Experimental Investigation of Mist Film Cooling and Feasibility Study of Mist Transport in Gas Turbines

Reda M. Ragab

University of New Orleans, rragab@uno.edu

Follow this and additional works at: https://scholarworks.uno.edu/td

Part of the Aerodynamics and Fluid Mechanics Commons, Computational Engineering Commons, Energy Systems Commons, and the Heat Transfer, Combustion Commons

Recommended Citation
https://scholarworks.uno.edu/td/1762

This Dissertation-Restricted is brought to you for free and open access by the Dissertations and Theses at ScholarWorks@UNO. It has been accepted for inclusion in University of New Orleans Theses and Dissertations by an authorized administrator of ScholarWorks@UNO. The author is solely responsible for ensuring compliance with copyright. For more information, please contact scholarworks@uno.edu.
Experimental Investigation of Mist Film Cooling and Feasibility Study of Mist Transport in Gas Turbines

A Dissertation

Submitted to the Graduate Faculty of the University of New Orleans in partial fulfillment of the requirements for the degree of

Doctor of Philosophy in Engineering and Applied Science

By

Reda Ragab

B.Sc., Zagazig University, Egypt, 2000
M.Sc., Zagazig University, Egypt, 2008

December, 2013
Acknowledgment

Thanks go first to Allah (SWT), the Almighty God, who gave me the support and patience to finish this work.

The author likes to pay his respect and gratitude to his dissertation advisor Dr. Ting Wang, Professor, mechanical engineering department, for his active guidance, constant inspiration and encouragement, invaluable suggestions and constructive criticism throughout the research that helped it to come up to the present state. Respects and gratitude are also paid to Dr. Kazim Akyuzlu, professor of mechanical engineering department, Dr. Paul Schilling, professor and chairman of mechanical engineering department, Dr. Ralph Saxton, Professor of math department, and Dr. Brandon Taravella, assistant professor of naval architecture department, for serving as the dissertation committee members.

The author also acknowledges the gratitude for MEE Industries Inc. for the donation of the fogging system used in the experimental studies. The help of Dr. Brandon Taravella in manufacturing the fan-shaped hole geometry, in the rapid prototyping lab, is highly appreciated.

The author also acknowledges the gratitude for the active support and inspiration of his family members, specially his parents, wife, children, and siblings.
# Table of Contents

LIST OF FIGURES ................................................................. vi
LIST OF TABLES ................................................................. xiv
NOMENCLATURE ................................................................. xv
ABSTRACT ........................................................................... xvii

CHAPTER

1 INTRODUCTION ................................................................. 1
   1.1 Gas Turbine Systems ..................................................... 1
   1.2 Turbine Blade Cooling Technologies ................................. 3
      1.2.1 Film Cooling .......................................................... 3
   1.3 Theories of Film Cooling .................................................. 5
      1.3.1 Fundamental Concepts of Film Cooling ......................... 5
      1.3.2 Adiabatic Wall Temperature ......................................... 8
      1.3.3 Film Cooling Performance Enhancement ....................... 9
   1.4 Mist/Film Cooling Concept .............................................. 13
   1.5 Objectives ................................................................. 21

2 EXPERIMENTAL SETUP ...................................................... 23
   2.1 Experimental Test facility ............................................... 23
      2.1.1 Wind Tunnel System (Main Flow) ............................... 23
      2.1.2 Mist Film Generation and Transport .............................. 25
      2.1.3 Test Section .......................................................... 27
      2.1.4 PDPA System .......................................................... 29
   2.2 Instrumentation ............................................................ 37
      2.2.1 Particle Measurement .................................................. 37
      2.2.2 Temperature Measurement ........................................... 40
      2.2.3 Flow Measurement .................................................... 46
   2.3 Uncertainty Analysis ..................................................... 49

3 EXPERIMENTAL RESULTS AND DISCUSSION ......................... 54
   3.1 Flow Conditions .......................................................... 54
   3.2 Heat Transfer Results ..................................................... 57
      3.2.1 Short Section with Cylindrical Holes ........................... 59
         3.2.1.1 Case1 and Case 2 .............................................. 59
         3.2.1.2 Case3 and Case 4 .............................................. 64
         3.2.1.3 Case5 and Case 6 .............................................. 66
         3.2.1.4 Atomizing Water Temperature Effect ....................... 67
      3.2.2 Extended Section with Cylindrical Holes (Insulated) ......... 69
         3.2.2.1 Case 7 and Case 8 ............................................ 69
         3.2.2.2 Case 9 and Case 10 ........................................... 72
         3.2.2.3 Case11 and Case 12 ........................................... 74
3.2.3 Extended Section with Cylindrical Holes (Non-Insulated) …… 76
  3.2.3.1 Case 13 and Case 14………………………………….…… 76
  3.2.3.2 Case 15 and Case 16 …………………………………… 78
  3.2.3.3 Case 17 and Case 18………………………………….…… 79
  3.2.3.4 Effect of the Test Section Insulation (Cylindrical holes) 80
3.2.4 Extended Section with Fan-Shaped Holes (Insulated) ………… 81
  3.2.4.1 Case 19 and Case 20…………………………………… 82
  3.2.4.2 Case 21 and Case 22 …………………………………… 86
  3.2.4.3 Case 23 and Case 24………………………………….…… 88
3.3 Droplet measurement …………………………………………….. 91
  3.3.1 Cylindrical Holes …………………………………………….. 91
    3.3.1.1 Droplet Size Distribution Plots ……………………….. 92
    3.3.1.2 Effect of Blowing ratio, M…………………………… 108
    3.3.1.3 The Proposed Coolant Air/Droplet Spreading Profile 116
    3.3.1.4 Linking Particle Data with Heat Transfer…………… 117
  3.3.2 Fan-Shaped (Diffusion) Holes………………………………… 120
    3.3.2.1 Size Distribution with Data Rate Plot ………………… 120
    3.3.2.2 The Proposed Coolant Air/Droplet Spreading Profile … 131
    3.3.2.3 Linking Particle Data with Heat Transfer………….. 132
    3.3.2.4 Effect of Blowing ratio, M…………………………… 133
3.4 Some 3D Aspects of the Film Layer………………………………….. 138

4 MIST TRANSPORT TO HIGH PRESSURE TURBINE COMPONENTS 143
4.1 Introduction………………………………………………………..…… 143
4.2 An Investigation of Mist Transport in vanes………………………… 147
  4.2.1 Studied Configuration………………………………………… 147
  4.2.2 Zero-Dimensional Model……………………………………….. 149
4.2.3 CFD Calculations…………………………………………………. 150
  4.2.3.1 Numerical method……………………………………….. 150
  4.2.3.2 Computational domain ………………………………… 154
  4.2.3.3 Boundary conditions……………………………………. 154
  4.2.3.4 Meshing and simulation procedure…………………… 158
4.2.4 Results and Discussion………………………………………….. 160
  4.2.4.1 Model Validation………………………………………. 160
  4.2.4.2 Base case results ……………………………………… 160
  4.2.4.3 Parametric study ……………………………………… 164
4.2.5 Conclusions……………………………………………………… 170
4.3 An Investigation of Mist Transport in Rotor Cavity Feed Channel…… 171
  4.3.1 Background………………………………………………… 171
  4.3.2 Objective and Scope………………………………………… 175
  4.3.3 One-Dimensional Model…………………………………… 176
  4.3.4 CFD Calculations…………………………………………… 178
    4.3.4.1 Mathematical model………………………………….. 178
List of Figures

Figure 1.1 Schematic of Gas Turbine Cycle……………………………………….. 2
Figure 1.2 GE 7FA natural gas turbine…………………………………………….. 2
Figure 1.3 Cooling air flow in a high pressure turbine stage…………………... 3
Figure 1.4 Ideal tangential slot film cooling……………………………………... 10
Figure 1.5 Defined geometries for four types of shaped film holes……………… 11
Figure 2.1 Schematic of mist cooling experiment apparatus…………………… 24
Figure 2.2 Photo of (a) the filter box and (b) the heating unit…………………… 24
Figure 2.3 Impaction pin atomizer…………………………………………………. 25
Figure 2.4 Structure of the mixing chamber………………………………………. 27
Figure 2.5 Test section outline and dimensions (a) short section (b) extension… 28
Figure 2.6 Test pieces with (a) cylindrical holes (b) fan-shaped holes…………… 29
Figure 2.7 Dual-beam scatter by a spherical droplet…………………………… 30
Figure 2.8 Measurement volume dimensions…………………………………… 31
Figure 2.9 Scattered light intensity variation……………………………………… 33
Figure 2.10 PDPA system overview……………………………………………… 34
Figure 2.11 PDPA Fiberlight™ Multicolor Beam Separator (Generator)……….. 35
Figure 2.12 PDPA Coupler………………………………………………………… 36
Figure 2.13 Calibration of the PDPA measurement……………………………… 37
Figure 2.14 Laser beams alignment correction…………………………………… 39
Figure 2.15 Overview of temperature measurement system…………………… 42
Figure 2.16 Thermocouples instrumentation and probe………………………… 42
Figure 2.17 Components used in the thermocouple measurement……………… 44
Figure 2.18 Temperature variations inside the isothermal box…………………... 44
Figure 2.19 Thermocouple calibration…………………………………………… 45
Figure 2.20 Thermocouples Layout (a) short test section (b) extension piece…… 45
Figure 2.21 Pressure gauge and flow meter……………………………………… 46
Figure 2.22 Pitot-Static tube………………………………………………………. 47
Figure 2.23 Calibration curve of the pressure transducer………………………. 49
Figure 2.24 Illustration of the concept of uncertainty………………………….. 50
Figure 3.1 Approaching Velocity profile measured at X/D= -4………………….. 56
Figure 3.2  Contour of cooling effectiveness for (a) Case 1 (M=0.6, air-only film) (b) Case 2 (M=0.6, mist/air film) ................................................................. 61

Figure 3.3  Cooling effectiveness and net enhancement (for Cases 1&2) compared with the experimental data from Goldstein et al. and Rhee et al. for M=0.66 (a) centerline data (b) spanwise averaged data ................................. 61

Figure 3.4  Contour of cooling effectiveness (a) Case 3 (M=1.0, air-only film) (b) Case 4 (M=1.0, mist/air film). ........................................................................ 64

Figure 3.5  Cooling effectiveness and net enhancement for M =1.0 (for Cases 3&4) (a) centerline data (b) spanwise averaged data ............................ 64

Figure 3.6  Contour of cooling effectiveness (a) Case 5 (M=1.4, air film only) (b) Case 6 (M=1.4, mist/air film). ................................................................. 66

Figure 3.7  Cooling effectiveness and net enhancement for M=1.4 (a) centerline data (b) spanwise averaged data ........................................................... 66

Figure 3.8  Effect of atomizing water temperatures (M=0.66, centerline data) on (a) cooling effectiveness and net enhancement (b) Diameter distribution at X/D=28 ................................................................. 68

Figure 3.9  Contour of cooling effectiveness with atomizing water at 70 °F (M=0.66, mist/air). ...................................................................................... 68

Figure 3.10  Contour of cooling effectiveness (a) Case 7 (M=0.66, Insulated, extended section, air-only film) (b) Case 8(M=0.66, Insulated, extended section, mist/air film). ................................................................. 70

Figure 3.11  Cooling effectiveness and net enhancement for M=0.66 (Case7&Case8) (a) Centerline data (b) Spanwise averaged data............. 71

Figure 3.12  Contour of cooling effectiveness (a) Case 9 (M=1.0, Insulated, extended section, air-only film) (b) Case 10 (M=1.0, Insulated, extended section, mist/air film). ................................................................. 73

Figure 3.13  Cooling effectiveness and net enhancement for M=1.0 (Case9&Ccase10) (a) Centerline data (b) Spanwise averaged data ...... 73

Figure 3.14  Contour of cooling effectiveness (a) Case 11(M=1.0, Insulated, extended section, air-only film) (b) Case 12 (M=1.0, Insulated, extended section, mist/air film). ................................................................. 75

Figure 3.15  Cooling effectiveness and net enhancement for M=1.4 (Case11&Case12) (a) Centerline data (b) Spanwise averaged data ...... 75

Figure 3.16  Contour of cooling effectiveness (a) Case 13(M=0.66, Non-insulated, extended section, air-only film) (b) Case 14 (M=0.66, Non-insulated, extended section, mist/air film). ................................................................. 77

Figure 3.17  Cooling effectiveness and net enhancement for M=0.66 (Case13&Case14) (a) Centerline data (b) Spanwise averaged data...... 77
Figure 3.18 Contour of cooling effectiveness (a) Case 15 (M=1.0, Non-insulated, extended section, air-only film) (b) Case 16 (M=1.0, Non-insulated, extended section, mist/air film) ................................................................. 78

Figure 3.19 Cooling effectiveness and net enhancement for M=1.0 (Case 15 & Case 16) (a) Centerline data (b) Spanwise averaged data ……… 78

Figure 3.20 Contour of cooling effectiveness (a) Case 17 (M=1.4, Non-insulated, extended section, air-only film) (b) Case 18 (M=1.4, Non-insulated, extended section, mist/air film) ................................................................. 79

Figure 3.21 Cooling effectiveness and net enhancement for M=1.4 (Case 17 & Case 18) (a) Centerline data (b) Spanwise averaged data ……… 79

Figure 3.22 Effect of test section insulation on net enhancement for M=0.6 (a) Centerline data (b) Spanwise averaged data …………………………… 80

Figure 3.23 Effect of test section insulation on net enhancement for M=1.0 (a) Centerline data (b) Spanwise averaged data …………………………… 81

Figure 3.24 Effect of test section insulation on net enhancement for M=1.4 (a) Centerline data (b) Spanwise averaged data …………………………… 81

Figure 3.25 Contour of cooling effectiveness for M=0.66 for Fan-shaped holes (a) Case 19, air-only film (b) Case 20, mist/air film ………………… 84

Figure 3.26 Cooling effectiveness and net enhancement for M=0.66 for Fan-shaped holes (a) Centerline data (b) Spanwise averaged data …………………………… 84

Figure 3.27 Cooling effectiveness enhancement produced by fan-shaped holes vs. cylindrical holes for M=0.66 (a) air-only case (b) air/mist ………… 85

Figure 3.28 Comparison of mist film cooling enhancement between fan-shaped holes and cylindrical holes with M=0.66 (a) centerline data (b) spanwise data ……………………………………………………………………… 85

Figure 3.29 Contour of cooling effectiveness for M=1.0 for Fan-shaped holes (a) Case 3, air-only film (b) Case 4, mist/air film …………………………… 87

Figure 3.30 Cooling effectiveness and net enhancement for M=1.0 for Fan-shaped holes (a) Centerline data (b) Spanwise averaged data …………………………… 87

Figure 3.31 Cooling effectiveness enhancement produced by fan-shaped holes vs. cylindrical holes for M=1.0 (a) air-only case (b) air/mist ………… 88

Figure 3.32 Comparison of mist film cooling enhancement between fan-shaped holes and cylindrical holes with M=1.0 (a) centerline data (b) spanwise data ……………………………………………………………………… 88

Figure 3.33 Contour of cooling effectiveness for M=1.4 for Fan-shaped holes (a) Case 23, air-only film (b) Case 24, mist/air film …………………………… 89

Figure 3.34 Cooling effectiveness and net enhancement for M=1.4 for Fan-shaped holes (a) Centerline data (b) Spanwise averaged data …………………………… 90
Figure 3.35  Cooling effectiveness enhancement produced by fan-shaped holes vs. cylindrical holes for M=1.4 (a) air-only case (b) air/mist .......................... 90
Figure 3.36  Comparison of mist film cooling enhancement between fan-shaped holes and cylindrical holes with M=1.4 (a) centerline data (b) spanwise data ................................................................. 90
Figure 3.37  Distributions of droplet size and data rate at different Y and X/D locations for Case 14 (M=0.66) [Cylindrical Holes, Extended Section] ................................................................. 94
Figure 3.38  Distribution of data rate and Reynolds shear stress at different Y and X/D locations for Case 14 (M=0.66) [Cylindrical Holes, Extended Section] .................................................. 96
Figure 3.39  Droplet size distribution of various Y locations at X/D=1, M=0.66 (Case14: cylindrical holes, extended section, with intensity validation) .................................................. 98
Figure 3.40  Droplet size distribution of various Y locations at X/D=13 (Cylindrical holes, Extended Section, M=0.66) ............................................................. 101
Figure 3.41  Clarification of the Bi-Modal size distribution through laser light intensity strength maps at M= 0.66 at X/D =13 (a) Y=15 mm (b) Y=16 mm ............................................................. 102
Figure 3.42  Droplet size distribution of various Y locations at X/D=28 (Cylindrical Holes, Extended Section, M=0.66) ............................................................. 103
Figure 3.43  Droplet size distribution of various Y locations at X/D=48 (Cylindrical Holes, Extended Section, M=0.66) ............................................................. 105
Figure 3.44  Droplet size distribution of various Y locations at X/D=64 (Cylindrical Holes, Extended Section, M=0.66) ............................................................. 106
Figure 3.45  Droplet size distribution of various Y locations at X/D=80 (Cylindrical Holes, Extended Section, M=0.66) ............................................................. 107
Figure 3.46  Distributions of droplet size and data rate at different Y and X/D locations for Case 14 (M=1.0) [Cylindrical Holes, Extended Section]................................. 109
Figure 3.47  Distributions of droplet size and data rate at different Y and X/D locations for Case 14 (M=1.4) [Cylindrical Holes, Extended Section] ................................................................. 110
Figure 3.48  Distributions of data rate and Reynolds shear stress at different Y and X/D locations for the cylindrical holes case with M=1.0 .......................... 111
Figure 3.49  Distributions of data rate and Reynolds shear stress at different Y and X/D locations for the cylindrical holes case with M=1.4 ......................... 112
Figure 3.50  Distributions of droplet size and the droplets mean velocity at different Y and X/D locations at M=0.66 with cylindrical holes .......................... 113
Figure 3.51  Distributions of droplet size and the droplets mean velocity at different Y and X/D locations at M=1.0 with cylindrical holes ......................... 114
Figure 3.52  Distributions of droplet size and the droplets mean velocity at different Y and X/D locations at M=1.4 with cylindrical holes

Figure 3.53  The relationship between the mist film cooling enhancement and the mid-plane shape of the mist/air coolant film layer for M=0.6 Case

Figure 3.54  The mid-plane shape of the mist/air coolant film layer at the hole centerline with the Net Enhancement of cooling effectiveness (a) M=1.0 (b) M=1.4

Figure 3.55  A qualitative illustration of droplet size distribution in reference to the proposed mist film profile for low blowing ratio M=0.6

Figure 3.56  Distribution of droplet size and data rate at different X/D locations for Case 20 (M=0.6) [Fan-Shaped holes]

Figure 3.57  Distribution of data rate and Reynolds shear stress at different X/D locations for Case 20 (M=0.6) [Fan-Shaped holes]

Figure 3.58  Droplet size distribution of various Y locations at X/D=1 (Fan-Shaped Holes, M=0.66)

Figure 3.59  Droplet size distribution of various Y locations at X/D=13 (Fan-Shaped Holes, M=0.66)

Figure 3.60  Droplet size distribution of various Y locations at X/D=28 (Fan-Shaped Holes, M=0.66)

Figure 3.61  Droplet size distribution of various Y locations at X/D=48 (Fan-Shaped Holes, M=0.66)

Figure 3.62  Droplet size distribution of various Y locations at X/D=64 (Fan-Shaped Holes, M=0.66)

Figure 3.63  Droplet size distribution of various Y locations at X/D=80 (Fan-Shaped Holes, M=0.66)

Figure 3.64  Illustration for droplet distribution envelope (or droplet layer) versus cooling film layer for low blowing ratio (M=0.6) mist film cooling [Fan-Shaped hole]

Figure 3.65  The mid-plane shape of the mist/air coolant film layer for M=0.6 Case at the hole centerline with the Net Enhancement of cooling effectiveness plotted on the secondary Y-axis

Figure 3.66  Distribution of droplet size and data rate at different X/D locations for case 22 (M=1.0) [Fan-shaped Holes]

Figure 3.67  Distribution of droplet size and data rate at different X/D locations for case 24 (M=1.4) [Fan-shaped Holes]

Figure 3.68  Distributions of data rate and Reynolds shear stress at different X/D locations for the Fan-Shaped holes case with M=1.0

Figure 3.69  Distributions of data rate and Reynolds shear stress at different X/D locations for the Fan-Shaped holes case with M=1.4
Figure 3.70  The mid-plane profiles of the mist/air coolant film layer at the hole centerline with the Net Enhancement of cooling effectiveness plotted on the right Y-axis  (a) M=1.0  (b) M= 1.4 ................................. 138

Figure 3.71  Lateral distributions of diameter and data rate at selected X/D and Y locations downstream of the cylindrical holes for Case 14 (M=0.66) .... 139

Figure 3.72  Lateral distributions of diameter and data rate at selected X/D and Y locations downstream of the fan-shaped holes for Case 20 (M=0.66) ... 142

Figure 4.1  Schematic of mist cooling of turbine vanes using water mist injected through the Cooling air passage. (Modified from (Kurzke, 2007))....... 147

Figure 4.2a  Cooling air flow in a high pressure turbine stage (Oldfield, 2007) .... 148

Figure 4.2b  Schematic of turbine vane with cooling cavity (yellow) and film cooling holes (Dennis, 2006).......................................................... 148

Figure 4.3  The computational domain ............................................................ 154

Figure 4.4  Contours of the static temperature [K] at the mid-plane in the dry base case (No water mist is injected) .................................................. 160

Figure 4.5  Contours of the static temperature (K) at the mid-plane in the mist base case (10% Mist Ratio, 30 µm initial droplet diameter, 900 K cavity wall temperature) ...................................................... 161

Figure 4.6  Contours of the mass fraction of water vapor at the mid-plane in the mist base case ................................................................. 162

Figure 4.7  Droplet traces colored by droplet diameter [cm] in the base case (10% Mist Ratio, 30 µm initial diameter, 900 K cavity wall temperature) .... 163

Figure 4.8  Droplet traces colored by residence time [S] in the base case (10% Mist Ratio, 30 µm initial diameter, 900 K cavity wall temperature) .... 163

Figure 4.9  Diameter [m] distribution histogram at exit holes in the base case (10% Mist Ratio, 30 µm initial diameter, 900 K cavity wall temperature) ...................................................... 164

Figure 4.10  Diameter [m] distribution histogram at exit holes with (a) 5% Mist Ratio (b) 15% Mist Ratio ............................................................. 165

