in the turbulent transition of the shear layer. This is due to the lack of a mechanism to convert the modeled turbulence to physical coherent structures needed in the LES regions. As a result the DES methodology was determined to be inappropriate for the problem at hand.

Regarding the implicit LES, the addition of coherent turbulent structures via SEM treatment effectively eliminated the transition delay observed in the DES method. Excellent agreement is obtained for mean and fluctuating temperature contours in both magnitude and distribution throughout the flow field. Very good agreement
between the LES and experiment is also obtained for mean and fluctuating velocity components in regions of probe validity, particularly along the centerplane of the jet.

The sensitivity of the LES to inflow boundary conditions was examined through a set of cases where the SEM boundary treatment was toggled on and off. While the full-SEM (turbulent inflow on both the jet and crossflow) showed the best agreement with the experiment, the differences between all of the configurations were modest. The conditions within the cooling tube are critical to accurate prediction of the Jet in a Crossflow (JICF) flow field. Unfortunately, it is difficult to unobtrusively experimentally measure the tube flow conditions.

There are a number of additional challenges associated with the measurement of turbulent heat fluxes within the jet in crossflow flow-field. The effect of the thin-wire low cutoff frequency, and its associated loss of statistical variance, must be addressed. Here this was done through the use of a numerical compensation technique to estimate the low-pass filtered content. The highly three-dimensional nature of the jet in crossflow limits the effectiveness of two-wire X-wire probes, which have an inherent assumption of planar 2-D flow. Therefore, the physical parameters and limitations of the experimental probes should always be considered when designing an experiment.

The LES and experimental heat fluxes exhibit excellent agreement. The asymmetry introduced by the physical limitations of the probe (i.e., offset between temperature and velocity wires) has been reproduced using the LES dataset. Future work should
be done to develop a correction technique to estimate these offset effects \textit{a priori}.

The LES resolved turbulent heat flux vectors are shown to be significantly different from those obtained by applying the gradient diffusion hypothesis inherent to most RANS flow solvers. This is somewhat expected given the complex three dimensional nature of the heated jet in crossflow, and inherent limitations of the GDH model.
REFERENCES


*Computers and Fluids, 120, 57 - 69.*


A. Rotation of the Reynolds Stress Tensor

The Reynolds stress tensor, $R_{ij}$, is calculated in Cartesian coordinates. Therefore, the tensor must be transformed to align the u-, v- and w- directions with the direction of the tube to evaluate the Reynolds stress. This is achieved through a tensor rotation of the Reynolds stress tensor about the spanwise direction, w. To transform a rank-2 tensor, first the coordinate transformation rotation matrix is needed. For rotation about the z-axis,

$$ \overrightarrow{X'} = \overrightarrow{Q} \overrightarrow{X} \quad (A.1) $$

$$ \begin{bmatrix} x' \\ y' \\ z' \end{bmatrix} = \begin{bmatrix} \cos(\theta) & -\sin(\theta) & 0 \\ \sin(\theta) & \cos(\theta) & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} x \\ y \\ z \end{bmatrix} \quad (A.2) $$

Here $\overrightarrow{X}$ is the Cartesian coordinate system, $\overrightarrow{X'}$ is the coordinate system aligned with the cooling tube, and $\overrightarrow{Q}$ is the rotation matrix. For the tensor rotation
however, the transpose of the vector rotation matrix is also required. The full
transformation, for rotation about the z-axis is:

\[ \overline{\tau}^T = \overline{Q}^T \overline{\vec{\tau}} \overline{Q} \]  
(A.3)
\[
\begin{bmatrix}
\tau_{x'} & \tau_{x'y'} & \tau_{x'z'} \\
\tau_{y'x'} & \tau_{y'y'} & \tau_{y'z'} \\
\tau_{z'x'} & \tau_{z'y'} & \tau_{z'z'}
\end{bmatrix}
= \begin{bmatrix}
cos(\theta) & \sin(\theta) & 0 \\
-\sin(\theta) & \cos(\theta) & 0 \\
0 & 0 & 0
\end{bmatrix} \begin{bmatrix}
\tau_{xx} & \tau_{xy} & \tau_{xz} \\
\tau_{yx} & \tau_{yy} & \tau_{yz} \\
\tau_{zx} & \tau_{zy} & \tau_{zz}
\end{bmatrix} \begin{bmatrix}
cos(\theta) & -\sin(\theta) & 0 \\
\sin(\theta) & \cos(\theta) & 0 \\
0 & 0 & 0
\end{bmatrix}
\]
(A.4)

\[
\begin{bmatrix}
\tau_{x'} & \tau_{x'y'} & \tau_{x'z'} \\
\tau_{y'x'} & \tau_{y'y'} & \tau_{y'z'} \\
\tau_{z'x'} & \tau_{z'y'} & \tau_{z'z'}
\end{bmatrix}
= \begin{bmatrix}
cos(\theta) & \sin(\theta) & 0 \\
-\sin(\theta) & \cos(\theta) & 0 \\
0 & 0 & 0
\end{bmatrix} \begin{bmatrix}
\tau_{xx} \cos(\theta) + \tau_{xy} \sin(\theta) & \tau_{xy} \cos(\theta) - \tau_{xx} \sin(\theta) & \tau_{xz} \\
\tau_{yx} \cos(\theta) + \tau_{yy} \sin(\theta) & \tau_{yy} \cos(\theta) - \tau_{yx} \sin(\theta) & \tau_{yz} \\
\tau_{zx} \cos(\theta) + \tau_{zy} \sin(\theta) & \tau_{zy} \cos(\theta) - \tau_{xz} \sin(\theta) & \tau_{zz}
\end{bmatrix}
\]
(A.5)

\[
\begin{bmatrix}
\tau_{x'} & \tau_{x'y'} & \tau_{x'z'} \\
\tau_{y'x'} & \tau_{y'y'} & \tau_{y'z'} \\
\tau_{z'x'} & \tau_{z'y'} & \tau_{z'z'}
\end{bmatrix}
= \begin{bmatrix}
\tau_{xx} \cos^2(\theta) + \tau_{yy} \sin^2(\theta) + \tau_{xy} \sin(2\theta) & \tau_{xy} \cos(2\theta) - \frac{\tau_{xx} \sin(2\theta)}{2} + \frac{\tau_{yy} \sin(2\theta)}{2} & \tau_{xz} \\
\tau_{xy} \cos(2\theta) - \frac{\tau_{xx} \sin(2\theta)}{2} + \frac{\tau_{yy} \sin(2\theta)}{2} & \tau_{yy} \cos^2(\theta) + \tau_{xx} \sin^2(\theta) - \tau_{xy} \sin(2\theta) & \tau_{yz} \\
\tau_{xz} \cos(\theta) + \tau_{zy} \sin(\theta) & \tau_{zy} \cos(\theta) - \tau_{xz} \sin(\theta) & \tau_{zz}
\end{bmatrix}
\]
(A.6)