Figure 4.11  Effect of changing Mist Ratio (%) on droplet exit diameter .......... 165

Figure 4.12  Diameter [m] distribution histogram at exit holes with (a) 20 µm initial diameter  (b) 40 µm initial diameter ............................................. 166

Figure 4.13  Effect of changing droplet injection (Initial) diameter on droplet exit diameter ................................................................. 166

Figure 4.14  Effect of changing cavity wall temperature on droplet exit diameter .... 167

Figure 4.15  Effect of changing wall droplet boundary condition on droplet exit diameter distribution .......................................................... 168
Figure 4.16  Diameter[m] distribution histogram at exit holes with Rossin-Rammler initial diameter distribution…………………………………………………………169
Figure 4.17  Heavy frame turbine secondary flow regions (Dennis, 2006)...........172
Figure 4.18  Conceptual picture of forced convective boiling (a) different phase-change regimes with a uniform heat flux boundary condition (b) subcooled boiling schematic…………………………………………………………174
Figure 4.19  Geometry and flow conditions in the simulated pressurized liquid water feed channel…………………………………………………………176
Figure 4.20  One-dimensional calculation of water mean temperature distribution for different water inlet temperatures……………………………………178
Figure 4.21  RPI Subcooled nucleate boiling model……………………………………180
Figure 4.22  Validation of the experiment of Bartolemei and Chanturiya (1967) using the RPI model…………………………………………………………184
Figure 4.23  The computational domain, boundary types, and meshes………………185
Figure 4.24  Mesh sensitivity analyses…………………………………………………187
Figure 4.25  Contours of (a) vapor volume fraction (b) average liquid static temperature (c) vapor static temperature at different sections of the channel (Baseline Case)…………………………………………………………191
Figure 4.26  Variation of temperature and volume fraction (right axis) with position along the channel ……………………………………………………………191
Figure 4.27  Velocity magnitude [m/s] at the axis (b) heat flux distribution………………192
Figure 4.28  Variation of the vapor volume fraction along the conjugate wall for different channel wall temperatures………………………………………193
Figure 4.29  Variation at the channel exit plane for different wall temperatures. (a) $V_f$ and vapor velocity $V$ (secondary axis) (b) Liquid velocity $V$ and liquid static temperature $T_{sl}$ (secondary axis)…………………………………………………………193
Figure 4.30  Variation of vapor volume fraction along the conjugate wall for three different channel lengths…………………………………………………194
Figure 4.31  Variation of the vapor volume fraction along the conjugate wall for different (a) inlet subcooling (b) operating pressures……………………196
Figure 4.32  Effect of gravity on the vapor volume fraction at the contact wall……….196
Figure 4.33  Effect of reducing number of channels on the vapor volume fraction at the contact wall ……………………………………………………………197
Figure 4.34  Schematic of Pre-Swirl nozzles, Seals, and Cover-plate disk cavity (Snowsill and Young, 2006)………………………………………………200
Figure 4.35  The computational domain………………………………………………203
Figure 4.36  Mesh sensitivity analyses (for the rotor study)…………………………205
Figure 4.37  Droplet traces in the rotor cover-plate disk cavity colored by (a) droplet diameter (m) (b) residence time (s) (Base Case: 10% Mist Ratio, 50 µm initial diameter, 650 K wall temperatures)………………………… 207

Figure 4.38  Contours of temperature [K] at the mid-plane in the base case (a) Static temperature dry (b) Static temperature wet (c) Relative total temperature dry (d) Relative total temperature wet…………………….. 208

Figure 4.39  Vectors of relative velocity [m/s] field in the cavity mid-plan in the base case ……………………………………………………………… 209

Figure 4.40  Exit relative total temperatures and water consumption [gal/min] for different (a) Mist ratios (b) Initial diameters …………………………… 210

Figure 4.41  Diameter [m] distribution histogram at the cavity exit (15% Mist Ratio, 50 µm initial diameter)………………………………………… 211

Figure 4.42  Droplets exit mean diameter [µm] results for different (a) initial diameters (b) mist ratios ………………………………………………… 212

Figure 4.43  Simplified cross section illustrating the cooling air flow path from a nozzle to a bucket via the nozzle diaphragm (Eldrid et al., 2002) …… 213

Figure 4.44  Computational domain and mesh for the nozzle diaphragm problem…. 214

Figure 4.45  Static temperature contours in the base case (10% Mist Ratio, 50 µm initial diameter) ………………………………………………… 216

Figure 4.46  Droplet traces in the nozzle diaphragm problem colored by (a) droplet diameter [m] (b) residence time [s] (Base Case: 10% Mist Ratio, 50 µm initial diameter) ………………………………………………… 216

Figure 4.47  Velocity field in the base case (10% Mist Ratio, 50 µm initial diameter)…………………………………………………………………… 217

Figure 4.48  Droplet exit diameter [µm] and temperature [K] results for different (a) Mist Ratios (b) Initial diameter ……………………………….. 218
List of Tables

Table 2.1  Uncertainty Analysis: Zeroth-Order and Pretest Uncertainties ............. 53
Table 2.2  Uncertainty Analysis: First-Order and Nth order Uncertainties ........... 53
Table 3.1  Main flow temperature [°C] measurement ........................................ 55
Table 3.2  Summary of flow conditions ............................................................... 57
Table 3.3  List of experimental cases ................................................................. 58
Table 3.4  The Y location for coolant film upper (peak data rate) and lower (2 HZ data rate) boundaries acquired from the data rate curve (Cylindrical holes) for different Blowing Ratios ................................................................. 116
Table 3.5  The Y location for coolant film upper and lower boundaries acquired from the data rate curve (Fan-Shaped holes) for different Blowing Ratios ................................................................. 122
Table 4.1  Droplet residence time and potential traveling distance for two different sizes of water droplets ................................................................. 149
Table 4.2  Boundary conditions ........................................................................ 155
Table 4.3  Base case conditions ......................................................................... 156
Table 4.4  The matrix of the parametric study .................................................... 158
Table 4.5  Grid independency analysis ............................................................... 159
Table 4.6  The base case values ....................................................................... 186
Table 4.7  The matrix of parametric studies ....................................................... 186
Table 4.8  Time step size [s] sensitivity study results ........................................... 188
Table 4.9  Droplet residence time and potential traveling distance for different sizes of water droplets ................................................................. 201
Table 4.10 Base case conditions for the rotor cavity .......................................... 204
Table 4.11 The matrix of the parametric study for the rotor cavity problem .......... 211
Table 4.12 The matrix of the parametric study for the nozzle diaphragm problem... 215
NOMENCLATURE

A  Area [m\(^2\)]
C  Vapor concentration [kg/m\(^3\)] or Specific heat [J/kg.K]
C_D Aerodynamic drag coefficient
D, d Diameter [m]
D_{10} Arithmetic Mean Diameter [m]
D_{32} Mean Sauter Diameter [m] (ratio of droplet volume to its surface area)
E  Total energy [J]
\(f_{bw}\) Bubble departure frequency
H  Total enthalpy [J]
h Convective heat transfer coefficient [W/m\(^2\).K], Species enthalpy [J/kg]
h_{fg} Latent heat, [J/kg]
k  Kinetic energy per unit mass [J/kg]
K  Thermal conductivity [W/m · K]
L  Length [m]
k_B Boltzmann constant [1.38 \times 10^{-23} J/mole · K]
k, k_c Mass transfer coefficient (units vary)
Nu Nusselt number (hL/k) (dimensionless)
N_w Nucleate site density
P  Pressure [Pa]
Pr Prandtl number (= \(\alpha/\nu\)) (dimensionless)
q'' Heat flux [W/m\(^2\)]
R, r Radius [m]
Re Reynolds number (UL/\(\nu\)) (dimensionless)
S_f Force source term, [N/m\(^3\)]
S_h Energy source term, [W/m\(^3\)]
S_j Source of the \(J^{th}\) species [kg/m\(^3\).s]
S_m Mass source from the DPM, [kg/m\(^3\).s]
Sh Sherwood number (hL/D)
Sc Schmidt number (v/D) (dimensionless)
T  Temperature [K, °C, °F]
ΔT_{sub}  Liquid subcool = T_{sat}-T_l
ΔT_{sup}  Wall superheat = T_w-T_{sat}
t  Time [s], thickness [m, ft, in]
U  Free-stream velocity [m/s]
u; v; w  Velocity components [m/s]
V_f  Vapor volume fraction (dimensionless)

Greek Letter
α  Thermal diffusivity [m²/s], Volume fraction (dimensionless)
ε  Emissivity (dimensionless), Turbulent dissipation rate [m²/s³]
μ  Dynamic viscosity [Pa·s]
ν  Kinematic viscosity [m²/s]
σ  Stefan-Boltzmann constant [5.67x10⁻⁸ W/m²·K⁴]
≈  Stress tensor [Pa]
τ  Shear stress [Pa], Time scale [s]
ρ  Density [kg/m³]
ζ  Random number

Subscripts
b  Bubble, bulk
C  Convection
E  Evaporation
i  Initial, Term number, Tensor index (1, 2, 3), liquid vapor interface
l  Liquid
Q  Quenching
Sat  Saturation
sp.  Single phase
sub  Subcool
w  wall
∞  Away from the computational cell
ABSTRACT

In the modern advanced gas turbines, the turbine inlet temperature may exceed 1500°C as a requirement to increase power output and thermal efficiency. Therefore, it is imperative that the blades and vanes are cooled so they can withstand these extreme temperatures. Film cooling is a cooling technique widely used in high-performance gas turbines. However, the film cooling effectiveness has almost reached plateau, resulting in a bottleneck for continuous improvement of gas turbines' efficiency.

In this study, an innovative cooling scheme, mist film cooling is investigated through experiments. A small amount of tiny water droplets with an average diameter about 10-15 µm (mist) is injected into the cooling air to enhance the cooling performance. A Phase Doppler Particle Analyzer (PDPA) system is used for droplet measurements. Mist film cooling performance is evaluated and compared against air-only film cooling. This study continues the previous work by (a) adding fan-shaped holes and comparing their cooling performance with the round holes, (b) extending the length of the test section to study the performance farther downstream the injection holds, and (c) using computational simulation to investigate the feasibility of transporting mist to the film cooling holes through gas turbine inside passages.

The results show that, with an appropriate blowing ratio, the fan-shaped holes performs about 200% better than round holes in cooling effectiveness and adding 10% (wt.) mist can further enhance cooling effectiveness 170% in average. Farther downstream away from the injection holes (X/D> 50), mist cooling enhancement prevails and actually increases significantly. PDPA measurements have shed lights to the fundamental physics of droplet dynamics and their interactions with thermo-flow fields. These experimental results lead to either using lower amount of cooling air or use fewer number of cooling holes rows. This means higher gas turbine power output, higher thermal efficiency, and longer components life which will reflect as a cheaper electricity bill.

Computational Fluid Dynamics (CFD) showed that it is feasible to transport the water mist, with initial diameters ranging from 30 µm-50 µm and mist ratio of 10-15%, to the cooling holes on the surface of the turbine vanes and rotors to provide the desired film cooling.

Keywords: Gas Turbines, Heat Transfer, Film/Mist Cooling, Experimental Study, Mist Transport, CFD, PDPA.
CHAPTER ONE

INTRODUCTION

In this chapter, the basic fundamentals of gas turbine cycle are introduced. Most common gas turbine hot section cooling technologies will be briefly mentioned and then the discussion will be extended focusing on film and mist cooling which is the focus of the current study. The motivations, objectives and tasks of the current study are then stated.

1.1 Gas Turbine System

Gas turbines play a vital role in today’s industrialized society. A gas turbine engine is essentially an energy conversion device that converts the energy in the fuel to useful mechanical energy, similar to the car (reciprocating) engines, only with much larger power over weight ratio than the reciprocating engines. Gas turbine engines are widely used as a prime mover to provide shaft work do drive, for example, ships, tanks, compressors, pumps, or power plant generators, etc. Gas turbines are also designed as aeroengines to provide powerful propulsions for airplanes and military jets.

Figure 1.1 shows the fundamental working principle of a gas turbine: the air is first drawn into the compressor to be compressed to high pressure from 5 bars to 30 bars, mixed with the fuel and ignited in the combustor. The resulting hot gas from combustion is then directed into the turbine section. The energy is extracted through the hot gas expansion process over high-speed spinning turbine blades which transform the thermal energy into output shaft power. Figure 1.2 shows a typical heavy-frame land-base GE 7FA natural gas turbine engine to generate about 211MW electric power.

Efficiency is one of the most important parameters to evaluate the performance of a gas turbine engine. GE estimates that with even a 1% efficiency increase, the savings of operating costs over the life of a typical power plant is estimated to be $2 million a year over today's most efficient (57%-58%) combined-cycle plants (Power Technology, 2007). One of the most effective ways to improve the gas turbine system thermodynamic efficiency is to increase the turbine inlet temperature (TIT). However, TIT is limited by the highest temperature that the
material of the turbine blade can withstand. Therefore, technologies are needed to maintain the TIT high while keeping the airfoil relatively cool without losing efficiency. One of the essential approaches, which is the focus of this research, is using cooling schemes to protect the turbine blade from the hot main flue gas.

Figure 1.1 Schematic of a Gas Turbine

Figure 1.2 GE 7FA natural gas turbine (Source: GE-Power)
1.2 Turbine Blade Cooling Technologies

Many cooling schemes have been used over the past decades, for instance: transpiration cooling (Flack, 2005, Boyce, 2006), effusion cooling (Andrew et al., 1985), water (liquid) cooling (Bayley and Martin, 1971), liquid thermosyphon cooling (Japikse, 1973), steam cooling (Mukherjee, 1984). **Film/mist cooling will be the focus of the current work and will be discussed in more details in the following sections.**

1.2.1 Film cooling

Film cooling is one of the earliest turbine airfoils cooling technologies developed and is widely used in different types of turbine engines. It uses the compressed air (at 600 K approximately) extracted from certain compressor stage as the coolant. Film injection holes are placed in the body of the airfoils (stationary vanes and rotating blades) to allow coolant to pass from the internal cavity to the external surface, see Figure 1.3.

![Figure 1.3 Cooling air flow in a high pressure turbine stage (Oldfield, 2007)](image-url)
The ejection of coolant gas results in a layer or “film” of coolant gas flowing along the external surface of the airfoil, and thus protects the airfoils from the invasion of the hot flue gas. Hence, the term “film cooling” is used to describe this cooling scheme. With appropriate implementation of the film cooling scheme, the temperature of the blades can be lowered to approximately 1000K, which is permissible for reliable operation of the engine.

Film cooling is a mature technology and various reliable design methodologies have already been developed and are continuously improved. Although film cooling has worked well to protect the airfoils, it also exerts adverse effects on gas turbine performance. Only when the benefits outweigh the drawbacks, does application of the film cooling scheme make sense. Some general comments of the adverse effects of film cooling are summarized as following:

- For higher TIT, a large amount of compressed air is required. It is estimated that, under the Advanced Turbine System (ATS) condition where TIT is over 1400°C, over 35% of the overall compressor air will be used to cool the hot path components (Wenglarz, et al., 1994). This consumed precious compressed air represents a great part of the useful work generated by the turbine (to drive the compressor). Also, less air enters the combustor and as a result the total power output is reduced.

- Since the cooling air must pass through roughened small internal cooling passages before it is ejected into main flow, the coolant needs to have a much higher pressure than the main flow to overcome the pressure lost. This essentially consumes a part of turbine power to pressurize the coolant air.

- Film cooling has detrimental effects on the cycle efficiency of the gas turbine system because the coolant jet interferes with the main flow field and therefore causes aerodynamic and thermal losses (Little, et al., 1993; Bannister, et al., 1994).

- Severe film cooling makes it very difficult to increase the gas turbine TIT to above 1400°C. The highest possible temperature in a turbine system is the flame temperature in the combustor, which must be kept under 1500~1600 °C (2700~2900 °F) to maintain low NOx emissions (Cook, 1993; Farmer and Fulton, 1995). Due to the thermal mixing effects of film cooling, the gas turbine inlet temperature is always several hundred degrees lower than the flame temperature. As a result TIT is limited
by this large temperature drop between the flame temperature in the combustor and turbine inlet.

- Air coolant film blends in the main flow quickly; as a result air film cooling is only effective for a short range. Consequently in order to cover the whole blade, many rows of film holes are needed which is detrimental to the blade integrity and mechanical strength.
- Considering all above adverse effects of using the film cooling scheme, if too much compressed air is used for airfoil cooling, eventually the gas turbine efficiency will reduce, instead of increase, even though the TIT is increased and the airfoils are protected and survive.

1.3 Theories of Film Cooling

As a good starting point toward understanding the mechanism of mist/film cooling, the flow and heat transfer characteristics of film cooling and the corresponding theories are reviewed in this section.

1.3.1 Fundamental concepts of film cooling

Film cooling is one of the essential techniques to reduce the airfoil temperatures and thermal stresses which tend to increase as the turbine inlet temperature is continuously raised to augment gas turbine performance. Air bled from the compressor flows into the airfoils for internal cooling and then is ejected through small holes to form a layer of cooling film that blankets and protects the airfoil's surface from the hot mainstream gases. The film cooling jets consume valuable compressed air and therefore it is essential to continuously searching for new schemes to enhance film cooling performance and minimize the cooling mass flow.

*The ultimate goal of introducing film cooling to turbine airfoils is to reduce the heat load on the blade, i.e. reduce surface heat flux and/or lower airfoil's temperature, so the life of turbine airfoils can be significantly extended due to lower thermal stress and less spallation over thermal barrier coating.* Thus it is always desired to know how much heat flux or blade temperature can be actually reduced after film cooling is employed. However, due to the experimental difficulty in directly measuring the heat flux, the **Heat Flux Ratio (HFR) q" / q"₀**
is often evaluated indirectly through a theoretical relation developed by Mick and Mayle (1988) between two characteristic factors of film cooling heat transfer: adiabatic film effectiveness ($\eta$) and film heat transfer coefficients ($h_{af}$ and $h_o$), as:

$$\frac{q''}{q''_o} = \left(\frac{h_{af}}{h_o}\right) (1-\eta/\phi)$$  \hspace{1cm} (1.1)

in which, the adiabatic film effectiveness is defined as:

$$\eta = \frac{(T_g-T_{aw})}{(T_g-T_j)}$$  \hspace{1cm} (1.2)

Where $T_g$ is the main flow hot gas temperature, $T_j$ is the coolant temperature at the cooling jet hole exit, and $T_{aw}$ is the adiabatic wall temperature. $\eta$ is an excellent indicator of film cooling performance by comparing the insulated wall surface temperature ($T_{aw}$) with the would-be perfect wall temperature, $T_j$. If the film cooling were perfect, $\eta = 1$ and the wall is protected as cold as the cooling jet temperature. The adiabatic film heat transfer coefficient is defined as:

$$h_{af} = \frac{q''}{(T_{aw}-T_w)}$$  \hspace{1cm} (1.3)

where $T_w$ is the airfoil wall surface temperature that comes immediately in contact with the hot main gas flow. This definition is clear if the wall boundary condition is not adiabatic, which means that the actual wall heat flux would be driven by the potential adiabatic wall temperature $T_{aw}$. In Eq. 1, the local heat flux without film cooling is given as:

$$q''_o = h_o (T_g-T_w)$$  \hspace{1cm} (1.4)

the film cooling effectiveness, $\phi$, is defined as:

$$\phi = \frac{(T_g-T_w)}{(T_g-T_j)}$$  \hspace{1cm} (1.5)

The definition of $\phi$ is very similar to $\eta$ except $T_{aw}$ in Eq. 2 is replaced with $T_w$. To reduce complexity, $T_j$ is assumed the same as internal coolant temperature, $T_{ci}$. Although we can say that $\eta$ is a special case of the more generically defined film cooling effectiveness ($\phi$) when the wall is insulated, it is convenient to use both terms by designating $\phi$ for all non-adiabatic wall
conditions and \( \eta \) only for the adiabatic wall condition. \( \varphi \) has also been named as **non-dimensional metal temperature** or the **overall cooling effectiveness** in other literatures. The phrase of non-dimensional metal temperature is misleading because a high \( \varphi \)-value actually indicates a low surface temperature.

For a perfect film cooling performance, the film cooling effectiveness would have a value of unity (\( \eta \) or \( \varphi = 1.0 \)), i.e. \( T_{aw} \) equals to the coolant temperature (\( T_j \)) at the exit of the jet injection hole; while a value of \( \eta \) or \( \varphi = 0 \) means that the film cooling has no effect in reducing the wall temperature, which is as hot as the mainstream gas.

Equation (1.1) has been widely used over two decades until Wang and Zhao (2011) discovered that this equation is not correct because it assumes a typical value of \( \varphi = 0.6 \) for all modern gas turbine components; however, no supporting material or data were provided. Wang and Zhao (2011) thus clearly distinguish the difference between the film-cooled wall temperature \( (T_w, \varphi) \) and the wall temperature without cooling \( (T_{w,o}, \varphi_o) \), and derived a revised equation as eqn. (1.6)

\[
\frac{q^n}{q''_o} = \frac{h_{af} T_{aw}}{h_o T_g} - \frac{T_w}{T_{wo}} = \frac{h_{af}}{h_o} \left(1 - \frac{T_g}{T_{aw}} \frac{T_{aw}}{T_{w}} + \frac{T_{wo} - T_{aw}}{T_{w}} \right)
\]

\[
= \frac{h_{af}}{h_o} \left(1 - \frac{(T_g - T_{aw})}{(T_g - T_{wo})} \frac{(T_g - T_{w})}{(T_g - T_{j})} \right) + \frac{h_{af}}{h_o} \frac{(T_{wo} - T_{w})}{(T_g - T_{wo})} \frac{(T_g - T_{j})}{(T_g - T_{j})}
\]

\[
= \frac{h_{af}}{h_o} \left(1 - \frac{\eta}{\varphi_o} \right) + \frac{h_{af}}{h_o} \frac{(T_{wo} - T_{w})}{(T_g - T_{j})} \frac{(T_g - T_{j})}{(T_g - T_{j})}
\]

\[
= \frac{h_{af}}{h_o} \left(1 - \frac{\eta}{\varphi_o} \right) + \frac{h_{af}}{h_o} \frac{\varphi - \varphi_o}{\varphi_o}
\]

Where \( \varphi_o = (T_g - T_{w,o}) / (T_g - T_j) \)

The difference between Eqs. 1.1 and 1.6 is:
Since the wall temperature of a system without film will supposedly be lowered after the film is added (which is the purpose of film cooling), $\phi$ should be larger than $\phi_o$. And typically, $T_w < T_{aw}$ because heat is transferred from the heat source (main flow) into the wall. As a result, $\phi$ is generally higher than $\eta$. Thus $((\phi - \eta) \cdot (\phi - \phi_o)) / (\phi \cdot \phi_o) > 0$ and as a result, Eq. 1 under-predicts HFR and over-predicts the actual heat flux reduction. They provided examples showing the difference between Eq. (1.1) and (1.6) can be up to 30% and sometimes Eq. (1.1) could lead to false heat flux reduction over 100% (Wang and Zhao, 2011).

1.3.2 Adiabatic Wall Temperature

In the film cooling heat transfer analysis mentioned above, one of the core concepts is to deem the film cooled adiabatic wall temperature ($T_{aw}$) as the driving potential for the actual heat flux over the film-cooled surface as proposed by Goldstein (1971). This concept can be approached in two ways. Firstly, $T_{aw}$ can be simply treated as the highest temperature that the wall can possibly obtain when the wall is perfectly insulated. Hence, after the insulation is removed and the wall is subject to cooling underneath, the heat flux moves from the wall to the underneath cooling flow. This can be treated as being driven by the difference between $T_{aw}$ and $T_w$. Secondly, theoretically the concept of treating the adiabatic wall temperature as the driving temperature potential is drawn from compressible flow when viscous dissipation becomes important and acts as a heat source to drive the heat flux. But, in conditions where viscous dissipation is negligible, which is common in experiments under laboratory conditions, the heat source is not from near the wall but from the main hot gas stream. It must be noted that the viscous dissipation phenomenon is fundamentally different from the physics in the film cooling condition. In follows that viscous dissipation is crucial in compressible flow, since dissipation is the actual energy source which converts the flow’s kinetic energy to thermal energy near the
wall; whereas in the film-cooling flow the only energy source is the hot gas stream (assuming viscous dissipation is negligible in the film-cooling flow). Therefore, the highest temperature value $T_{aw}$ in the viscously dissipative flow is related to how much the converted thermal energy can be recovered by the wall via the recovery factor; whereas, in the film-cooling flow, the highest temperature that $T_{aw}$ can possibly reach is $T_g$. Thus, using the concept of recovery factor ($r$) or recovering temperature ($T_r$) in the film-cooling flow to explain $T_{aw}$ by some researchers seems artificial and shy of a support of flow physics if viscous dissipation is negligible. Lack of awareness of this difference has caused some confusion in explaining the film-cooling results.

1.3.3 Film cooling performance enhancement

There are numerous studies in the open literature discussing methods to improve the film cooling performance. The studies can be summarized in three different categories representing different perspectives to improve film cooling: coolant hole geometry and configuration, coolant/mainstream conditions, and airfoil geometry effects.

**Coolant hole geometry and configuration:** In this category, the major factors affecting the film cooling performance are shape of the hole, hole orientations (injection angle, compound angle, etc), length of the hole, hole spacing ($p/d$), number of rows and row spacing. The first two factors are comparatively more significant in determining the film cooling performance. The key reason behind changing the coolant hole geometry and configurations is to modify the flow field to allow the cooling film to cover as much as the surface area and as uniform as possible.

One of the goals of film cooling in gas turbines has been the achievement of ideal cooling films, such as those from two-dimensional (2D) continuous slots with uniformly distributed cooling supply, as shown in Figure 1.4. Due to the many competing constraints of turbine design (e.g., aerodynamics, thermal and mechanical stress, fabrication, and integrity of the blade), it is generally impractical to place such slots into the high temperature surfaces of the turbine components. As a consequence, film cooling has been implemented almost exclusively through the use of discrete holes and rows of evenly spaced holes. Some recent research has investigated the film cooling concept using slot jet with embedded holes (Wang, et.al, 2000) or so called trenched holes (Lu et. al, 2005; Bunker, R.S., 2002). In practical applications, both commercial and military, all film cooling holes are either round or shaped. Round (cylindrical) holes had been used for many decades until a single significant improvement has been made to transition
from round film holes to shaped film holes (Bunker, 2005). The use of the term “shaped,” while allowing a potentially vast number of geometries, is actually limited again to a single class of geometry. Shaped holes are composed of round metering or throat sections with a uniform and symmetric expanded exit region on the hot gas surface. Most commonly, all shaped holes applied in practice have fan diffuser exits with divergence angles between 10 and 15 deg on each lateral side as well as on the side into the surface. The great majority of axial film hole data also focuses on centerline angles of 30–35 deg relative to the surface tangent. These factors are common in practice. In all cases, it is the goal of discrete geometry film cooling to approach the formation of an ideal tangential slot injection leading to a continuous layer of film over the surface as shown in Figure 1.4.

![Figure 1.4 Ideal tangential slot film cooling (Bunker, 2005)](image)

The move to shaped film holes, first in military engines and then in commercial engines, is a natural extension of the round film hole toward a more slot like behavior, especially in the case of a row of film holes. This shaping is usually confined to the near-surface region, perhaps the outer 20–50% of the wall thickness, to maintain acceptable hole and hole-to-hole stress concentrations. The target for shaped film holes is to expand the exit area in the plane of the surface of the injection jet by a factor of 2–3 times that of the round jet without separation. Depending on the flow conditions, this may or may not correspond to an equivalent diffusion of the jet itself. This jet diffusion can lead to lower blowing ratios, lower aerodynamic mixing losses, and greater lateral coolant coverage, all of which may benefit cooling effectiveness and turbine efficiency.
Shaped film holes have limitations, however, requiring available wall thickness and surface distance to form the geometry. They must also be amenable to the application of protective coatings for oxidation resistance and thermal insulation. There are also places where shaping adds little or no value, for example on some airfoil pressure side regions where destabilizing effects may override shaped benefits. If possible though, nearly every film hole applied in practice today would be a shaped hole, or a modification of a shaped hole (Bunker, 2005). Virtually all shaped film hole studies can be classified into one of four hole geometries as depicted in Fig. 1.5 (Bunker, 2005). Geometry A is the classic shaped film hole that includes both lateral expansion, also known as fan-shaped, and expansion into the surface, also known as laidback. Geometry B contains only lateral exit expansion, while geometry C contains only laidback expansion. Geometry D is a conical film hole that expands from inlet to exit equally in each direction around its centerline. In actual applications, geometry A is the most common, in part due to performance, but also because manufacturing methods will usually produce diffusion in both directions. It is more difficult to produce the pure single expansion directions of B and C. Type D holes are not widely used, and the data pertaining to this type of shape are extremely limited.

Figure 1.5 Defined geometries for four types of shaped film holes (Bunker, 2005).
The study of Goldstein et al. (1974) was generally accredited as the first to research the use of shaped coolant holes to improve film cooling performance. They tested a 10° spanwise-diffused hole and compared the performance with that of cylindrical holes. With the same cooling mass flow, the shaped hole reduced the coolant momentum flux of the coolant jet and as a result, the coolant had less penetration into the mainstream compared to the cylindrical-hole cases, providing a better cooling effectiveness. Many studies followed his approach and there are still large numbers of ongoing researches focusing on the details of changing the hole shape combined with orientation angles and other previously mentioned factors to improve film cooling performance. For example, Thole et al. (1996) measured the flow fields for three types of injection holes: a cylindrical hole, a laterally diffused hole, and a forward-laterally diffused hole. Their results showed that diffusing the injection hole reduces the coolant penetration into the mainstream and reduces the intense shear regions when compared to cylindrical holes. Bell et al. (2000) found laterally diffused compound angle holes and forward diffused compound angle holes produce higher effectiveness over much wider ranges of blowing ratio and momentum flux ratio compared to the other three simple-angle configurations tested. Jia et al. (2003) studied the optimum inclination angle and they found that the recirculation bubble downstream the jet vanishes when the inclination angle is 30° or less. Several researchers focused on injection from a double row of cylindrical holes in order to approach a two dimensional film cooling situation. Jubran and Brown (1985), Jabbari et al. (1978), and Jubran and Maiteh (1999) all showed that for the same injected mass flow rate per unit span, the double row provides better cooling protection than the single row. The increased area ratio of the double row lowers the momentum of the coolant which provides better lateral spreading of the coolant. Spacing the holes closer together also increases the effectiveness in the lateral direction. Ligrani et al. (1996) showed that a compound angle orientation of the holes in the second row also increases the effectiveness. In addition, staggered rows of holes show better performance than inline rows of holes.

Coolant/Mainstream Conditions: Since the mixing between the coolant and main flow is partially controlled by turbulence diffusion, the turbulence intensity of both the main flow and the coolant jet affect the film cooling performance. High mainstream turbulence levels degrade film cooling performance by increasing heat transfer coefficients and generally decreasing film effectiveness. This is shown in the study of Bogard et. al (1996, 1998) simulating the large scale
turbulence with levels of Tu = 10% to 17% and an increase in heat transfer coefficient of 15% to 30% was found. Ekkad et al. (1995) also studied the effects of freestream turbulence on the heat transfer into a film cooled turbine blade. In their study it is concluded that high main flow turbulence results in an increased heat transfer coefficient and a slightly reduced film cooling effectiveness. On the other hand, for coolant jet, Mayhew et al. (2004) showed that low inlet turbulence intensity keeps the coolant close to the wall when the blowing ratio is low; while high inlet turbulence intensity helps bring the coolant back to the wall when the blowing ratio is high. This result was adopted in the studies of Brittingham and Leylek (2002) and Adami et al. (2002) in their numerical studies. An inlet plenum, as well as the flow arrangement, was included in their simulation in order to consider the effect of upstream flow conditions for the coolant flow. It was concluded that an accurate prediction of coolant discharge and wall coverage of cooling film requires computation of flow field in the coolant air supply plenum and duct.

Airfoil Geometry Effect: Surface curvature is a significant factor affecting film cooling performance. Turbine blade leading edge and suction side are featured by the convex curvature while pressure side being a concave surface. The curvature effect was extensively studied by Ito et al. (1978) and Schwarz et al. (1990). It is summarized in their studies that for typical operational blowing ratios, convex surface will generate increased $\eta$, and decreased $\eta$ is generally found for concave surface except at high momentum flux ratios. The surface curvature effect can be explained by the wall normal pressure gradients associated with wall curvature. When the momentum of the jet is less than that of the mainstream, the pressure gradients push the coolant jets towards the wall for convex surfaces on the suction side. Also the inward pressure has a positive effect of spreading the jet wider on the surface, and keeps the jet attached for higher momentum flux ratios. For concave curvature the opposite occurs, i.e. the coolant jets are pushed away from the wall. Waye and Bogard (2000) studied film cooling adiabatic effectiveness for axial and compound angle holes on the suction side of a simulated turbine vane. Some more examples studying the curvature effect can be found in the work of Zhang et al. (2009) and Ladisch et al. (2009).

1.4 Mist/Film Cooling Concept

One potential new cooling technique is mist cooling. The basic idea is to inject small amounts of tiny water droplets (mist) into the cooling air to enhance the cooling performance.
The key mist cooling enhancement mechanism is attributed to the latent heat that droplet evaporation will absorb when moving along the coolant air. The most important feature of mist cooling is its "distributed cooling" characteristics. Each droplet acts as a cooling sink, and it flies a distance before it completely vaporizes. The reduced temperature near the surface due to droplet evaporation near the wall plays a major role in protecting the surface from the hot gas. Direct contacts between water droplets and the wall further take the thermal energy away at a fast pace, which significantly enhance cooling effectiveness. Furthermore, continuous droplets evaporation can last longer and go farther into the downstream region where single-phase air film cooling becomes less effective. The key behind the mist cooling concept is the utilization of the high latent heat of water to improve the heat transfer performance of film cooling. This cooling scheme is an attractive approach to cool turbine blade for the following reasons:

- Mist cooling can provide higher cooling effectiveness. Due to the latent heat of evaporation, the water droplets serve as numerous heat sinks in the mist film flow. This results in a higher effective specific heat for the mist film mixture. Consequently, the mean bulk temperature of the mist film flow will be lower than that of the corresponding single-phase air flow and gives better cooling protection for the blade surface.

- As a result of the higher cooling effectiveness of mist cooling, the amount of coolant air can be reduced. More compressed air can be saved for combustion and the corresponding turbine output power can be increased.

- Mist cooling can provide better blade coverage. Also due to the high latent heat of water, it takes longer time to heat up the mist film mixture comparing with the air only flow. As a result the mist/film coolant travels further downstream on the blade, covering a bigger area of the blade before being heated up and blend into the main hot gas. Thus mist cooling can overcome the short coverage problem of film cooling

- As a result of the better blade coverage of the mist cooling, the number of rows of coolant holes can be reduced. Consequently the blade mechanical integrity can be improved. Also it provides the potential to reduce aerodynamics lost since less jet-main flow reaction is expected due to the smaller number of coolant holes needed.

- Furthermore, mist/film cooling can take advantage of accumulated experiences and research results of the mature and extensively studied film cooling technology. Any
benefit from the introduction of the mist is an addition to existing film cooling performance.

- The similar cooling scheme of mist/steam cooling has already been proved to be effective through both experimental and simulation studies. (Guo et al. 1999, Li et al. 2000, 2001, Dhanasekaran et al. 2010)

It is important to notice that mist cooling is different from spray cooling. Spray cooling consists of employing streams of high concentrations of liquid droplets that are atomized under high momentum and are moving through inertia, whereas mist cooling employs a low concentration of water droplets suspended in a gas stream, with the droplets moving along the gas streamlines via gas-droplet interfacial drag. Spray cooling is usually employed close to the target surface: it can easily flood the surface with liquid layer. Mist cooling, on the other hand, is usually applied away from the target surface and it is meant to cool the surface without wetting the surface.

Mist has been used to enhance heat transfer in gas turbine systems in different ways. Gas turbine inlet air fog cooling (Chaker et. al, 2002) is a common application where the droplets evaporate to lower the compressor air inlet temperature until the relative humidity reaches 100%. In addition, fog overspray is used in industry to provide additional evaporative cooling inside the compressor.

One of the early studies of mist/air heat transfer was done by Takagi and Ogasawara (1974) to investigate mist/air flow and heat transfer inside a vertical rectangular tube. In this study, compressed air was mixed with water droplets generated by an atomizing nozzle before it went through the test section. The average droplet sizes were in the range of 50~150 μm. By measuring the wall temperature, they found that the two-phase heat transfer characteristics were considerably affected by heat flux or wall temperature. The heat transfer coefficient (h) decreased as the wall temperature increased. Besides, the heat transfer coefficient increased as the droplet concentration or the air flow rate increased or as droplet size decreased.

The first experimental study of the mist/air cooling of a highly heated tube was performed by Mori, et al. (1982). The experiment was conducted in a highly heated vertical tube of 1.8 mm ID. Local wall temperatures were measured by thermocouples directly welded to the tube. It was found that the heat transfer along the tube axis could be divided into three typical
regions, namely, liquid film region, dryout region, and gas-phase forced convection region. In
the liquid film region, the heat transfer coefficient is almost ten times higher than that without
mist. In the gas-phase forced convection region, the heat transfer coefficient follows the single-
phase convective heat transfer correlations.

Janssen, et al. (1986) performed another study on mist/air cooling of a very hot tube. The
objective of this study was to identify, based on the viewpoint of heat transfer, the difference
between the following two cases. One case was to inject liquid water to evaporatively cool the
compressed air prior to its contact with the heated test section (inlet thermal equilibrium case).
The other case was to inject cold liquid water into the compressed air with the evaporation of the
water droplets being still in progress when the cooling air was in contact with the heated test
section (inlet thermal non-equilibrium case). The first case was actually a single-phase flow in
the test section. The experimental setup of Janssen, et al. was similar to that of Mori, et al.
Basically, the test section was made of a 304 SS tube with 1.6 mm ID and 1.8 mm OD and 150
mm in length. Water droplets with an estimated Sauter mean diameters on the order of 10 μm
were generated by an air-assisted atomizing nozzle. Experimental results demonstrated that a
mixture of hot air and cold water droplets (inlet thermal non-equilibrium case) would cool a hot
surface more effectively than the same mixture introduced to the same surface after the droplets
have totally evaporated (inlet thermal equilibrium). Considering that the two cases were
maintained at the same inlet conditions, these experimental results actually suggested that the
droplet dynamics play an important role in the heat transfer enhancement in addition to the effect
of latent heat. In the experiment, it was also found that along the test section, there also existed a
liquid film region, a mist (dryout) region and a gas-only convection region. A heat transfer
coefficient 10 times higher than that in the gas only region was also observed in the liquid film
region.

Nazarov et. al. (2009) conducted an experimental work to investigate the heat transfer
processes in cooling surfaces by a pulsed gas-droplet stream. Their application is actually closer
to spray cooling with the surface being flooded with a liquid layer. The experiments were
performed in the regime of evaporation of the liquid precipitated on the surface in the form of
separate drops, rivulets, and a continuously flowing sheet. It has been shown that depending on
the time parameters of the pulse spray, the integral heat transfer can effectively be controlled
over a wide range. They also showed that a concurrent air supply, with the jets, leads to a
significant intensification of the heat transfer between the spray and a vertical obstacle. Recently, Pakhomov et al. [2010] performed a numerical simulation to study the flow structure and heat transfer of impact mist jet with low concentration of droplets (liquid mass flow $\leq 1\%$). Their mathematical model was based on the solution to RANS equations for the two-phase flow in Euler approximation. Their results showed that droplets addition causes a substantial increase of heat transfer rate (several times) compared with one-phase air impact jet.

The real industrial engine application study of mist/air cooling of gas turbine vane blades has been carried out by Nirmalan, et al. (1996). In this study, finely dispersed mist/air flow that impinges on the internal surfaces of turbine airfoils is utilized to produce a very high cooling rate. The droplets are generated by the atomizing nozzle in a plenum (which is located on the top of the vane) and are forced through an impingement tube (which is located inside the vane) onto the vane surface. It has been found that by using mist/air cooling, cooling airflow can be reduced more than 50% to reach the same overall cooling levels of air-only cooled vanes. The experimental results also show that the cooling effectiveness is not uniformly distributed along the vane surface. Some undesirable overcooling occurs when the surface was flooded with liquid droplets. This non-uniformity is speculated to be attributed to undesirable droplet generation and distribution by the atomizing nozzle and to unsatisfactory droplet delivery characteristic through the impingement tube. No specific information was given on the mist/air ratio. (Please check if mist/air ratio is given.)

To explore an innovative approach to cooling future high-temperature gas turbines, a research group led by Professor T. Wang has conducted a series of mist/steam cooling experimental studies. Guo et. al. (2000I) studied the mist/steam flow and heat transfer in a straight tube under highly superheated wall temperatures. It was found that the heat transfer of steam could be significantly improved by adding mist into the main flow. An average enhancement of 100 % was achieved with less than 5 % mist. In an experimental study with a horizontal 180 degree tube bend, Guo et. al (2000II) found that both the outer and the inner walls of the test section exhibited a significant and a similar heat transfer enhancement. The overall cooling enhancement of mist/steam flow increased as the main steam flow increased, but decreased as the wall heat flux increased. The mist/steam experimental results by Guo et al. (2000) showed that the outer wall of 180-degree bend had a better cooling than the inner wall for both steam and mist/steam cases. They revealed that the mist-cooled outer wall had a maximum
cooling at the 45\(^0\) location and speculated that this enhancement was caused by the direct mist impingement on the outer wall due to inertia of water droplets coming from the upstream straight section plus the additional centrifugal force. Their experimental results also showed that the average mist cooling enhancement reached 200-300\% under lower heat flux conditions, but it deteriorated to about 50 \% under higher heat flux conditions. It was concerned then that the mist cooling might not be beneficial under the elevated wall heat flux condition in the real gas turbine operating environment. However, they further increased the Reynolds number to simulate higher Reynolds number in the gas turbine and observed the cooling enhancement returning back to above 100\%. For jet impingement cooling over a flat surface (Li et al., 2003), a 200\% cooling enhancement was shown near the stagnation point by adding 1.5\% mist (in mass). In jet impingement on a concave surface as studied by Li et al., enhancements of 30 to 200\% were achieved within five-slot distance with 0.5\% (weight) mist.

Li and Wang (2005) conducted a numerical simulation of air/mist film cooling. They showed that injecting a small amount of droplets (2\% of the coolant flow rate) could enhance the cooling effectiveness about 30\% – 50\%. The cooling enhancement takes place more strongly in the downstream region, where the single-phase film cooling becomes less effective. Three different holes were used in their study, including a 2-D slot, a round hole, and a fan-shaped diffusion hole. They performed a comprehensive study on the effect of flue gas temperature, blowing angle, blowing ratio, mist injection rate, and droplet size on the cooling effectiveness. Analysis on droplet history (trajectory and size) was undertaken to interpret the mechanisms of droplet dynamics. Li and Wang (2005) further conducted a more fundamental study on investigating the effect of various models on the computational results including the turbulence models, dispersed-phase modeling, different forces models (Saffman, thermophoresis, and Brownian), trajectory tracking model, near-wall grid arrangement, and mist injection scheme. The effects of flow inlet boundary conditions (with/without the air supply plenum), inlet turbulence intensity, and the near-wall grid density on simulation results were also investigated. Using a 2-D slot film cooling simulation with a fixed blowing angle and blowing ratio, they showed that injecting mist of 2\% coolant mass flow rate can increase the adiabatic cooling effectiveness about 45\%. The RNG k-\(\varepsilon\) model, RSM and the standard k-\(\varepsilon\) turbulence model with the enhanced wall treatment produce consistent and reasonable results, while the turbulence dispersion has a significant effect on mist film cooling through the stochastic trajectory.
calculation. The thermophoretic force slightly increases the cooling effectiveness, but the effect of Brownian force and Saffman lift is imperceptible. The cooling performance is affected negatively by the plenum, which alters the velocity profile and turbulence intensity at the jet discharge plane. Both of these studies were conducted at conditions of low Reynolds number, temperature, and pressure. Actually, most other studies discussed above were also conducted at low Reynolds number, temperature, and pressure conditions.

As a continuous effort to develop a realistic mist film cooling scheme, Wang and Li (2008) examined the performance of mist film cooling under gas turbine operational conditions, featured by high pressure (15 atm), velocity (128 m/s), and temperature (1561k). The enhancement of the adiabatic cooling effectiveness was found less attractive than the cases with low pressure, velocity and temperature conditions. However, due to high surface temperature under the GT operating condition, the additional wall temperature reduction could achieve 60K even though the enhancement of adiabatic cooling effectiveness is only 10%. Dhanasekaran and Wang (2012) studied the effect of using mist film cooling for rotating gas turbine blades under lab and elevated (real) operating conditions. Their results showed that the average mist cooling enhancement of about 15% and 35% are achieved on the laboratory and elevated conditions, respectively. This can translate into a significant blade surface temperature reduction of 100-125 K with 10% mist injection at elevated condition. This temperature reduction can be critical to the airfoil life expectancy of gas turbines.

In addition to the effect of latent heat and increased Cp value that contribute to mist cooling enhancement, due to the evaporation of the liquid droplets, there are momentum, heat and mass transfer between the droplets and the air film. These interactions are expected to induce mixing and turbulence inside the main flow and the boundary layer. Two effects are generated: 1) reduced wall temperature; 2) enhanced heat transfer coefficient. The first effect is favorable yielding higher cooling effectiveness while the latter is detrimental to the film cooling's purpose of reducing heat flux into the blade. Investigations are needed to clearly identify both effects for mist/film cooling. And of course, beside the need to better understand the mechanisms of mist/film cooling, some practical questions need to be addressed in order to apply this technology in gas turbine engines, for example where and how to feed the mist into the turbine system.
Zhao and Wang (2012) conducted the first set of experiments under laboratory conditions to prove the concept of mist/film cooling. Their study aims to: a) experimentally prove that the proposed concept of injecting mist into air can enhance film cooling performance, b) investigate the associated multiphase flow physics, and c) achieve a better understanding of the connection between the droplet dynamics and the mist cooling heat transfer characteristics. Mist film cooling performance was evaluated and compared against air-only film cooling in terms of adiabatic film cooling effectiveness and film coverage. They used a row of five circular cylinder holes, injecting at an inclination angle of 30° into the main flow. For the 0.6 blowing ratio cases, the net enhancement in adiabatic film cooling effectiveness reached a maximum 190% locally and 128% overall at the centerline, the cooling coverage increased by 83%, and more uniform surface temperature is achieved. When they increased the blowing ratio from 0.6 to 1.4, both the cooling coverage and net enhancement were reduced to below 60%. They concluded that it is more beneficial to choose a relatively low blowing ratio to keep the coolant film attached to the surface when applying the mist cooling. They also introduced the concept of Film Decay Length (FDL) and proved that it is a useful guideline to quantitatively evaluate the effective cooling coverage and cooling decay rate.

The current work is a continuation of the previous experimental work, conducted by Zhao and Wang (2012). Based on the previous work reviewed, the mist cooling scheme worked very well both experimentally and numerically. The concept was experimentally proved to be valid and the performance was found impressive. The current work is a trial to further assess the scheme performance and find methods to further enhance the performance. Based on the gas turbine industry requirements, film cooling should be effective farther downstream of the injection holes, favorably up to a downstream distance of 100 times the injection hole diameter (i.e X/D=100). This motivates the continuation of the experimental work to assess the performance at these far distances downstream of the injection holes. Also, the fan-shaped hole performance was found to achieve a great enhancement (200%) from the cylindrical holes in the air-only case, as been reviewed earlier. This triggers the motivation to try that geometry change in case of mist. Finally, the issue of transporting mist under real gas turbine conditions is still questionable due to the extreme operating conditions. Is it possible for these tiny water droplets to survive and complete its journey internally in the cooling channels and externally on the blade surfaces? This question needs to be answered for different parts of the engine to be cooled, i.e
vanes, rotors, and secondary cooling channels. As this task is difficult and costly, a cost effective CFD simulation will be conducted.

1.5 Objectives

Motivated by the necessity to find an effective cooling technique for modern extremely high temperature turbines, this research will perform fundamental study of the mist/film cooling. As a step toward verifying the validity and also to achieve better understanding of the mist/film cooling scheme, these specific objectives will be met:

I. Experimentally validate the previous experimental work done by (Zhao and Wang, 2012) of mist/film cooling under lab conditions (low pressure and low temperature).

II. Experimentally verify that mist cooling becomes more effective than air-only film in farther downstream region. In these regions, droplets are hypothesized to fly further and each droplet serves as a distributed energy sink to absorb latent heat. Longer streamwise coverage is always favorable because it implies that the cooling performances decays slower, less streamwise temperature gradient, and, thus, lower thermal stresses. Also, a lower number of film cooling holes rows can be used which will lead to greater integrity of the blade, lower mixing loses (lower entropy generation), and lower machining costs.

III. Experimentally study the effect of cooling hole geometry and operating parameters on performance enhancement. This is motivated by the remarkable performance enhancement of air-only film effectiveness in case of fan-shaped holes, as been discussed in sec 1.3.4.

IV. Numerically investigate the possibility of transporting mist to high pressure turbine component under real gas turbine conditions.

To reach those specific goals, the following tasks will be performed:

1. Conduct a heat transfer experiment for both film cooling and mist cooling to evaluate the cooling performance enhancement using the previous short test section (up to X/D=40) as been used by (Zhao and Wang, 2012). This duplication was necessary to ensure the repeatability and to have a set of reference runs
2. Extend the test section length twice to cover a distance hundred times the cooling hole diameter (i.e. up to X/D=100). This allows the study of mist cooling performance farther downstream the cooling holes as been recommended by gas turbine industry engineers.

3. Study the effect of the test section insulation on the cooling effectiveness enhancement.

4. Manufacture and use a set of fan-shaped (diffusion) cooling holes.

5. Conduct a 3D CFD feasibility study to judge the possibility of transporting water mist to high pressure turbine components under real gas turbine conditions. Frame7FA natural gas turbine engine will be used as an example, and FLUENT/ANSYS will be used as the simulation tool.
CHAPTER TWO
EXPERIMENTAL SETUP

The test rig used in this study was constructed in Energy Conversion and Conservation Center’s Aerothermal Laboratory. In this chapter, a description of the experimental facility is presented along with the instrumentation used. The major part for the facility and instrumentation system were established by Zhao and Wang (2012); some minor retrofits have been made including adding a new fan-shaped hole test plate and extending the test section twice longer.

2.1 Experimental Test Facility

A schematic of the overall experimental facility is shown in Figure 2.1. The experimental facility consists of the wind tunnel system providing main flow, the coolant film system providing the secondary air for film cooling purpose, the atomizing system providing the mist, and the test section where the film cooling holes and thermocouples are instrumented.

2.1.1 Wind Tunnel System (Main Flow)

The wind tunnel system employed in this study was constructed and tested in the ECCC’s (Energy Conversion and Conservation Center) Aerothermal Laboratory in the University of New Orleans. The wind tunnel is an open-circuit, blowing type design, rated to provide 3000 CFM airflow. The room air is first heated by the heating unit installed inside the wind tunnel filter box, and then is drawn into the fan inlet to pass through a diffuser to the flow straightening section, which includes a screen pack and a settling chamber. Two contraction sections are designed and constructed to connect the settling chamber to the test section, speeding up the flow and transitioning the flow to the flow passage matching the dimensions of the test section inlet. With the current setup (two contractions installed), a highest air speed of 40 m/s (130.58 ft/s or 89 mph) can be achieved in the test section. The wind tunnel schematic is shown in Figure 2.2 along with the heating unit. The wind tunnel is retrofitted with a heating unit to heat the air sufficiently before entering the test section. The heating unit consisting of 11 heat guns is placed in the filter box in front of the blower inlet, see Fig. 2.2. The rotating motion of the fan acts
naturally as a good mixer, so that the preheated air from the heat gun mixes well with the main flow, generating a uniformly heated main flow.

Figure 2.1 Schematic of mist cooling experiment apparatus (Zhao and Wang, 2012)

Figure 2.2 Photo of (a) the filter box and (b) the heating unit

Figure 2.2 Photo of (a) the filter box and (b) the heating unit
2.1.2 Mist-Film Generation and Transport

Compressed air from the building compressor is used as the secondary air. A spring-and-piston type flow meter (Omega FLMG-, Range 0-100 SCFM) is used to measure the secondary air flow rate (approximately 30 SCFM). A pressure gauge is installed between the flow control valve and the flow meter to monitor the inlet air pressure. The Compressed air is then cooled in an ice chest (120 Quarts, 38¼” L x 17.38” W x 18.06” H) with a 2 inch thick insulation wall. Air passes through a copper coil (3/8” OD, 1/4” ID, and 6’ long) placed in the ice chest.

A pressure atomizing system, provided by Mee Industry Inc., was adopted in this study to generate mist. The system consists of a water filter, a high pressure pump, and a nozzle. Filtered water is compressed to a pressure of up to 1500 psi by a pump with a motor (Baldor Industrial Motors, 3/4 hp) and then goes through the nozzle to be atomized. The nozzle consists of a stainless steel body with ruby-orifice, an impingement pin and an extended polypropylene filter to avoid trapping particles in the base of the nozzle. High-pressure water reaches the nozzle, shooting a fine liquid jet against an impingement pin resulting in atomization, see Figure 2.3. Discussion of different atomizers can be found in the study of Guo and Wang (Guo, T., et. al., 2000).

Figure 2.3 Impaction pin atomizer (Mee Industries, Inc.)

A mixing chamber is used to blend mist from the atomizer with the coolant air. Detailed design of the mixing chamber is shown in Figure 2.4. Baffling plates are installed in the mixing chamber to enhance air-mist mixing. Both the compressed air injection and the mist-generating
atomizer are located at the bottom of the mixing chamber. The top part of the mixing chamber serves as the blender. A large portion of the mist droplets hit the walls of the mixing chamber or the baffle plates, agglomerating into streaks of water which are flowing down to the bottom of the mixing chamber. The water flowing down to the bottom of the mixing chamber is drained out and re-used in the mist generation. The bottom of the mixing chamber is inclined toward the drainage hole to prevent water accumulating at the bottom plate. Water jamming at the bottom plate needs to be avoided because when the compressed air breaks up the water layer at the mixing chamber bottom (due to water jamming), random sized mist particles are generated which is hard to be controlled.

One technical challenge identified through the preliminary tests is the droplet agglomeration into bigger droplets in the supplying channel immediately below the film cooling holes. These larger droplets could slowly creep out to the test section surface and move along the test surface, serving as unwanted "water" cooling on the test surface. This effect should be minimized. Many different practices have been designed and tested (Zhao and Wang, 2012) to resolve the mist agglomeration problem. The principles for the design process are as follows:

- Preventing the air-mist flow from shooting directly at the film hole inlets.
- Directing the flow to go over multiple baffle surfaces so that agglomeration can take place before reaching the injection plate.
- Contouring the flow field near the injection plate so that right before entering the film holes, the air/mist mixture has a velocity direction roughly parallel to the injection hole internal surface. In this way water droplets landing on the wall of the injection hole supply channel is found to be minimized.

To accomplish the design goal of minimizing water droplets and water film creeping out of the injection hole wall, three stages have been added into the mixing chamber. The first stage is a perforated plate placed 2 inches on top of the atomizers. The purpose of the perforated plate is to

a) block large water droplets/clouds entrained by the strong air jet directly to the injection holes;

b) divide the mixing chamber into sections, preventing recirculation of water droplets;

c) increase the resistance so that the chamber is pressurized, the flow can become
more uniform and the otherwise inertial driven flow directly hitting the film injection hole inlet is avoided.

The second stage is the main flow blocker. The purpose is to channel the mist/film mixture to go over multiple surfaces so that mist agglomeration can happen in an earlier stage before going into the film holes. It is noted that a 1/4 inch gap is left purposely on the front and back side of the mixing chamber so that the return water from the blocker can come back to the drainage sliding down on the walls to avoid being re-entrained by the jet.

The third stage is the side blockers, serving as flow directors and insulator. The flow is guided in parallel to the film hole direction before entering the holes. Also they serve as the insulator to insulate the film injection plate from the bottom.

![Figure 2.4 Structure of the mixing chamber (Zhao and Wang, 2012)](image)

2.1.3 Test Section

The test section was designed for studying the mist cooling scheme performance on a flat plate with cylindrical holes. Transparent acrylic sheet (manufactured by Piedmont Plastics.inc) is used to construct the test section with the wall thickness of 0.354 inches (0.90 cm). The test section is a 4” (width) x 6” (height) x 40” (length) constant area channel with a partially open-top (4” x 30”) design implemented to accommodate the use of an infrared camera to measure surface temperature. The bottom surface is the test surface where thermocouples are mounted to measure the surface temperature. The original experiment of Zhao and Wang (2012) was conducted with a short section flat plate 14 inches in length. The current study is conducted with
a 16 inch extension piece attached to the original section to study the effect of mist cooling at a distance X/D=100 farther downstream of the cooling hole as shown in Fig. 2.5. Per advices from engineers from gas turbine industry for their need to examine film cooling effectiveness much farther downstream, that extension was performed.

A special slide in piece design is implemented to conveniently inter-change the test piece with different film hole geometry without changing other setups. A 2 inch wide trench in the slide-in piece, covered with a metal sheet, is carefully manufactured to accommodate the proper mounting of the thermocouples without breaking the wires during the installation process. The trench is filled with silicon glue to fasten the thermocouple wires and provide insulation. Two types of cooling holes are used in this study. The first consisting of five cylindrical holes with an inclination angle of 30° is employed to inject mist/air film, as shown in Fig. 2.6 (a). The hole is ¼ inch (6.35 mm) in diameter, the hole length is 0.4 inches (10 mm) and p/D is 3.2, where p is the pitch. The second hole geometry used is a row of five Fan-Shaped (diffusion) holes shown in Fig. 2.5 (b). The diffusion hole throat diameter (Dt) is ¼ in, the length is 0.708 inches, the lateral diffusion angle, $\beta = 14^\circ$, the axial inclination angle, $\alpha = 30^\circ$, with the same pitch (P/Dt=3.2) as the cylindrical hole case. Figure 2.6 (b) shows the extended test section used.

![Figure 2.5 Test Section outline and dimensions: (a) the original short section used by Zhao and Wang(2012) (b) the extension piece (current work)](image-url)
2.1.4 PDPA System

 particle sizing is an important part of this program since the particle size and its distribution play an essential role in determining film/mist heat transfer. A large number of particle sizing methods have been developed in the past (a review can be found in the paper by Swithinbank, 1991). A Phase Doppler Particle Analyzer (PDPA) system, first introduced by Bachalo in the 1980's (Bachalo, 1980; Bachalo and Houser, 1984), was used in this study.

 The Phase Doppler Method is based upon the principles of light scattering interferometry. Two collimated, monochromatic, and coherent laser beams are made to cross at a point (measurement point), where they interfere and generate light fringes. A particle moving across

![Figure 2.6 Test pieces with (a) cylindrical holes (b) fan-shaped holes](image-url)
the fringes will reflect light, which is picked up by a receiving lens located at a certain off-axis collection angle. A set (usually 3) of detectors in the receiver is used to capture the signal. The temporal frequency of the signal is used to determine particle velocity and the spatial frequency can be used to calculate particle size.

Based on geometric optics, the relative phase shift for any incidental light passing through a spherical droplet, as shown in Fig. 2.7, can be calculated by the following equation (Bachalo and Houser, 1984):

\[ \phi = 2\alpha (\sin \tau - m \sin \tau') \]  \hspace{1cm} (2.1)

Where \( \phi \) is the relative phase shift between any light passing through the droplet and a theoretical, reference light undergoing no phase shift; \( \alpha \) is the non-dimensional size parameter, \( \pi d/\lambda \), in which \( d \) is the droplet diameter, and \( \lambda \) is the wavelength of the incident light; \( \tau \) is the angle formed by the incident light and the surface tangent of the droplet; \( \tau' \) is the angle formed by the first refracted light and the surface tangent of the droplet; \( m \) is the relative (to air) index of refraction for the droplet.

**Figure 2.7 Dual-beam scatter by a spherical droplet (Bachalo, 1980)**
The measurement volume is the body bounded by the ellipsoidal surface shown in Fig. 2.8, and it corresponds to the surface on which the light intensity of the fringes is $1/e^2$ of the maximum intensity, which occurs at the center of the measurement volume. This width definition is also called the D86 width. If the light intensity profile follows a circular Gaussian profile, when it is integrated down to $1/e^2$ of its peak value, it contains 86% of its total power. Fringe spacing decreases with increasing angle $\kappa$, see Figure 2.8.

Where

$N_{FR}$ = number of fringes  
$V$ = volume of measurement volume  
$d_f$ = fringe spacing  
$d_m$ = diameter of the measurement volume  
$l_m$ = length of the measurement volume  
$f$ = focal length of the lens  
$\lambda$ = laser wavelength

\[
\begin{align*}
D_{e^{-2}} &= \frac{4f\lambda}{\pi D_{e^{-2}}} \\
D_{e^{-2}} &= \frac{d_m}{\cos \kappa} \\
l_m &= \frac{d_{e^{-2}}}{\sin \kappa} \\
V &= \pi d_{e^{-2}}^3 / (6 \cos^2 \kappa \sin \kappa) \\
N_{FR} &= \frac{d_m}{d_f} = \frac{1.27d}{D_{e^{-2}}} \\
\end{align*}
\]

Fringe Spacing

\[
d_f = \frac{\lambda}{2 \sin \kappa} - \frac{f\lambda}{d} - 0.5 \times \frac{f}{d}
\]

Figure 2.8 Measurement Volume dimensions (TSI operations manual, 2006)
It is noted that one distinguished feature of Phase Doppler measurement, comparing with conventional interferometric techniques, is that the change in phase is independent of the incident intensity or scattering amplitudes of light, but is directly proportional to the droplet diameter. **PDPA measures the relative phase shift of the Doppler signal directly as a linear function of the droplet diameter.** As such, PDPA measurements are dependent on the wavelength of the scattered light, which is not easily affected by environmental conditions. This feature gives PDPA measurement a clear advantage over the conventional intensity based measurement method (such as laser Doppler anemometry, LDA) which is sensitive to background noise. Another noted feature of the PDPA system is that its detector is commonly placed off the plane of the transmitting beams at an angle close to $30^\circ$, see Figure 2.9. This is because the intensity of the scattered light, which affects the Signal-to-Noise Ratio (SNR), can vary by several orders of magnitude depending on the receiving angle and the droplet concentration. Generally, within the angle of $0^\circ$-$20^\circ$ off the transmitting axis, light is scattered primarily by diffraction, which contains no information for the PDPA process. At an angle between $20^\circ$-$90^\circ$, which is called the forward-scattering region, the scattered light, mostly by refraction for non-opaque droplets, has the highest SNR, thus the size sensitivity is the best. However, this configuration is difficult to traverse and the optical accessibility is poor because the transmitting and receiving optics are on the opposite side of the flow. For angles between $90^\circ$-$180^\circ$, which is called the back-scattering region, most light will be scattered by reflection, which has an SNR lower than the forward-scattering and more laser power is needed. However, the optical accessibility and traversibility of backward-scattering configuration are better.

Figure 2.10 shows an overview of the PDPA system used in the measurements. The existing PDPA system has a fiberoptic probe with a focal length of 350 mm and a beam waist diameter of 115 μm. The beam spacing for the probe is 50 mm. The focal length of the receiver for the PDPA system is 500 mm.

The laser system is an Argon-Ion type water cooled system with 4 watt maximum power output. The generated monocolor laser beam is split into 3 pairs of lasers with different colors and wave lengths through the beam separator shown in Fig 2.11. The separation of the laser beams occurs in the Bragg cell. Bragg interaction is characterized by the diffraction of a high percentage of the incident light intensity (>80 percent) into the first order beam. Typically, more
than two beams exit the Bragg cell. By adjusting the angle between the cell and the incident beam, the intensity of the first-order output beam can be maximized. By adjusting input signal strength, the intensity of the two beams coming out (zero-order and first-order) can also be equalized. Used in this manner, the cell can be both a frequency shifter and beamsplitter. The major disadvantage of this approach is the very small angle between the two beams (TSI Operations manual, 2006).

These pairs of laser are coupled to the transmitting Fiberoptic through the coupler shown in Fig. 2.12. This coupler is used to steer the laser light through the Fiberoptic. These Fiberoptics transmit the laser light to and from the Probes. Adjusting knobs, shown in Figure 2.12 (b), are used to align the laser beam with the Fiberoptic core axis in x- and y-directions. Z-direction alignment is done using the Focusing Ring; see section 2.1.1 for more details.

![Mie Scattering Theory](image)

**Figure 2.9 Scattered light intensity variation (TSI Operations manual, 2006)**
Figure 2.10  PDPA system overview (a) a schematic of complete PDPA system (b) a schematic of PDPA transmitting and receiving layout and (c) photos of the PDPA system for this study
Figure 2.11 PDPA Fiberlight™ Multicolor Beam Separator (Generator)  (a) Outside look  (b) Inside structure (TSI manual, 2007)
Figure 2.12  PDPA coupler (a) Connection with other parts  (b) adjusting knobs detail (TSI manual, 2007)
2.2 Instrumentation

This section includes the instrumentation for measurements of particles, temperature, flow and pressure.

2.2.1 Particle Measurement

Before taking the mist/air film heat transfer data, the PDPA system was verified against Polymer Latex particles (manufactured by Duke Scientific Corp.) of known-sizes. These particles, with moderately low standard deviations for a given size are microspheres with a density of $1.05 \times 10^3$ kg/m$^3$ and a light refraction index of 1.59 at 590 nm. The verification process was performed by putting the Polymer Latex particles into a container made of the acrylic sheet with the same thickness as the test section wall. By doing so, the plastic window effect on the particle measurement as that would be included during the real test section measurement will be evaluated. The container was filled with water and the particles are suspended in the liquid. A small propeller driven by a mini-motor is immersed in the container. The purpose of the small propeller is to induce particle motion in the water so that a certain number of particles can be captured passing through the measurement volume to generate enough data rate in the PDPA measurement system for statistics purpose. Three different particle sizes were used. Figure 2.13 shows the result of the PDPA measured particle sizes compared with the true particle sizes. It can be seen that in the size range of 2~40 microns, the measured particle sizes from PDPA agree well with the actual particle sizes. It is noted that the acrylic sheet does not have a significant effect in particle size measurements.

![PDPA Calibration Curve](image)

Figure 2.13 Calibration of the PDPA measurement (Zhao and Wang, 2012)
**Droplet Size Measurement**

PDPA is essentially a point measurement system. It measures the droplet size and velocity across the measuring volume during a certain period of time. To obtain the droplet size distribution, the PDPA system is traversed in the test section by means of a specially designed traverse system which synchronizes the movement of the transmitter and the receiver of the PDPA system. It is noted that for this study, the droplet density is a function of both location and time. A great change in the particle density exists within the test section. As a result, the rate of particles being captured by the PDPA per unit time can vary from less than 10 Hz to more than 20,000 Hz. For different data rate, the set up of the PDPA operation parameters and signal filtering criteria need to be adjusted correspondingly to correctly measure the particle size and velocity. Few notes for PDPA system adjustments are summarized as following:

a) It is always suggested to have a rough idea about the size distribution and velocity range of the particles to be measured, which gives a rough estimate of the parameters to be used in the measurements.

b) A reasonable combination of down-mixing voltage and signal filtering range is essential in getting correct velocity data. Checking the history plot for data frequency is helpful in identifying whether the set up is reasonable. Another tip is to check the data rate change corresponding to different set-up. A sharp decrease in data rate usually suggests meaningful data was filtered out incorrectly, and vise versa.

c) Adjusting of PMT voltage is often necessary based on the particle sizes. Higher PMT (600~1000 volts) is needed to capture particle less than 10 μm but it will induce higher noise since reflection signals are amplified as well.

d) It is very important to check the laser lights alignment often. The data rate is quite sensitive to how well the laser light beams are aligned and the alignment can be easily knocked off even by the vibration during the probe-receiver traversing process (Fig. 2.14). When the data rate is really low or there is no data at all after adjusting the receiver location/angles, it is suggested to check the laser alignment.
Mean Droplet Sizes

The most important parameter which represents the droplet characteristics is the mean diameter of the droplets. The mean droplet diameter can be evaluated in various ways. The most commonly used mean diameter is the Arithmetic Mean Diameter ($d_{10}$):

$$d_{10} = \frac{\sum_{i=1}^{n} d_i}{n} \quad (2.2)$$

Where $n$ is the total number of droplets.
Other commonly used mean diameters are the Area Mean Diameter \(d_{20}\), Volume Mean Diameter \(d_{30}\), and the Sauter Mean Diameter \(d_{32}\) which represents the ratio of volume to area:

\[
\begin{align*}
\bar{d}_{20} &= \sqrt{\frac{\sum_{i=1}^{n} d_i^2}{n}}, \\
\bar{d}_{30} &= \sqrt[3]{\frac{\sum_{i=1}^{n} d_i^3}{n}}, \\
\bar{d}_{32} &= \frac{\sum_{i=1}^{n} d_i^3}{\sum_{i=1}^{n} d_i^2}
\end{align*}
\] (2.3)

Obviously, if all the droplets are of the same sizes, then \(d_{10} = d_{20} = d_{30} = d_{32} = d\). For droplets with different sizes, \(d_{10}\), among all the mean diameters, has the smallest value, while \(d_{32}\) has the largest. Generally, \(d_{10}\) represents the diameters of most droplets; \(d_{32}\) represents the diameters of large droplets. For example, if we have 10 spherical droplets with 9 of them having diameters of 1, and 1 having a diameter of 10, then \(d_{10} = 1.9, d_{20} = 3.3, d_{30} = 4.7,\) and \(d_{32} = 9.3\).

### 2.2.2 Temperature Measurement

Two methods of temperature measurements are utilized in this study: thermocouple measurements and infrared thermograph measurement. Generally, a thermocouple has a more accurate measurement capability and yields less uncertainty than the infrared thermograph measurement. However, a thermocouple measurement is essentially a point measurement. The space resolution is limited by the necessity of keeping the surface condition and heat transfer path intact, as well as being limited by the capacity of the thermocouple instrumentation. Also, a great deal of effort is required to install the thermocouples underneath the test piece, especially when the bottom of the test piece is not easily accessible. On the other hand, infrared measurement is easier to accomplish. Moreover, it gives the information of the complete surface (within the view angle) which enables one to generate useful contour plots. This information is essential in evaluating a continuous cooling coverage on the surface. However, infrared measurement suffers from lower accuracy (smallest detectable temperature variation is about 0.1 °C) and high uncertainties because of the varying values of the surface emissivity condition due to uneven surface properties, surface oxidation, and non-uniform surface temperature. In this study, the emissivity changes abruptly if the surface is wet with traveling liquid droplets. A combination of both measurement methods used in this study enables one to conduct a more informative analysis. Since an accurate infrared measurement is more sensitive to the variation
of surface emissivity, the thermocouple measurements also serve as the in-situ calibration standard for the infrared measurements.

The overview of the temperature measurement system is shown in Figure 2.15. All of the temperatures are measured by using E-type (chromel-constantan) 36 gauge (0.01 inch) thermocouples, manufactured by Omega’s Fine Duplex Insulated Thermocouple Wire (TT-E-36-1000). The maximum measuring temperature of this kind of thermocouple is 480 °C. The measured temperatures are monitored by a FLUKE data logger (Model 2250). Two Keithley model 2700 multimeter/data acquisition systems are used to measure the thermocouple voltage readings. The Model 2700 Multimeter/Data Acquisition System is a high-performance, half-rack instrument that combines the functionality and high channel count of a data logger with the accuracy, convenience, and stability of 6-1/2-digit Digital Multimeter. The Model 2700 provides 80 channels or 40 differential channels of multiplexed measurement and control. The two multimeters give a total of 160 channels or 80 differential channels. The Model 2700 reads as low as 0.1 μV and provides a digitizing capability with the equivalent of 22-bit A/D resolution. The highest scanning rate is 65 channels/second and the highest sampling speed is 2000/sec, which is more than sufficient for thermocouple measurements.

The boundary layer flow is important in determining the cooling effectiveness. In order not to disturb the boundary layer flow, the thermocouples are mounted from underneath the test surface (not penetrating). Holes (d=1/32”) are drilled from the bottom of the test plate penetrating 9 mm deep, leaving a thickness of 0.5 mm from the top surface (Fig. 2.16). In order to minimize the uncertainty of the temperature measurement, the thermocouples are carefully mounted in a consistent manner. Since the plate is transparent and also the clearance thickness (between the hole tip and the top surface) is thin, visual examination is conducted to make sure the tips of thermocouple joints have direct impact on the surface leaving a noticeable black dot seen from the top surface. Silicon glue is then used to fill the hole to assure that the thermocouple junctions are well protected and fixed in place.
The temperature measurement system uses in-house designed isothermal box, which is fabricated by 1-inch thickness aluminum blocks (Fig. 2.17). The isothermal box provides a massive body to isolate the thermocouple-copper junctions inside the isothermal box from the ambient temperature fluctuation and therefore reduces the unsteadiness uncertainty of the
experiment. The constant ice melting temperature of the ice bath is used as a reference temperature. The readings from the data acquisition system indicate the relative temperature difference between the thermocouple measurement location and the isothermal box temperature. To determine the absolute temperature at the thermocouple junction, the temperature difference between the isothermal box and ice bath is added on the data acquisition readings. Therefore, the actual temperature of the isothermal box is not important; rather the uniformity and steadiness of the isothermal box temperature provide a system that renders a good quality temperature measurement. A test has been conducted to verify the uniformity as well as the steadiness of the isothermal box temperature. Two measurements at both the center of the box and corner of the box within a 45 min period have been taken. As shown in the plot in Figure 2.18, the temperature inside the isothermal box is very uniform (dT < 0.03 °C within 102.6 mm from center to corner) and the steadiness is estimated as dT/dt = 0.002 °C/minute.

The thermocouples, along with the data logger, were calibrated against a standard Resistance Temperature Device (RTD) system (Zhao and Wang, 2012). Basically, the thermocouple junctions were attached to a copper block and placed in a furnace. The temperature readings from the data logger were compared with those from RTD. Figure 2.19 shows the calibration results. It can be seen that in the calibrated range, temperature measurement from the data logger system agrees very well (± 3 °C) with the RTD measurement. A total number of 94 thermal couples are used in this study, see Figure 2.20. The location and number of thermal couples are shown (top view) in Figure 3.20. X is the streamwise direction and Z is the lateral direction. D is the injection hole diameter (1/4 inch, 0.635cm). Z/D=0 is the center line of the middle hole and Z/D=3.2, 6.4, -3.2, -6.4 are the other four hole center lines.

The infrared camera used is MikroScan 7200, provided by Micron Infrared Inc. Zhao and Wang (2012) compared the temperatures measured by the infrared camera with the temperature measured by the thermocouples. The thermocouple measurements agreed fairly well with the infrared camera measurement (± 0.2°C).
Figure 2.17 Components used in the thermocouple measurement (a), (b) and (c) are internal views of isothermal box setting up steps; (d), (e) and (f) are copper wires and RS232 connectors used to connect isothermal box and data acquisition board; (g), (h) and (i) are data acquisition board and its connectors (Zhao and Wang, 2012).

Figure 2.18 Temperature variations inside the isothermal box (Zhao and Wang, 2012)
Figure 2.19 Thermocouple calibration

Figure 2.20 Thermocouples layout (a) baseline short test section (b) extension piece
2.2.3 Flow Measurement

**Secondary Air Flow Rate:** Compressed air from the building compressor is used as the secondary air. A spring-and-piston type flow meter (Omega FLMG-, Range 0-100 SCFM) is used to measure the secondary air flow rate (approximately 30 SCFM), see Fig. 2.21. A pressure gauge is installed between the flow control valve and the flow meter to monitor the inlet air pressure. The flow rate is cross checked with the measurement at the jets exit. Uncertainty of the coolant flow rate is estimated at 5% with a typical flow rate of 25 SCFM.

![Pressure gauge and flow meter](image)

**Figure 2.21 Pressure gauge and flow meter**

**Mist Mass Flow Rate:** The mist flow rate is acquired by measuring the difference between the water flow rate coming into the atomizer and that returning to the reservoir. Both flow rates are measured in-situ by using the catch-and-weigh method. The uncertainty of the mist flow rate is estimated at 7% to 10% of a typical flow rate of 0.3 gallon/hour depending on
the coolant film flow rate. The relatively high uncertainty is attributed to the nature of droplets agglomeration related unsteadiness inside the mixing chamber.

**Main Flow Velocity and Pressure Measurements**: A Pitot-tube combined with a pressure transducer is used in this study to measure the main flow velocity. The dynamic pressure is due to the movement of the fluid. The static and dynamic pressures together make up the total or stagnation pressure. The stagnation pressure is measured by placing the pitot-static tube (Figure 2.22) directly parallel to the flow and connecting it to a manometer. A manometer with the attached microtector, connected to the pitot-static tube, is used to acquire the pressure difference between the stagnation point and the static port by measuring the height of the water column.

![Figure 2.22 Pitot-Static tube](image)

After using Bernoulli’s equation and solving for the velocity, we have

\[
V = \sqrt{\frac{2g_c}{\rho_{air}} \left( P_{stg} - P_{st} \right)}
\]  

(2.4)

The gravitational constant is required if English units are being utilized, particularly if the unit of mass of the density is given in pound-mass (lbm). However, if SI units are being used the gravitational constant will be absent from the equation thereby reducing to
By applying the hydrostatic equation the pressure difference is derived in the following manner:

$$V = \sqrt{\frac{2(P_{stg} - P_{st})}{\rho_{air}}}$$ (2.5)

where $\Delta h$ is the height of the water column. The vertical measurement $x$ is measured from the centerline of the pitot-static tube to the immediate water interface in either leg of the manometer. For instance, if the manometer is located above the pitot-static tube, $x$ is measured from the centerline to the lowest water interface. However, if the manometer is located above the pitot-static tube, $x$ is measured from the centerline to the highest water interface. In either case, the terms containing $x$ cancel which show that $x$ is arbitrary. After the $x$ terms cancel and rearrangement

$$\Delta P = P_{stg} - P_{st} = \Delta h \left(\rho_{water} - \rho_{air}\right)g$$ (2.7)

In the preceding equation $\rho_{water} \gg \rho_{air}$ therefore, $\rho_{air}$ can be neglected thereby reducing to

$$\Delta P = P_{stg} - P_{st} = \Delta h \rho_{water} g$$ (2.8)

A pressure transducer converts a measured pressure into an analog electrical signal. Although there are various types of pressure transducers, one of the most common ones is the strain-gage base transducer. The conversion of pressure into an electrical signal is achieved by the physical deformation of strain gages that are bonded into the diaphragm of the pressure transducer and wired into a Wheatstone bridge configuration. Pressure applied to the pressure transducer produces a deflection of the diaphragm which introduces strain to the gages. The strain will produce an electrical resistance change proportional to the pressure. The electrical signal is then transferred to a signal conditioner. Before using the pressure transducer, a
calibration process is performed to link the electrical signal \((V)\) to the measured pressure difference \((\text{Psi})\). The calibration curve is shown in Figure 2.23.

![Calibration curve of the pressure transducer](image)

Figure 2.23 Calibration curve of the pressure transducer

2.3 Uncertainty Analysis

Uncertainty analysis, deals with assessing the confidence in the results of a specific experiments. In fact, error creeps into every measurement, from the simplest to the most complex. Recognizing this is the first step of uncertainty analysis. The range of possible errors to different sources of errors is known as the uncertainty, which is usually expressed as a range and a probability. Thus, rather than expressing a mass as "10 kg," it would more correctly be "10 kg, plus or minus 30 g, with a 95% probability." To understand the concept of uncertainty, the following terms are defined and used:

1. Error: it is the actual difference between the true and the observed value.
2. Accuracy: it is the deviation of the observed value from the true value.
3. Precision: it is the degree of agreement between repeated observed values.
4. Systematic error: also called “fixed error” or “bias error” is repetitive and of a fixed value, recurring consistently every time the measurement is made.
5. Random error: also called “precision error” or “stochastic error” is that error which varies from reading to reading.

6. Independent Variables: it is defined by Kline and McClintock (1985), as a basic quantity observed directly in the laboratory as opposed to the result, which is obtained by making corrections to the recorded values of the independent variables. The recorded values of the variables are called “data”, in few cases the result will be the same as the data.

Uncertainty is a possible value that the error might take on in a given measurement. For a single observation, the error, which is the difference between the true value and the measured or observed value (Fig. 2.24), is a certain fixed number and can not be a statistical value. Since the true value is not known in almost all the applications, so the true value of error is not obtainable. Therefore, the uncertainty, or what one think the error might be is the more appropriate approach for evaluating experimental results. The uncertainty value may vary considerably depending on the particular circumstances of the observation. Propagation of uncertainty is the way in which uncertainties in the variables affect the uncertainty in results.

![Illustration of the concept of uncertainty with the statistical distribution of possibility that the true value may fall around the measured value (Moffat, 1982).](image)

**Figure 2.24 Illustration of the concept of uncertainty with the statistical distribution of possibility that the true value may fall around the measured value (Moffat, 1982).**
Moffat (1982) defined three replication levels for uncertainty analysis – zeroth, first and Nth order replications. **Zeroth-order** is considered by the following conditions: time itself is frozen; the display of each instrument is considered to be invariant under replication; the only component of uncertainty at this order is the *interpolation uncertainty*, i.e., the inability of independent human observers to assign the same numerical value to the displayed $X_i$. The values of uncertainty at this level are often assigned as "one-half the smallest scale division" or some similar rule of thumb. This order of uncertainty is denoted $\delta x_{i,0}$ for the $i$th variable.

**First-Order:** At this order, time is the only variable; with the experiment running, the display for each instrument is assumed to vary stochastically about a stationary mean, $X_i$. The first order uncertainty interval includes the *timewise variation of the display* and its *interpolation uncertainty*. The value of uncertainty at this order is denoted $\delta x_{i,1}$ and is larger than $\delta x_{i,0}$ for any real process. No changes in instruments are considered at this level. The value of $\delta x_{i,1}$ can be estimated from a set of repeated observations of the value of $X_i$ with the apparatus operating at its set point. The set of readings should be made during steady-state operation or should be adjusted for any monotonic trend in the mean during the observation period. The intent is to arrive at a valid estimate of the standard deviation of the population of possible values of $X_i$, from which future (single-sample) experimental observations will be taken. A diagnostic sample of 20 to 30 elements allows a confident estimation.

**Nth-Order:** At this order, time and the instrument identities are considered to be variables. For each conceptual replication, each instrument is considered to have been replaced by another of the same type. This makes "instrument identity" a variable, and introduces the uncertainty due to the calibration of the instrument used. The Nth-order uncertainty, $\delta x_{i,N}$, is always larger than the first order uncertainty.

In the current study, an uncertainty analysis is performed to assist in identifying large uncertainty sources and planning for experimental procedure. The method used in this study for the uncertainty analysis is based on the theories of Kline and McClintock (1985) and Moffat (1982) and closely follows the method used by Wang and Simon (1989) and will not be detailed here. The results of the uncertainty analysis are summarized below.

The first step of the uncertainty analysis is to identify the independent primary measurement variables for the data reduction process. For this study, three temperatures are used to evaluate cooling effectiveness: main flow temperature, wall temperature, and coolant
temperature. The second step is to calculate the sensitivity coefficient \( \frac{\partial X_R}{\partial X_i} \) of each component—i.e., the influence of each independent variable on the final result (the heat transfer coefficient in this study) due to a small variation of each independent variable. An Excel spreadsheet is developed based on the data reduction process. The sensitivity coefficient is obtained by subjecting each independent variable to a certain perturbation (±1%) from its nominal value and recomputing the output. The third step is to choose a nominal value \( (X_i) \) and an uncertainty value \( (\delta X_i) \) for each independent variable with a 95% confidence level, and the fourth step is to calculate the composite or total uncertainty using the following equation:

\[
\delta X_R = \left\{ \sum_i \left[ \left( \frac{\partial X_R}{\partial X_i} \right)^2 \delta X_i \right] \right\}^{1/2}
\]

where \( X_i \) is the i-th independent variable and \( X_R \) is the resultant.

The results of the uncertainty analysis vary with the level of replication on which the analysis is based. Zeroth-order (Moffat, 1982) and pretest uncertainty (Wang and Simon, 1989) analyses have been performed during the development stage of the experimental system. The first-order and Nth-order uncertainty analyses have been performed during the data-gathering stage. The zeroth-order analysis is made in the early stage of the experiment when only partial hardware existed. The zeroth-order analysis is helpful in making decisions about the precision required for purchasing new equipment or the adequacy of using the existing equipment. At this stage, the only uncertainties recognized are those associated with the interpolation or round-off error of the proposed instrumentation. For this analysis, the uncertainty value, \( \delta X_i \), is taken to be one-half of the smallest scale division for analog instruments and one-half of the value of an increment of the least significant digit for digital instruments. All calibrations are presumed to be accurate, and the facility and instrumentation are assumed to be steady and free of bias errors.

In the pretest analysis, the instrument imprecision uncertainty discussed earlier and labeled as the zeroth-order contribution is combined with the best estimate of the calibration uncertainties. These calibration uncertainties are estimated based on previous experience, including knowledge of the calibration capability of the laboratory and the best estimate of uncertainties associated with the removal of known bias errors in the calibration.
The first-order analysis is performed after the construction of the experimental system is completed. Basically, the effect of unsteadiness is considered in the first-order analysis in combination with the zeroth-order analysis. The Nth-order uncertainty analysis includes the uncertainties caused by imprecision, unsteadiness, calibration error, and errors in the correction models incorporated to minimize all known bias errors. The Nth-order analysis is calculated by the following equation:

\[
\delta X_{i,N} = \left( \delta X_{i,0} \right)^2 + \left( \delta X_{i,u} \right)^2 + \left( \delta X_{i,c} \right)^2 \right)^{1/2}
\]

(2.10)

Results for the zeroth-order and pretest uncertainty analyses for the adiabatic cooling effectiveness are documented in Table 2.1, and the first-order and Nth-order uncertainty analyses for are documented in Table 2.2.

Table 2.1 Uncertainty Analysis: Zeroth-Order and Pretest Uncertainties

<table>
<thead>
<tr>
<th>Independent variable</th>
<th>Nominal Value</th>
<th>dx_r/dx_i</th>
<th>Uncertainty of Imprecision</th>
<th>Uncertainty of Calibration</th>
<th>Uncertainty of Unsteadiness (film only)</th>
<th>Uncertainty of Unsteadiness (with mist)</th>
<th>Zeroth Order magnitude</th>
<th>Zeroth Order (%)</th>
<th>Pretest Uncertainty magnitude</th>
<th>Pretest uncertainty (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(T_{aw}) (°C)</td>
<td>45.000</td>
<td>0.024</td>
<td>0.017</td>
<td>0.200</td>
<td>0.140</td>
<td>0.200</td>
<td>0.000</td>
<td>0.166</td>
<td>0.201</td>
<td>1.958</td>
</tr>
<tr>
<td>(T_{a}) (°C)</td>
<td>53.000</td>
<td>0.019</td>
<td>0.017</td>
<td>0.200</td>
<td>0.280</td>
<td>0.280</td>
<td>0.000</td>
<td>0.130</td>
<td>0.201</td>
<td>1.536</td>
</tr>
<tr>
<td>(T_{j}) (°C)</td>
<td>12.000</td>
<td>0.005</td>
<td>0.017</td>
<td>0.200</td>
<td>0.325</td>
<td>0.465</td>
<td>0.000</td>
<td>0.033</td>
<td>0.201</td>
<td>0.384</td>
</tr>
<tr>
<td>Total Uncertainty of (\eta) (%)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.213</td>
<td></td>
<td>2.519</td>
</tr>
</tbody>
</table>

Table 2.2 Uncertainty Analysis: First Order and Nth Order Uncertainties

<table>
<thead>
<tr>
<th>Independent variable</th>
<th>First order Uncertainty magnitude (film only)</th>
<th>First order Uncertainty magnitude (mist/film)</th>
<th>First order uncertainty magnitude (film only) (%)</th>
<th>First order uncertainty magnitude (mist/film) (%)</th>
<th>Nth order Uncertainty magnitude (film only)</th>
<th>Nth order uncertainty magnitude (film only) (%)</th>
<th>Nth order Uncertainty magnitude (mist/film)</th>
<th>Nth order uncertainty magnitude (mist/film) (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(T_{aw}) (°C)</td>
<td>0.141</td>
<td>1.376</td>
<td>0.201</td>
<td>1.958</td>
<td>0.245</td>
<td>2.388</td>
<td>0.283</td>
<td>2.764</td>
</tr>
<tr>
<td>(T_{a}) (°C)</td>
<td>0.281</td>
<td>2.147</td>
<td>0.281</td>
<td>2.147</td>
<td>0.345</td>
<td>2.637</td>
<td>0.345</td>
<td>2.637</td>
</tr>
<tr>
<td>(T_{j}) (°C)</td>
<td>0.325</td>
<td>0.623</td>
<td>0.465</td>
<td>0.891</td>
<td>0.382</td>
<td>0.731</td>
<td>0.506</td>
<td>0.970</td>
</tr>
<tr>
<td>Total Uncertainty of (\eta) (%)</td>
<td>2.625</td>
<td>3.040</td>
<td></td>
<td></td>
<td>3.632</td>
<td></td>
<td>3.942</td>
<td></td>
</tr>
</tbody>
</table>
Chapter 3

EXPERIMENTAL RESULTS AND DISCUSSION

As been mentioned earlier, the current work is a continuation of a project aims at investigating the applicability of employing mist cooling technique. In this chapter, the experimental results for mist cooling are presented and discussed for different geometrical and flow parameters. The strategy of the current study is to first obtain the knowledge of: 1) overall heat transfer performance through measurements and 2) droplet behavior through the analysis based on the droplet information measurements; then combine the information of both the overall heat transfer results and the droplet measurements to reach the goal of an in-depth understanding of mist film cooling. Investigation of the droplet behavior and finding its impact on overall heat transfer performance for diffusion holes geometries are highlights of the analysis.

3.1 Flow Conditions

The main flow mean velocity at the inlet is 20.54 m/s with the turbulence intensity of the streamwise component at 3.5%. The mean velocity is measured by a Pitot-static tube with a pressure transducer, and the turbulence intensity is based on the PDPA velocity measurements. Even though smaller droplets would serve better for turbulence measurements, they are not present at the prescribed height for meaningful mean flow measurement because of evaporation caused by the heated main flow. The turbulence intensity value is expected to be undervalued due to the relaxation time and slip velocity of the relatively large droplets, which results in differences between the flow velocity and the droplet velocity.

In order to measure main flow temperature distribution at the inlet, a thermocouple probe is inserted into the test section perpendicular to the bottom plate at X/D = -2. An array of four thermocouples uniformly spaced 2 inches apart is used. The bottom thermocouple is 2 inches (Y/D=8) elevated away from the plate. A measurement is taken every 10 seconds over a total duration of two minutes. The results are documented in Table 3.1. Based on the results, the temperature variation in the Y-direction is estimated as 0.13°C per inch. And the temperature variation in time (unsteadiness) is estimated as ±0.28°C per minute. Similar measurements are repeatedly taken by traversing the probes in the X- and Z-directions (lateral direction) and the
temperature gradient is estimated less than ±0.15°C per inch in Z-direction and 0.14 per inch in X-direction.

**Table 3.1 Main flow temperature (°C) measurement**

<table>
<thead>
<tr>
<th>( t ) (s)</th>
<th>( T,°C ) (Y/D=8)</th>
<th>( T,°C ) (Y/D=16)</th>
<th>( T,°C ) (Y/D=24)</th>
<th>( T,°C ) (Y/D=32)</th>
<th>Space Average</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00</td>
<td>54.01</td>
<td>53.67</td>
<td>53.57</td>
<td>53.72</td>
<td>53.74±0.19</td>
</tr>
<tr>
<td>10.00</td>
<td>54.03</td>
<td>53.67</td>
<td>53.23</td>
<td>53.93</td>
<td>53.71±0.36</td>
</tr>
<tr>
<td>20.00</td>
<td>53.06</td>
<td>53.78</td>
<td>53.22</td>
<td>53.37</td>
<td>53.36±0.31</td>
</tr>
<tr>
<td>30.00</td>
<td>54.24</td>
<td>52.91</td>
<td>53.37</td>
<td>53.87</td>
<td>53.60±0.58</td>
</tr>
<tr>
<td>40.00</td>
<td>53.79</td>
<td>53.50</td>
<td>52.68</td>
<td>53.84</td>
<td>53.45±0.54</td>
</tr>
<tr>
<td>50.00</td>
<td>53.59</td>
<td>53.58</td>
<td>53.34</td>
<td>53.43</td>
<td>53.49±0.12</td>
</tr>
<tr>
<td>60.00</td>
<td>53.82</td>
<td>53.72</td>
<td>53.55</td>
<td>53.59</td>
<td>53.67±0.12</td>
</tr>
<tr>
<td>70.00</td>
<td>53.11</td>
<td>53.47</td>
<td>53.77</td>
<td>53.39</td>
<td>53.44±0.27</td>
</tr>
<tr>
<td>80.00</td>
<td>54.05</td>
<td>53.29</td>
<td>53.46</td>
<td>53.92</td>
<td>53.68±0.36</td>
</tr>
<tr>
<td>90.00</td>
<td>53.40</td>
<td>53.69</td>
<td>53.40</td>
<td>53.17</td>
<td>53.42±0.21</td>
</tr>
<tr>
<td>100.00</td>
<td>54.01</td>
<td>53.65</td>
<td>53.41</td>
<td>53.55</td>
<td>53.66±0.26</td>
</tr>
<tr>
<td>110.00</td>
<td>53.53</td>
<td>53.27</td>
<td>53.47</td>
<td>53.44</td>
<td>53.43±0.11</td>
</tr>
<tr>
<td>120.00</td>
<td>53.34</td>
<td>53.24</td>
<td>53.65</td>
<td>53.69</td>
<td>53.48±0.22</td>
</tr>
<tr>
<td>Time Average</td>
<td>53.69±0.38</td>
<td>53.50±0.25</td>
<td>53.39±0.27</td>
<td>53.61±0.24</td>
<td>53.55±0.28</td>
</tr>
</tbody>
</table>

The approaching velocity profile in this study is estimated using a very thin Pitot tube (0.03 in diameter) measurement. The Single hole Pitot tube is attached to the high pressure tap of a pressure transducer with the other low pressure tap connected to the atmosphere, see Fig. 3.1. With the velocity profile as shown in Fig. 3.1, the displacement thickness (\( \delta^* \)) and the momentum thickness (\( \theta \)) are calculated as follows:

\[
\delta^* = \int_0^\infty \left( 1 - \frac{u}{U} \right) dy = 1.276 \text{ mm} \tag{3.1}
\]

\[
\theta = \int_0^\infty \frac{u}{U} \left( 1 - \frac{u}{U} \right) dy = 0.687 \text{ mm} \tag{3.2}
\]

where \( u \) is the local velocity, and \( U \) is the free stream velocity (20.54 m/s). The shape factor (\( H = \delta^*/\theta \)) is 1.857 based on the measurements. The higher the value of \( H \), the stronger the adverse pressure gradient is. Conventionally, \( H = 2.59 \) (Blasius boundary layer) is typical of laminar flows, while \( H = 1.3 - 1.4 \) is typical of turbulent flows.
The flow conditions under different operating parameters are documented in Table 3.2. Three blowing ratios \([M = \frac{(\rho V)_j}{(\rho V)_g}]\) are employed in this study: 0.66, 1.07 and 1.4. Those are typical values corresponding to real values employed in gas turbine film cooling applications. However, if the effect of density in the real gas turbine environment is considered, these blowing ratios correspond to 1.5, 2 and 2.5, respectively. Efforts have been made to reduce the uncertainties of the measurements, including reducing the effect of variations in environmental condition, main flow speed fluctuations, main flow temperature non-uniformity, and unsteadiness in jet flow speed and mist mass flow rates. The major source of the experimental uncertainty arises from the unsteadiness feature of the mist flow rate. In order to acquire good-quality mist, which would suspend in the air, filters and blockers are employed in the mixing chamber constructed by Zhao and Wang (Zhao and Wang, 2012). Mist agglomeration and the actions of water droplets breaking up processes that may occur on the surface of mixing chamber structures are transient and unsteady in nature. As a result, the unsteadiness of mist flow is identified as the main contribution to wall temperature uncertainty.
Table 3.2 Summary of flow conditions

<table>
<thead>
<tr>
<th>Main Flow</th>
<th>Secondary Flow</th>
</tr>
</thead>
<tbody>
<tr>
<td>U (m/s)</td>
<td>M</td>
</tr>
<tr>
<td>20.54</td>
<td>0.66</td>
</tr>
<tr>
<td>Re_Dh</td>
<td>V (m/s)</td>
</tr>
<tr>
<td>133238</td>
<td>12.05</td>
</tr>
<tr>
<td>T_∞ (°C)</td>
<td>Tj (°C)</td>
</tr>
<tr>
<td>53.55</td>
<td>15.8</td>
</tr>
<tr>
<td>Tu (%)</td>
<td>19.19</td>
</tr>
<tr>
<td>3.5</td>
<td>11.73</td>
</tr>
<tr>
<td></td>
<td>26.52</td>
</tr>
<tr>
<td></td>
<td>9.23</td>
</tr>
</tbody>
</table>

3.2 Heat Transfer Results

A list of the cases studied is presented in Table 3.3. In order to evaluate the validity of mist/air film cooling performance, two groups of heat transfer experiments are carried out. The first group is the air-only film case. In this group of experiments, only the air is injected as the coolant. The wall temperature is measured after steady state is reached. The second group of cases is the air/mist film case, in which mist is added into the air film and injected into the main flow channel. Again, temperature measurements at the same location as in the air-only case are taken after steady state is reached. The temperature measurements are then processed to produce the adiabatically cooling effectiveness by incorporating both the main flow temperature and the jet temperature for each case. Comparisons are then conducted to evaluate the performance of the proposed mist/air film cooling as opposed to traditional air film cooling. Evaluation is made based on the following criteria: local cooling performance, lateral averaged cooling performance, and the cooling effect coverage in both the lateral direction and the streamwise direction.

As been mentioned earlier, the current work is a continuation of a project aims at investigating the applicability of mist cooling technique. The previous experimental work, conducted by Zhao and Wang (Zhao and Wang, 2012), was performed with a short test section covers a distance 40 times the cylindrical hole diameter used, i.e. X/D =40 . The purpose was to qualitatively estimate the applicability of the technique and launch the first set of experimental runs. The results were encouraging and directed a further investigations by (a) extending the test section in length to examine mist cooling effectiveness further downstream to a distance hundred times of the cooling hole diameter, i.e. X/D=100 and (b) changing the film cooling hole from circular to fan-shaped diffusion holes. The extended test section was insulated in a set of runs (case 7-case12), and the fan-shaped diffusion holes in another set of runs (cases13-24), as
shown in Table 3.3. Based on the heat transfer and flow results, only representative cases will be selected for droplet data measurements. Detailed description of these runs and the corresponding results will be discussed in the following sections.

Table 3.3 List of experimental cases. The measurement of droplet behavior is conducted for bold-letter cases.

<table>
<thead>
<tr>
<th>Short Section</th>
<th>Extended Section</th>
<th>Extended Section</th>
<th>Extended Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>(Cylindrical Holes)</td>
<td>(Cylindrical Holes)</td>
<td>(Cylindrical Holes)</td>
<td>(Fan-shaped Holes)</td>
</tr>
<tr>
<td>(Non-insulated Wall)</td>
<td>(Insulated Wall)</td>
<td>(Non-insulated Wall)</td>
<td>(Insulated Wall)</td>
</tr>
<tr>
<td>Case #</td>
<td>Film</td>
<td>M</td>
<td>Case #</td>
</tr>
<tr>
<td>Case1</td>
<td>Air Only</td>
<td>0.6</td>
<td>Case7</td>
</tr>
<tr>
<td>Case2</td>
<td>Air/Mist</td>
<td>0.6</td>
<td>Case8</td>
</tr>
<tr>
<td>Case3</td>
<td>Air Only</td>
<td>1.0</td>
<td>Case9</td>
</tr>
<tr>
<td>Case4</td>
<td>Air/Mist</td>
<td>1.0</td>
<td>Case10</td>
</tr>
<tr>
<td>Case5</td>
<td>Air Only</td>
<td>1.4</td>
<td>Case11</td>
</tr>
<tr>
<td>Case6</td>
<td>Air/Mist</td>
<td>1.4</td>
<td>Case12</td>
</tr>
</tbody>
</table>

Two methods of temperature measurement are utilized in this study: thermocouple measurement and infrared thermograph measurement. Generally, a thermocouple provides a more accurate measurement and yields less uncertainty than the infrared thermograph measurement. However, a thermocouple measurement is essentially a point measurement. The space resolution is limited by the number and locations of thermocouple instrumentation. Also, as shown in the previous chapter, a great deal of effort was required to install the thermocouples underneath the test piece, especially when the bottom of the test piece is not easily accessible. On the other hand, infrared measurement is easier to accomplish. Moreover, it gives the information of the complete surface (within the view angle), which enables one to generate useful contour plots. This information is essential in evaluating a continuous cooling coverage on the surface. However, infrared measurement suffers from lower accuracy (smallest detectable temperature variation is about 0.1°C) and high uncertainties due to the varying values of the surface emissivity condition under the influence of uneven surface properties, surface oxidation, and non-uniform surface radiation losses. In this study, the emissivity changes abruptly if the
surface is wet with traveling liquid droplets. A combination of both measurement methods used in this study enables one to conduct more informative analysis. Since an accurate infrared measurement is more inclined to the variation of surface emissivity, the thermocouple measurements also serve as the in-situ calibration standard for the infrared measurements.

3.2.1 **Short Section with cylindrical holes (non-insulated)**

To satisfy the objectives of the current work, the experiments of the short section manufactured and used by Zhao and Wang (Zhao and Wang, 2012) was duplicated to ensure the repeatability and to have a set of reference runs. The test section length was 40 times the cylindrical hole diameter. This means that it only covers up to X/D =40. In the following paragraphs, the results of the short test section (Cases1-6) are presented for different blowing ratios. Three blowing ratios of 0.66, 1.0, and 1.4 are used in the current study. Heat transfer results will be presented first and then the droplet measurements will be presented in the next section but only for the extended section.

3.2.1.1 **Case 1 and Case 2 (Air-only vs. mist film with M=0.6)**

The cooling effectiveness contour for Case 1 and Case 2 are shown in Fig. 3.2 (a) and (b), respectively. The contour plots are produced based on the infrared camera image following the procedures described above. The adiabatic cooling effectiveness is defined as:

\[ \eta = \frac{(T_{aw} - T_g)}{(T_j - T_g)} \]  

where \( T_g \) is the main flow hot gas temperature, \( T_j \) is the coolant temperature at the cooling jet hole exit, and \( T_{aw} \) is the adiabatic wall temperature. \( \eta \) is an excellent indicator of film cooling performance by comparing the insulated wall surface temperature (\( T_{aw} \)) with the would-be perfect wall temperature, \( T_j \). If the film cooling were perfect, \( \eta = 1 \) and the wall is protected as cold as the cooling jet temperature. \( \eta = 0 \) implies that film is not effective at all: the adiabatic wall temperature is as hot as the main flow temperature. With the definition of cooling effectiveness explained, the blue color in the contour plots indicates low cooling effectiveness, meaning that the blade surface is not protected well against the hot main flow with a high adiabatic wall temperature. On the other hand, the red color indicates that the cooling scheme is effective, resulting in a lower surface temperature.
The contour plots of adiabatic film cooling effectiveness in two hole pitches are shown in Fig 3.2. A symmetric and periodic pattern of cooling effectiveness distribution is generally followed in case of air film flows. The imperfection of symmetry between the two coolant injections is due to the coolant holes’ machining defects or nonuniform cooling air flow distribution through the air supply plenum. For the mist cases, the imperfection of symmetry may be further attributed to the nonuniform distribution of mist between the holes. This will be covered by the end of this chapter when some 3-D aspects of the mist/film layer are discussed.

From Fig. 3.2 (a), the cooling effectiveness decays smoothly as X/D increases. The trace of the coolant coverage is clearly identifiable up to X/D about 20 (\(\eta = 0.2\)). In-between the coolant holes, the surface is not covered well by the coolant, leaving a low cooling effectiveness in this area. Also it is noticed that there is no significant interaction between the neighboring injections. The two injections traces seem to be separated from each other.

After mist is added with water temperature at 50°F, as shown in Fig. 3.2 (b), the cooling effectiveness is noticeably higher. Moreover, the coolant coverage is much better in both the streamwise direction and the lateral direction. Looking at the streamwise direction, the cooling clearly covers up to a distance of X/D beyond 30 (\(\eta = 0.4\)). Also, it is noticed that the cooling coverage in-between holes is much better. The uncovered area is much smaller compared with the air-only film case. This implies that either the coolant spreading is enhanced as mist is added into the coolant or the droplets actually spread wider than the coolant itself to provide wider cooling protection.

There are two key aspects through which the film cooling performance is evaluated: the cooling effectiveness and the film coverage. Film cooling effectiveness has always been evaluated in a quantitative manner using variables like adiabatic film cooling effectiveness, heat transfer coefficients, or net heat flux reduction. However, through the literature studies of film cooling, there is no clearly defined variable to quantitatively evaluate the cooling effect coverage enhancement.
Figure 3.2 Contours of cooling effectiveness for (a) Case 1 (M=0.6, air-only film)  (b) Case 2 (M=0.6, mist/air film)

Figure 3.3 Cooling effectiveness and net enhancement (for Cases 1&2) compared with the experimental data from Goldstein et al. and Rhee et al. for M=0.66 (a) centerline data (b) spanwise averaged data

The cooling effectiveness at the injection hole centerline and spanwise averaged result for the M = 0.6 cases (Case 1 and Case 2) are shown in Fig. 3.3 (a) and (b), respectively. The plots are produced based on the thermocouple measurements. The cooling effectiveness is defined the same as it was in the previous section. To evaluate the cooling enhancement of adding mist into the air film, the net enhancement is plotted on the secondary y-axis on the right-hand side. The net enhancement is defined as follows:

$$\text{Net Enhancement} = \left( \eta_m - \eta \right) / \eta$$  \hspace{1cm} (3.4)

The subscript “m” means mist is added. Without any subscript, it means air-only film is used. From the definition, net enhancement is zero if the mist cooling effectiveness is the same
as the air-only cooling effectiveness. In order to verify the current experimental results, the experimental data from the studies of Goldstein et al. and Rhee et al. are also plotted in the same figure for comparison. It is noted that the running conditions are slightly different from the current study, but the comparison among the data are good. For example, the blowing ratio in both Rhee and Goldstein’s study is 0.5 while a value of 0.6 is adopted in the current study. Also, the inclination angle of the film injection holes for Rhee’s experiment is 35° corresponding to the 30° holes used in the current study. Looking at the spanwise averaged data, results show that Rhee’s data is reasonably close to that of the current study in both the changing trend and the values of the adiabatic film effectiveness, \( \eta \). In the center-line data for the air-only case, a peak value of \( \eta = 0.4 \) is found at around \( X/D = 3 \). The reason for the peak is attributed to the blowing off and reattachment flow patterns of the coolant jet flow. Within the blowing off area (\( X/D < 3 \)), the hot main flow wraps from the side, resulting in a hotter surface temperature and a correspondingly low cooling effectiveness. The cooling effectiveness decays from the peak value of 0.4 at around \( X/D = 3 \) to about 0.14 at \( X/D = 30 \).

Also for the centerline data, after mist is added, the mist cooling effectiveness, \( \eta_m \), is higher than that of the air-only case. The general pattern of the cooling effectiveness distribution of the mist case is similar to that of the air-only case with the peak at about the same location. This implies that adding mist in the air film does not change the general cooling pattern drastically, so that knowledge acquired through air-only film cooling can generally be applied into mist cooling. However, the cooling effectiveness is changed greatly, and so is the coolant coverage. Furthermore, the decaying rate of \( \eta \) for the mist case is slower than that of the air-only case. In other words, the effectiveness of mist/air film lasts longer than the air-only film. This feature is very favorable in considering applying mist cooling in gas turbines because:

1. the well-accumulated knowledge of air film cooling is still applicable in mist/air film cooling;
2. cooling effectiveness is enhanced without changing the cooling pattern, so that favorable cooling hole geometries and arrangements can be kept the same;
3. the liquid droplets in the film provides a more extended film cooling coverage effect than the air film cooling.

62
Therefore, it can be concluded that mist film cooling keeps all the merits of air film cooling while being more effective. Therefore, retrofitting the old air film cooling systems with mist cooling seems attractive.

It is observed that the centerline net cooling effectiveness enhancement, plotted in Fig. 3.3 (a), keeps approximately a constant value of 50% between X/D = 3 and X/D = 7, and then increases to 156 % as X/D increases to 38. This result is supported by the observation that the decaying rate of cooling effectiveness is much slower than that of the air-only case. A similar curve of net enhancement is found for the spanwise averaged plots. It is noted that the net enhancement is slightly lower (9% - 15%) than the centerline plot, which is expected because the spanwise averaged result includes area subject to low film cooling effectiveness.

The distribution pattern of the net cooling effectiveness enhancement reveals an attractive feature of mist cooling – the cooling enhancement actually increases as X/D increases after mist is added. This implies that adding mist in the air film not only increases the cooling effectiveness locally, but also the liquid droplets takes some time to evaporate and extends the cooling coverage longer downstream. Furthermore, mist generates a more uniform surface temperature distribution which is critical for reducing wall thermal stresses (assuming that there are no overcooled spots). From the perspective of application in gas turbine blade cooling, a longer coverage is always favorable because it implies that the cooling performance decays slower, streamwise temperature gradient is lessened, and lower thermal stresses is resulted. Also, lower number of film cooling holes rows could be used which means greater integrity of the blade, lower jet mixing loses, and lower manufacturing costs.

One important notice should be mentioned: results shown in Figs. 3.2 (a) and (b) indicate a cooling enhancement between the holes at X/D = 0 and even at negative values of X/D, i.e. upstream of the cooling holes. This is caused by conduction heat transfer back through the test section wall. In this study, no effort was done to reduce this up-stream conduction because in the real turbine airfoils, up-stream conduction also occurs. So, the cooling effectiveness enhancement presented in this study represents the net enhancement including the conduction effect in the substrate.
3.2.1.2 Case 3 and Case 4 (Air-only vs. mist film with M= 1.0)

When the blowing ratio is increased to 1.0, the result of cooling effectiveness for Case 3 and Case 4 are plotted in Fig. 3.4 (a) and (b), respectively. Comparing with the plots for Case 1 and Case 2, where the blowing ratio was set to 0.6, the cooling effectiveness of Case 3 and Case 4 is apparently lower for both with and without mist cases. Also, the coolant film coverage area seems to be smaller than Cases 1 and 2 in both the streamwise and lateral directions. The un-cooled area between injections is larger than the M = 0.6 cases. Injections from the adjacent holes are further separated from each other resulting in a bigger un-cooled gap. Moreover, it is noticed that the cooling effectiveness of the M = 1.0 cases decays more quickly than the M = 0.6 cases.

Figure 3.4 Contour of cooling effectiveness (a) Case 3 (M=1.0, air-only film) (b) Case 4 (M=1.0, mist/air film)

Figure 3.5 Cooling effectiveness and net enhancement for M =1.0 (for Cases 3&4) (a) Centerline data (b) Spanwise averaged data
After mist is added, as shown in Fig. 3.4 (b), again, significant cooling enhancement is found, especially in the area close to the injection holes. The mist cooling effectiveness is higher, and the effective cooling coverage area is wider in lateral direction in this area than without mist. It is noted that the gap between injections remains almost the same as without adding mist for the area with 5<X/D > 15, indicating that mist in this case with M= 1 is not as effective as in the previous case with M = 0.6.

The effectiveness at the injection hole centerline and spanwise averaged result for the M = 1.0 cases (Case 3 and Case 4) are shown in Fig. 3.5 (a) and (b), respectively. Looking at the centerline cooling effectiveness in Fig. 3.5 (a) for Case 3 (air-only film), a peak value of 0.34 is found around X/D = 3. This peak value is lower than the corresponding peak value of 0.40 for Case 1 in Fig. 3.3a. The cooling effectiveness decays quickly in the area 5 < X/D < 15, from 0.34 to 0.14, and levels out slowly downstream of X/D = 15, eventually drops to 0.12 at X/D = 38. In general, the pattern of cooling effectiveness distribution for Case 3 is similar to that of Case 1, only that the cooling effectiveness values are lower in Case 3. It should be noted that the cooling effectiveness is still enhanced with mist in Case 4 compared to Case 3 for all locations.

To look at this phenomenon from another perspective, the net enhancement plot in Fig. 3.5 (a) is examined. The centerline net enhancement (M=1.0) is almost constant at about 42 % in the region 3< x/d< 9. Then it increases to 75 % (almost doubles) at X/D = 30 and then drops again to 64 % at X/D = 38. This change in the net enhancement is slightly different in trend from that of M = 0.6 case. The obvious difference in trend is that the enhancement in the M=0.6 case is always increasing. However, the net enhancement in the M=1.0 case is constant, increasing, and then decreasing. This behavior in the case of M=1.0 may be caused by a slight lift off in the cooling jets close to the injection holes due to the higher jet momentum, and followed by the reattachment (or “bending back”) of the jet after X/D = 9. In the region very close to the injection area (X/D < 5), the net enhancement of the two cases is very close. The highest cooling enhancement for the M = 1 case appears slightly after the cooling hole and decays more quickly than the M = 0.6 cases. Application of mist under the M = 0.6 condition is apparently superior to doing so for the M = 1.0 condition due to both the higher overall enhancement and the longer coverage. Again, please note that under real gas turbine conditions, the density difference between the hot gas and the coolant could be between 2 -2.5 times, so the effect of M=0.6 in the lab condition could be equivalent of M=1.2 -1.5 in the real gas condition.
3.2.1.3 Case 5 and Case 6 (Air-only vs. mist film with M= 1.4)

The contour of cooling effectiveness for Case 5 (M = 1.4, air-only film) and Case 6 (M = 1.4, air/mist film) are shown in Fig. 3.6 (a) and (b), respectively. Comparing this with the corresponding plots for Cases 1-4, it is first noticed that the film coverage is clearly smaller. The un-cooled gap between injections is bigger, indicating poor film coverage. Similar to the M = 1.0 cases, cooling effectiveness is enhanced in the region close to injection hole area (X/D < 7). The film cooling behavior in Figure 3.6 (a) is expected because the high-momentum coolant jet lifts off from the surface, resulting in low cooling effectiveness and poor cooling coverage. Figure 3.6 (b) shows that adding mist will not significantly improve the cooling effectiveness or coverage, especially in the downstream area away from the coolant injection holes.

![Figure 3.6 Contour of cooling effectiveness (a) Case 5 (M=1.4, air-only film) (b) Case 6 (M=1.4, mist/air film)](image)

![Figure 3.7 Cooling effectiveness and net enhancement for M=1.4 (a) centerline data (b) spanwise averaged data](image)
The cooling effectiveness at the injection hole centerline and the spanwise averaged result for the \( M = 1.4 \) cases (Case 5 and Case 6) are shown in Fig. 3.7 (a) and (b), respectively. Looking at the centerline cooling effectiveness for both Case 5 (\( \eta \)) and Case 6 (\( \eta_m \)), it is noticed that \( \eta_m \) is apparently higher than \( \eta \) at all points. The net enhancement of cooling effectiveness is first decreasing from the value of 51\% at \( X/D = 3 \) to 30 \% at \( X/D = 5 \). It increases again in the region \( 5 < X/D < 30 \) to reach the peak value of 57 \% at \( X/D = 30 \). Then it decreases again to 47.2 \% at \( X/D = 38 \). The pattern of the net enhancement curve is very similar to that of the \( M = 1.0 \) cases, except in the region close to the injection hole where the values are lower for the \( M = 1.4 \) cases. Both the \( M = 1.0 \) and \( M = 1.4 \) cases produce very different net enhancement curves from the \( M = 0.6 \) cases, indicating different mechanism of cooling performance enhancement.

### 3.2.3.4 Atomizing water temperature effect

Atomizing water temperature was managed at 50 °F in all the runs in the current work by adding ice cubes periodically. It is anticipated that lower water temperature will produce higher performance enhancement and longer coverage. This makes sense because lower water temperatures (sub-cooled water) mean higher cooling capacity mist and longer life time of the droplets. Both merits increase the net enhancement in the cooling effectiveness and the film coverage. However, an investigation is performed to quantify the effect of water temperature. Considering the fact that the water source temperature is usually higher than 50°F in the real applications, Case2, originally at 50 °F, is repeated for a water temperature of 70 °F. Figure 3.8(a) shows the centerline effectiveness and net enhancement for the \( M=0.66 \) case for two atomizing water temperatures. It is clear that the net enhancement of 50 °F water temperature case is much better (approximately 35% higher) than 70°F. Figure 3.8 (b) shows the mean droplet diameter distribution at \( X/D=28 \) with the vertical distance from the wall (Y) for two different atomizing water temperatures. There is no clear trend to show the effect of water temperature on the droplet diameter. The contour of the cooling effectiveness is also shown in Fig. 3.9. The area of low \( \eta \)-value between the injections in the 70 °F case is larger compared to the original Case 2 at 50°F. This indicates poor coverage in the lateral direction when water temperature increases.

To conclude this section results, it can be said that as the blowing ratio increases from 0.5 to 1.4, film coverage actually decreases in the centerline of the streamwise direction. This is
understandable since, as the blowing ratio increases, the coolant film has more momentum to be lifted further away from the surface and mix with the hot main flow. As a result, the film coverage over the surface becomes less than the lower blowing ratio cases even though more mass flow of coolant is injected into the flow channel. Among these three blowing ratios, Case 2 with $M = 0.6$ is clearly the most superior in the sense that it reaches the highest film cooling effectiveness and net enhancement as well as the widest film cooling coverage areas. Also, when the coolant film is not detached from the surface, adding mist will significantly improve both the cooling effectiveness and also the film coverage. However, in the blowing off cases, adding mist slightly improves the cooling performance in both the effectiveness and the coverage. Therefore, it is important to bear in mind that when applying mist cooling; choose the low blowing ratios to keep the coolant film attached to the surface to exploit the full benefits of mist cooling.

![Graphs](image)

**(a)** Figure 3.8 Effect of atomizing water temperatures (M=0.66, centerline data) on (a) Cooling effectiveness and net enhancement (b) Diameter distribution at X/D=28

**(b)** Figure 3.9 Contour of cooling effectiveness with atomizing water at 70°F (M=0.66, mist/air)
Regarding atomizing water temperature, the selected temperature of 50°F was proven to give higher values of net enhancement and better lateral coverage. Furthermore, since the main flow temperature in not high in the lab condition (54°C), it is necessary to reduce the water temperature to increase the temperature difference between the coolant and the main stream to enhance the experimental resolution. As expected, lower water temperature is always recommended, so 50°F water will be used in all upcoming heat transfer runs.

3.2.2 Extended section with cylindrical holes (with insulated wall)

From the perspective of application in gas turbine blade cooling, a longer streamwise coverage is always favorable because it implies that the cooling performance decays slower, less streamwise temperature gradient, and, thus, lower thermal stresses. Also, a lower number of film cooling holes rows can be used which will lead to greater integrity of the blade, lower mixing loses (lower entropy generation), and lower machining costs. Per advices from engineers from gas turbine industry for their need to examine film cooling effectiveness much farther downstream, the test section was elongated to reach X/D =100. After initial runs were performed, it was noticed that the test section average wall temperature of the extension part was 3°C different from the old section in the non-cooled case. That variation was thought to be attributed to the different cooling conditions on the bottom wall because it was not insulated. It is necessary to reduce this difference in order to reduce the overall uncertainty in the measurement. When the test section bottom wall was insulated, the temperature difference between the two non-cooled sections reduced to less than 0.5°C. In the next section of this chapter, comparison will be made with cases without insulation to separately identify the effect of wall insulation on the net enhancement of cooling effectiveness.

3.2.2.1 Case 7 and Case 8 (Extended section, insulated, M=0.6, air-only vs. mist)

The cooling effectiveness contour for Case 7 and Case 8 are shown in Fig. 3.10 (a) and (b), respectively. As the test section is lengthened, a single IR image is no longer sufficient to cover the whole length. Therefore, two IR images are taken for each test: one for the old section and the other is for the new extension, as shown in Fig. 3.10.

From Fig. 3.10 (a), the cooling effectiveness decays smoothly as X/D increases. The trace of the coolant coverage is clearly identifiable up to X/D about 37 (η =0.12). In-between
the coolant hole, the surface is not covered well by the coolant, leaving a low cooling effectiveness in the area. Also it is noticed that there is no significant interaction between the injections. The two injections traces seem to be separated from each other.

After mist is added, as shown in Fig. 3.10 (b), the cooling effectiveness is noticeably higher. Moreover, the coolant coverage is better in both the streamwise direction and the lateral direction. Looking at the streamwise direction, the cooling clearly covers up to a distance of $X/D$ beyond 47 ($\eta = 0.3$). Also, it is noticed that the cooling coverage in-between holes is better. The low-$\eta$ area is smaller compared with the air-only case which implies that the coolant spreading is enhanced as mist is added into the coolant. However, the injection traces still seem to be separated from each other producing no significant interaction. The coolant does not have a mechanism to reach that area between the injections.

![Figure 3.10 Contour of cooling effectiveness (a) Case 7 (M=0.66, Insulated, extended section, air-only film) (b) Case 8 (M=0.66, Insulated, extended section, mist film)](image)

The cooling effectiveness at the injection hole centerline and spanwise averaged result for the $M = 0.6$ cases (Case 7 and Case 8) are shown in Fig. 3.11 (a) and (b), respectively. The plots
are produced based on the thermocouple measurements. Please notice that the centerline data and the spanwise averaged data is almost identical in the extended section. This is because the film cooling effectiveness is pretty uniform in the spanwise direction downstream of X/D = 45.

![Graphs showing cooling effectiveness and net enhancement for M=0.66 (Case 7 & Case 8)](image)

**Figure 3.11 Cooling effectiveness and net enhancement for M=0.66 (Case 7 & Case 8) (a) Centerline data (b) Spanwise averaged data**

In the centerline data for the air-only case, Fig. 3.11 (a), a peak value of $\eta = 0.34$ is found at around X/D = 3. The reason for the peak is attributed to the blowing off and reattachment flow patterns of the coolant jet flow. The cooling effectiveness decays from this peak value of 0.34 at X/D = 3 to about 0.07 at X/D = 100. Also for the centerline data, after mist is added, the cooling effectiveness, $\eta_m$, is higher than that of the air-only case at all points. The general pattern of the cooling effectiveness distribution of the mist case is similar to that of the air-only case with the peak at about the same location. This implies that adding mist in the air film does not change the general cooling pattern drastically, so that knowledge acquired through film cooling can generally be applied into mist cooling. However, the cooling effectiveness is changed greatly, and so is the coolant coverage. Furthermore, the decaying rate of $\eta$ for the mist case is slower than that of the air-only case. In other words, the effectiveness of mist/air film lasts longer than the air-only film. This feature is very favorable in considering applying mist cooling in gas turbines because it can be retrofitted in an existing film cooling system while giving the extended film covering, as been discussed after Case 2.
It is observed that the centerline net cooling effectiveness enhancement, plotted in Fig. 3.11(a), keeps approximately a constant value of 60% between X/D = 3 and X/D = 7. Then the net enhancement increases to 174% as X/D increases to 100. This result is supported by the observation that the decaying rate of cooling effectiveness is much slower than that of the air-only case. A similar curve of net enhancement is found for the spanwise averaged plots but with slightly lower values (except for the extension part). Comparing results of Case 8 with that of the short section, Case2, reveals that the extended section gives a higher net enhancement at all points. For example, between X/D=3 and X/D =7, the centerline net enhancement for the short section is 50% compared to 60% for the extended section. Also, at the end of extended section (X/D=100), the centerline net enhancement is 174% compared to the 165% obtained in the short section.

From the previous discussion of the extended section results for the M=0.66, the following conclusions can be obtained:

Film cooling enhancement can be observed in the extended section. As noticed in Fig. 3.11 (a) and (b), the net enhancement of mist cooling increases as X/D increases up to X/D =100. This supports the hypothesis that the effect of mist becomes more important farther downstream because some droplets take longer time to evaporate and they fly farther downstream with each droplet serving as a distributed energy sink. This long-lasting cooling capacity is considered as one of the major merits of mist cooling.

### 3.2.2.2 Case 9 and Case 10 (Extended section, insulated, M=1.0, air-only vs. mist)

Contours of cooling effectiveness for Case 9 and Case 10 are plotted in Fig. 3.12 (a) and (b), respectively. Comparing with the plots for Case 7 and Case 8, where the blowing ratio was set to 0.6, the cooling effectiveness of Case 9 and Case 10 is apparently lower for both with and without mist cases. Also, the coolant film coverage seems to be less than Cases 7 and 8 in both the streamwise and lateral directions. The low-η area between injections is larger than the M = 0.6 cases. Injections from the adjacent holes are further separated from each other, resulting in a larger low-η gap. Moreover, it is noticed that the cooling effectiveness of the M = 1.0 cases decays more quickly than the M = 0.6 cases. It should be noted that the cooling effectiveness is still enhanced after mist is added in Case 10 compared to Case 9 for all locations.
Figure 3.12 Contour of cooling effectiveness (a) Case 9 (M=1.0, insulated, extended section, air-only film) (b) Case 10 (M=1.0, insulated, extended section, mist/air film)

Figure 3.13 Cooling effectiveness and net enhancement for M=1.0 (Case 9 & Case 10) (a) Centerline data (b) Spanwise averaged data

The cooling effectiveness at the injection hole centerline and spanwise averaged result for the M = 1.0 cases (Case 3 and Case 4) are shown in Fig. 3.13 (a) and (b), respectively. Looking at the centerline cooling effectiveness in Fig. 3.13 (a) for Case9 (air-only film), a peak value of
0.34 is found around $X/D = 3$. The cooling effectiveness decays quickly in the area $5 < X/D < 25$, from 0.34 to 0.13, and levels out slowly downstream of $X/D = 25$, eventually dropping to 0.11 at $X/D = 100$. In general, the pattern of cooling effectiveness distribution for Case 9 is similar to that of Case 7, only that the cooling effectiveness values are lower in Case 9. To look at this phenomenon from another perspective, the net enhancement plot in Fig. 3.13 (a) is examined. The net enhancement ($M=1.0$) is around 20% immediately after the injection holes. It decreases to 16% at $X/D=7$, and then increases again to reach the maximum value of 63% at $X/D=30$. This behavior in the case of $M=1.0$ is caused by the slight lift off in the cooling jets close to the injection holes due to the higher jet momentum, and followed by the reattachment of the jet. The net enhancement continues to decrease to reach 50% at $X/D = 100$. The same trend is noticed in Fig. 3.13 (b) for the spanwise averaged data but with slightly lower values.

The highest cooling effectiveness net enhancement for the $M = 1.0$ cases appears after the cooling hole and decays more quickly than the $M = 0.6$ cases. More concentrated cooling in the $M = 1$ cases may induce more thermal stress. Application of mist under the $M = 0.6$ condition is apparently superior to the $M = 1.0$ condition due to both the higher overall cooling enhancement and the much longer coverage.

### 3.2.2.3 Case 11 and Case 12 (Extended section, insulated, $M=1.4$, air-only vs. mist)

The contour of cooling effectiveness for Case 11 ($M = 1.4$, air-only film) and Case 12 ($M = 1.4$, mist film) are shown in Fig. 3.14 (a) and (b), respectively. Comparing this with the corresponding plots for Cases 7-10, it is first noticed that the film coverage is clearly significantly smaller. The low-$\eta$ gap between injections is bigger, indicating poor film coverage. Similar to the $M = 1.0$ cases, cooling effectiveness is enhanced in the region close to injection hole area ($X/D < 7$). Figure 3.14 (a) is expected because, from the experience of air film cooling studies, with a blowing ratio of 1.4 under the current study’s hole geometry, the coolant film will have enough momentum to be detached from the blade surface, resulting in low cooling effectiveness and poor cooling coverage. Fig. 3.14 (b) shows that adding mist can also improve the cooling effectiveness or coverage with higher blowing ratio although the enhancement ratio is not as high as when the film is not lift off in the lower blowing ratio. The maximum cooling effectiveness enhancement reaches about 55% near $X/D=25$ and maintains between 35% and
25% in the extended area starting at X/D=40 to the end. Again, this shows the long-last cooling enhancement of mist film cooling for blowing ratios higher than unity.

Figure 3.14 Contour of cooling effectiveness (a) Case 11 (M=1.0, insulated, extended section, air-only film) (b) Case 12 (M=1.0, insulated, extended section, mist/air film)

Figure 3.15 Cooling effectiveness and net enhancement for M=1.4 (Case 11 & Case 12) (a) Centerline data (b) Spanwise averaged data
The cooling effectiveness at the injection hole centerline and the spanwise averaged result for the M = 1.4 cases (Case 11 and Case 12) are shown in Fig. 3.15 (a) and (b), respectively. As the blowing ratio increases, the coolant film has more momentum to be lifted further away from the surface and mix with the hot main flow. As a result, the film coverage over the surface is less than the lower blowing ratio cases even though more mass flow of coolant is injected into the flow channel. The pattern of the net enhancement curve is very similar to that of the M = 1.0 cases but with lower $\eta$-values except in the region close to the injection hole.) only that the values are lower for the M = 1.4 cases. Both the M = 1.0 and M = 1.4 cases produce different net enhancement curves from the M = 0.6 cases, indicating the different mechanism (most likely in fluid mechanics and droplet dynamics) of cooling performance enhancement between those cases. In conclusion, extending the test section longer than X/D=40 has provided valuable information to prove the hypothesis that mist can notably extend the cooling coverage longer downstream with decent cooling enhancement from 160% for M=0.66 case to 30% for M>1 cases.

3.2.3 Extended section with cylindrical holes (without insulated wall)

In the previous section, the extension of the test section was proven useful to reveal the longer coverage of mist cooling downstream of cylindrical holes. As a necessity to reduce the uncertainty in wall temperature measurement, the test section bottom wall was insulated. The results showed higher net enhancement in cooling effectiveness with an increasing trend up to X/D=100. To isolate the effect of insulation, Cases 7-12 are repeated without insulation and the same procedure is repeated. The data will be presented first without analysis, following by a detailed comparison and discussion between insulated and corresponding non-insulated bottom wall cases.

3.2.3.1 Case 13 and Case 14 (extended section, non-insulated, M=0.6, ai-only vs. mist)

The cooling effectiveness contour for Case 13 and Case 14 are shown in Fig. 3.16 (a) and (b), respectively. From Fig. 3.16 (a), the cooling effectiveness decays smoothly as X/D increases. The coolant coverage and the net enhancement are very similar to Cases 7 and 8 above. The cooling effectiveness at the injection hole centerline and spanwise averaged result for Case 13 and Case 14 (M = 0.6) are shown in Fig. 3.17 (a) and (b), respectively.
Figure 3.16 Contour of cooling effectiveness (a) Case 13 (M=0.66, non-insulated, extended section, air-only film) (b) Case 14 (M=0.66, non-insulated, extended section, mist/air film)

Figure 3.17 Cooling effectiveness and net enhancement for M=0.66 (Case13 & Case14) (a) Centerline data (b) Spanwise averaged data
3.2.3.2 Case 15 and Case 16 (Extended section, non-insulated, M=1.0, air-only vs. mist)

The cooling effectiveness contour for Case 15 and Case 16 are shown in Fig. 3.18 (a) and (b), respectively. These cases are similar to Cases 9 and 10 above with very slight differences. Also, the cooling effectiveness at the injection hole centerline and spanwise averaged result for Case are shown in Fig. 3.19 (a) and (b), respectively.

![Figure 3.18 Contour of cooling effectiveness](image1)

(a)

![Figure 3.18 Contour of cooling effectiveness](image2)

(b)

Figure 3.18 Contour of cooling effectiveness (a) Case 15 (M=1.0, non-insulated, extended section, air-only film) (b) Case 16 (M=1.0, non-insulated, extended section, mist/air film)

![Figure 3.19 Cooling effectiveness and net enhancement](image3)

(a)

![Figure 3.19 Cooling effectiveness and net enhancement](image4)

(b)

Figure 3.19 Cooling effectiveness and net enhancement for M=1.0 (Case15&Case16) (a) Centerline data (b) Spanwise averaged data
3.2.3.3 Case 17 and Case 18 (Extended section, non-insulated, M=1.4, air-only vs. mist)

The cooling effectiveness contour for Case 17 and Case 18 are shown in Fig. 3.20 (a) and (b), respectively. These cases are similar to Cases 11 and 12 above with slight differences. Also, the cooling effectiveness at the injection hole centerline and spanwise averaged result for Case are shown in Fig. 3.21 (a) and (b), respectively.

![Figure 3.20 Contour of cooling effectiveness (a) Case 17(M=1.4, non-insulated, extended section, air-only film) (b) Case 18 (M=1.4, non-insulated, extended section, mist/air film)]

![Figure 3.21 Cooling effectiveness and net enhancement for M=1.4 (Case17& Case18) (a) Centerline data (b) Spanwise averaged data]
3.2.3.4 Effect of the test section insulation (cylindrical holes)

As mentioned in the previous section, the main purpose of adding the insulation to the bottom wall was used to increase the temperature difference between wall and the main stream, and thus, increase the experimental resolution for cooling capacity and reduce the uncertainty in the effectiveness calculations. The effect of adding the insulation to the bottom wall of the test section is presented in Figs (3.22) – (3.24) for different Blowing Ratios. Figure 3.22 (a) and (b) shows the net enhancement in cooling effectiveness at the injection hole centerline and the spanwise averaged results for the M=0.66 cases for both the insulated and the corresponding non-insulated runs. It is clear that the insulated case has higher net enhancements (22 % - 40%). This increase is attributed to the higher wall temperature after insulation. For the M=1.0 cases, Figure 3.23 (a) and (b) show that the difference between the insulated and non-insulated cases are almost identical and are within data uncertainty interval. For the M=1.4 cases, Figure 3.24 (a) and (b) show that up to X/D = 25, there is no differences in cooling enhancement results between the insulated and the non-insulated cases. However, for X/D >25, the non-insulated case gives higher net enhancement in cooling effectiveness. This is because the η-value for M=1.4 case becomes lower than other cases further downstream, the calculation of cooling enhancement gives higher values.

![Figure 3.22](image-url)  

**Figure 3.22 Effect of test section insulation on net enhancement for M=0.6**  
(a) Centerline data (b) Spanwise averaged data
In conclusion, as the insulation is added, the trend of the net enhancement does not change. Only the magnitude of enhancement will either change or remain constant depending on the Blowing ratio. For M=0.66, the mist enhancement notable increases 22%-40 %; for M=1.0, the difference is negligible, and for M=1.4, the mist enhancement decreases slightly.

3.2.4 Extended section with fan-shaped holes (insulated wall)

As being known, optimizing geometrical parameters can lead to more effective film cooling over the surface of the airfoils. One of the most challenging issues has been the attainment of the ideal cooling film, like the one developing from 2-D slots (or trench) which can
uniformly cover the target surface. A geometric configuration that has been found to provide a good approximation of the 2-D ideal case is using a row of fan-shaped film-cooling holes. The literature concerning experimental and numerical results deriving from the analysis of such geometries is extremely broad and detailed. Since the 80’s, and especially from the 90’s, a great amount of publications have dealt with both geometrical and fluid dynamical parameters that might play a role in cooling performances, including hole shaping, orientation, spacing, length to diameter ratio, surface roughness and curvature, blowing and density ratios, turbulence intensity. The benefits of shaped cooling holes over cylindrical ones both for flat-plates and airfoils were comprehensively reviewed by Bunker (2005). For the conventional air film cooling, 200% enhancement in cooling effectiveness was harvested after using fan-shaped holes (compared with the cylindrical ones). The question is whether mist cooling can further augment the film cooling effectiveness over the already significantly enhanced film cooling performance produced by fan-shaped holes? Intuition predicts that the net enhancement in the case of diffusion holes could be lower than that of cylindrical holes because the room for more improvement is reduced with fan-shaped holes – This hypothesis will be tested in this section.

The cost of implementing fan-shaped holes is the increased manufacturing difficulty and cost (four to eight times higher) (Colban et al., 2006). However, the benefits of using fan-shaped holes have apparently outweighed the increased manufacturing cost because the fan-shaped holes have been commonly used in the modern gas turbine airfoils. If mist film cooling can provide significant enhancement of cooling effectiveness, a lower number of rows of fan-shaped holes could be used and therefore, the saving of manufacturing cost could be an attractive justification for implementing mist film cooling.

3.2.4.1 Case 19 and 20 (Fan-shaped holes, insulated, M=0.66, air-only vs. mist)

The contour of cooling effectiveness for Case 19 (M = 0.66, air-only film) and Case 20 (M = 0.66, air/mist film) are shown in Fig. 3.25 (a) and (b), respectively. Comparing this with the corresponding plots for the cylindrical holes Cases 7 and 8, it is first noticed that the film coverage is almost the same in the axial direction. However, the lateral coverage of the film is much better in case of fan-shaped holes as there are no un-cooled gaps between injections. The values of effectiveness are much higher in case of the fan-shaped holes. Fig. 3.25 (b) shows that even with highly augmented cooling effectiveness produced by fan-shaped holes, adding mist
will greatly enhances the effectiveness in both lateral and axial directions. These enhancements are further displaced in Figures 3.26 (a) & (b) in terms of centerline and the spanwise averaged net enhancement, \([(\eta_m - \eta)/\eta]\), for the M=0.66 case, respectively. The net enhancement trend is always increasing downstream of the cooling hole which typically matches the nature of mist evaporation along the droplet trajectory. This confirms the extended coverage capability of the mist cooling technique for extended distances further downstream of the injection holes. For X/D<15, the centerline net enhancement is increasing until it reaches 73 % at X/D =15. For 38<X/D<15, the net enhancement increase rate is small from 73% to 80%. For X/D>38, the enhancement starts to increase rapidly up to 170 % at X/D =100. The same trend is obtained for the spanwise averaged values. Unlike the cylindrical holes, the spanwise average net enhancement is slightly higher than the centerline values for X/D <40., as shown in Fig 3.26 (b), indicating that some local off-center cooling is better than in the centerline. This is attributed to the lateral diffusion of coolant and mist as a result of the lateral geometrical enlargement in the cooling hole. Later in this chapter, lateral measurement of droplet distribution will show more coolant presence between holes than at the cooling hole centerline.

In comparison with the cylindrical hole case, as being expected, the film cooling effectiveness is enhanced in the fan-shaped hole cases as shown in Fig.3.27 (a) and Fig. 3.27 (b) for air only and air/mist cases, respectively ( for a blowing ratio M=0.66). The enhancement produced by the fan-shaped holes in the air-only case is 76 % immediately after the hole and deceases gradually to 3 % at X/D = 100. Also, in the mist case, the fan-shaped produced cooling enhancement has the same trend but with a bit low values, which drops to 0 at X/D = 65.

Mist film cooling performance in case of fan-shaped holes is evaluated and compared against cylindrical holes as shown in Figs 3.28 (a) and (b). The comparison is performed in terms of the net enhancement of the adiabatic film cooling effectiveness in the centerline and spanwise directions, respectively. Performance enhancement with an increasing trend is achieved till X/D=100 for both cylindrical and fan-shaped holes. This matches the nature of mist evaporation along the droplet trajectory downstream of the injection holes. As was hypothesized, employment of mist with fan-shaped holes shows enhancement in adiabatic film cooling effectiveness, but the enhancement percentages are lower than the corresponding cases using the cylindrical holes. Nonetheless, the magnitudes of the mist cooling effectiveness using fan-shaped holes are still much higher than using the cylindrical holes.
Figure 3.25  Contour of cooling effectiveness for $M=0.66$ for fan-shaped (diffusion) holes
(a) Case 19, air-only film  (b) Case 20, mist/air film

Figure 3.26 Cooling effectiveness and net enhancement for $M=0.66$ for fan-shaped (diffusion) holes
(a) Centerline data (b) Spanwise averaged data
Figure 3.27 Cooling effectiveness enhancement produced by fan-shaped holes vs. cylindrical holes for M=0.66 (a) air-only case (b) air/mist

Figure 3.28 Comparison of mist film cooling enhancement between fan-shaped holes and cylindrical holes with M=0.6 (a) centerline data (b) spanwise averaged data

In conclusion, employment of fan-shaped holes is beneficial in achieving a great lateral film coverage and higher film cooling efficiency enhancement. As a great enhancement from the cylindrical holes has been achieved in the air-only case when the fan-shaped hole is employed, employing mist cooling still harvests significant cooling enhancement, especially in the region further away from the jet holes with X/D >40, but the enhancement magnitude is lower than the cylindrical hole cases. This result agrees with what was previously hypothesized.
3.2.4.2 Case 21 and Case 22 (Fan-shaped holes, insulated, M=1.0, air-only vs. mist)

The contour of cooling effectiveness for Case 21 (M = 1.0, air-only film) and Case 22 (M = 1.0, air/mist film) are shown in Fig. 3.29 (a) and (b), respectively. Comparing this with the corresponding plots for the cylindrical holes (Cases 9 & 10), it is first noticed that the film coverage is better in the axial direction. In addition, the lateral coverage of the film is much better in fan-shaped hole cases as there are no low-η gaps between injections. Fig. 3.29 (b) shows that adding mist will greatly enhances the effectiveness in both lateral and axial directions.

Figures 3.30 (a) & (b) show the centerline and the spanwise averaged net enhancement for the M=1.0 case, respectively. Both figures show that the adding mist greatly enhances the cooling effectiveness for the fan-shaped holes. The net enhancement trend is always increasing downstream of the cooling hole. For X/D<25, the centerline net enhancement is increasing rapidly until it reaches 40% at X/D =25. For X/D>25, the net enhancement increase slightly from 40% to 50% at X/D=100. The same trend is obtained for the spanwise averaged values. Again, unlike the cylindrical holes, the spanwise average net enhancement is slightly higher than the centerline values at all points, indicating that some local off-center cooling is better than in the centerline.

In comparison with the cylindrical hole case, as being expected, the film cooling effectiveness is enhanced in the fan-shaped hole cases as shown in Fig.3.31 (a) and Fig. 3.31 (b) for air-only and air/mist cases, respectively ( for a blowing ratio M=1.0). The enhancement produced by fan-shaped holes in the air-only case increases to 135 % at X/D=15, then it decreases gradually to 28 % at X/D = 100. When mist is employed, the cooling effectiveness enhancement produced by the fan-shaped holes has the same trend with slightly higher values.

Mist film cooling performance in case of fan-shaped holes is evaluated and compared against cylindrical holes as shown in Figs 3.32 (a) and (b). The comparison is performed in terms of the net enhancement of the adiabatic film cooling effectiveness in the centerline and spanwise directions, respectively. The mist performance enhancement with an increasing trend is achieved till X/D=100 for both cylindrical and fan-shaped holes. This matches the nature of mist evaporation along the droplet trajectory downstream of the injection holes. As was hypothesized, employment of mist with fan-shaped holes shows enhancement in adiabatic film cooling effectiveness, but the enhancement percentages are slightly lower than the corresponding cases.
using the cylindrical holes (8% lower). Nonetheless, the magnitudes of the mist cooling effectiveness using fan-shaped holes are still much higher than using the cylindrical holes, as shown previously in Figs. 3.31 (a) and (b). In comparison with $M=0.6$, the cooling enhancement is significantly reduced from about 160% to 50% at $X/D=100$.

![Figure 3.29 Contour of cooling effectiveness for $M=1.0$ for Fan-shaped (diffusion) holes (a) Case 3, air-only film (b) Case 4, mist/air film](image)

![Figure 3.30 Cooling effectiveness and net enhancement for $M=1.0$ for fan-shaped (diffusion) holes (a) Centerline data (b) Spanwise averaged data](image)
3.2.4.3 Case 23 and 24 (Fan-shaped holes, insulated, $M=1.4$, air-only vs. mist)

The contour of cooling effectiveness for Case 23 ($M = 1.4$, air-only film) and Case 24 ($M = 1.4$, air/mist film) are shown in Fig. 3.33 (a) and (b), respectively. Comparing this with the corresponding plots for the cylindrical holes Cases 11-12, it is first noticed that the film coverage is much better in both the axial and the lateral directions. Fig. 3.33 (b) shows that adding mist will greatly enhances the effectiveness in both lateral and axial directions. Figures 3.34 (a) & (b) show the centerline and the spanwise averaged net enhancement for the $M=1.4$ case, respectively. Both figures show that the adding mist slightly enhances the cooling effectiveness for the fan-shaped holes. The maximum enhancement obtained is around 18% in the centerline direction and 20% in the spanwise direction. These enhancement magnitudes are significantly lower than those in lower blowing ratio with $M=0.6$. 

Figure 3.31 Cooling effectiveness enhancement produced by fan-shaped holes vs. cylindrical holes for $M=1.0$ (a) air-only case (b) air/mist

Figure 3.32 Comparison of mist film cooling enhancement between fan-shaped holes and cylindrical holes with (a) $M=1.0$, centerline data (b) $M=1.0$, spanwise averaged data
In comparison with the cylindrical hole cases, as being expected, the film cooling effectiveness is enhanced in the fan-shaped hole cases as shown in Fig. 3.35 (a) and Fig. 3.35 (b) for air-only and air/mist cases, respectively (for a blowing ratio M=1.4). The enhancement in the air only case is increasing to reach 250% at X/D=15, then it decreases gradually to 36% at X/D=100. These enhancement magnitudes are much higher than cases with lower blowing ratios of M=0.6 and 1.0. Also, the mist cooling efficiency enhancement has the same trend with slightly lower values.

Mist film cooling performance in case of fan-shaped holes is evaluated and compared against cylindrical holes as shown in Figs 3.36 (a) and (b). As was hypothesized, employment of mist with fan-shaped holes shows enhancement in adiabatic film cooling effectiveness, but the enhancement percentages are slightly lower than the corresponding cases using the cylindrical holes (20-35% lower). Nonetheless, the magnitudes of the mist cooling effectiveness using fan-shaped holes are still much higher than using the cylindrical holes, as shown previously in Figs. 3.35 (a) and (b). In comparison with M=0.6, the cooling enhancement is significantly reduced from about 160% to 40% at X/D=100.

![Figure 3.33 Contour of cooling effectiveness for M=1.4 for Fan-shaped (diffusion) holes](image-url)

(a) Case 23, air-only film  (b) Case 24, mist/air film

89
Figure 3.34 Cooling effectiveness and net enhancement for \( M=1.4 \) for Fan-shaped (diffusion) holes (a) Centerline data (b) Spanwise averaged data

Figure 3.35 Cooling effectiveness enhancement produced by fan-shaped holes vs. cylindrical holes for \( M=1.4 \)  (a) air-only case (b) air/mist

Figure 3.36 Comparison of mist film cooling enhancement between fan-shaped holes and cylindrical holes with (a) \( M=1.4 \), centerline data (b) \( M=1.4 \), spanwise averaged data
3.3 Droplet Measurement Results

To understand the mist cooling heat transfer results presented in the previous section, an understanding of the droplets behavior and their interactions with the flow field and temperature field is essential. Thus, droplet measurement is an integral part of this experimental study of mist cooling. The PDPA system, as described in Chapter Two, is employed in this study to measure the droplet information. The droplet information that is of interest to the current study includes droplet size distribution, droplet velocity, and droplet density. The droplet velocity includes the instantaneous velocity, the averaged velocity, velocity fluctuations in both the streamwise and vertical (perpendicular to the wall) directions.

It must be noted that there are some features associated with the droplet data that requires special attention.

1. The PDPA measurement is essentially a point measurement method. It documents the droplet that passes by the specific point of measurement. Traversing of the laser beams is needed to map out the information in a plane or space to give a more general picture of the whole field. This requires a lot of time and effort to accomplish, and as a result, the spatial resolution is limited to the number of measurement points.

2. It is very important to understand the physics and the nature of the flow field to be able to take sufficient data and correctly interpret the results. It is neither practical nor necessary to measure droplet data at all points in the flow field. There are some places in the test section that present some difficulty to measure droplet data like places near corners and places very close to the test section surface.

3. The droplet signal is random in nature. A certain number of samplings are needed to give meaningful measurements. As a result, all of the measurement data needs to be treated in a statistical manner. Interpretation of the data also requires an understanding of the random nature of the data. For example, the Fourier Transform for the random data is very different from that of a continuous measurement. Some assumptions need to be made to perform the Fourier Transform analysis over scattered and non-periodic data.

3.3.1 Cylindrical holes

The purpose of this section is to study the effect of the cooling holes geometry and operating parameters on droplet data. Then, these droplet data are used to interpret the heat
transfer results obtained in the previous section and used to find some correlations between droplet dynamics and the thermal-flow field. Since the PDPA measurements are extensive, only the results of the extended test section without insulation are shown here for the purpose of discussion. The reasons of choosing non-insulated case are because it is easier to conduct droplet measurements with a clear bottom surface and the effect of insulation to droplet dynamics is negligible. Cylindrical holes geometry with Blowing Ratios of 0.66, 1.0, and 1.4 (i.e., Cases 14, 16, and 18) are studied here. In the next section, the same will be done for the Fan-shaped holes with cases 20, 22, and 24. Then the results will be compared between the cylindrical and fan-shaped holes.

3.3.1.1 Droplet size distribution plots

In order to give more detailed information about the droplet behavior, the average droplet size distribution is plotted together with the droplet data rates in Fig. 3.37 for Case 14 (M=0.66). By putting them together in the same plot, some correlation patterns are expected to be discovered. The average droplet size as a function of the vertical distance from the surface is plotted in Fig. 3.37 for different streamwise locations. On the chart, the X-axis is the vertical distance (Y) from the plate surface and the Y-axis on the left is the average droplet size. The second Y-axis on the right is the Data Rate (HZ). Six streamwise locations, X/D = 1, 13, 28, 48, 64, and 80 are plotted in Fig. 3.37 (a)-(d), respectively.

Looking at the averaged droplets size (D_{10}) distribution for X/D = 1, the first measurement point is 3.75 mm away from the surface. It is noted that below this point, the data rate (and the burst efficiency: Refer to the Burst Monitor in the appendix) is too low to produce a meaningful measurement. This means that there is a very minimal number of droplets present under a certain elevation from the surface. From a real physical point of view, this result may imply that the droplets are actually lifted away from the surface in the region very close to the injection hole (X/D=1). This is understandable because mist droplets are carried into the main flow chamber by the coolant jet. With the help of the initial coolant jet's injection momentum, droplets will be lifted off. The average droplet size is by and large uniform between 8 and 9μm from 4 mm to 6 mm away from the surface. Then, the average droplet size increases quickly, reaching to 22.18 μm at Y = 9 mm. Again, it is noted that beyond Y = 9 mm, the data rate is too low for any meaningful measurements (but the burst efficiency is high: Refer to Burst Monitor in
the appendix). This indicates that the droplets are almost absent beyond \( Y = 9 \text{ mm} \) or they are scattered and not forming a stream that can be measured through the measuring volume. This can be explained by the fact that the momentum of the droplets acquired from the coolant jet is not strong enough to shoot the droplets this high.

Examining Fig. 3.37 (b) at \( X/D = 13 \), the general pattern of \( D_{10} \) is shown to fluctuate between 8 and 10 \( \mu m \). Beyond \( Y = 13 \text{ mm} \), the droplet size increases markedly to reach 22.16 \( \mu m \) at \( Y = 16 \text{ mm} \) from the surface. These large droplets can be interpreted as being related to those large droplets shot high near the hole. Now looking at Fig. 3.37(c) at \( X/D = 28 \), the \( D_{10} \) distribution has few fluctuations and becomes relatively more uniform in a broader region from \( Y = 5 \text{ mm} \) to \( 19 \text{ mm} \). No large increase of droplet diameter is seen beyond \( Y = 18 \text{ mm} \). In Fig. 3.37 (d)-(f), the size (\( D_{10} \)) distribution at \( X/D = 48, 64, \) and \( 80 \) become more uniform within \( Y = 2 – 20 \text{ mm} \) with the average droplet diameter increases from below 10 \( \mu m \) at \( X/D = 28 \) and 48 to above 10 \( \mu m \) at \( X/D = 60 \) and 80. Judging from the reduced data rates at \( X/D = 60 \) and 80, the trend of increased droplet diameter demonstrates that the population of small droplets have evaporated and consumed and only larger droplets survive but with diminished populations.

The droplet data rate distribution will be examined as well, which is expected to help in the interpretation of the data. Again, the droplet data rate is the number of droplets that are captured and accounted per unit time at the measurement location. Since the droplet data rate is affected by both the droplet velocity and the density, in order to simplify the analysis, it is first assumed that the droplet velocity does not significantly affect the droplet data rate. The validity of this assumption will be re-examined later in this chapter. With this assumption made, the droplet data rate can now be interpreted to be directly proportional to the droplet's density. The droplet data rate is plotted on the right Y-axis in Fig. 3.37 (a)-(f) for different \( X/D \) locations. The X-axis is the vertical distance from the plate surface. It is found that a roughly “wedge” shape is observed in all six plots. The “wedge” shape is not as obvious at \( X/D = 64 \) and 80 locations as it is in the other four upstream locations, but it is still recognizable. In general, the droplet data rate increases first and then decreases, with some local exceptions. In a nutshell, the wedge shape of data rate distribution can be translated into an “envelope” of air containing measurable droplets with sufficient population within the flow. This flow “envelope” is characterized by an apex showing the highest droplet density and bounded by the upper and lower boundaries depicted by low data rates. Outside this envelope, the droplet density quickly drops to almost
zero. A first impression is that this envelope might represent the trace of the cooling film jet’s width, but it will be found later that this envelope enclosed the droplets traces and is wider than the cooling film layer.

![Graphs showing distributions of droplet size and data rate at different \( Y \) and \( X/D \) locations for Case 14 (\( M=0.66 \) [Cylindrical Holes, Extended Section])](image)

**Figure 3.37** Distributions of droplet size and data rate at different \( Y \) and \( X/D \) locations for Case 14 (\( M=0.66 \)) [Cylindrical Holes, Extended Section]
In order to further investigate the relationship between the droplet size distribution and coolant stream boundaries, the Turbulence Reynolds Shear Stress ($\overline{uv}$) is plotted on the secondary Y-axis in the same figure of the data rate curve in Fig. (3.38) for $M=0.66$ case (Case 14). The turbulence Reynolds shear stress is expected to be higher in the shear layer between the coolant film layer and the main flow. Looking at these figures, the locations of the peak of the Reynolds shear stress shows reasonable agreement with the location of the peak data rate. The corresponding patterns are circled in each figure. Even though some of the patterns are not clearly identifiable, the corresponding patterns are still discernible in most of the cases. It should be noticed that Reynolds shear stresses values obtained are approximate as it is calculated from the droplet velocity analysis. As droplets have a slip velocity with the main flow, even if it is negligible, the quantitative Reynolds shear stress values may deviate from that calculated from the main flow velocity analysis, but the trend of Reynolds shear stress variation should be similar—thus, the location of where the maximum Reynolds shear stress should be identical. For this reason, only small droplet diameters up to 5 µm are used in the shear stress calculations.

One important observation by considering both the Reynolds shear stress distribution and the data rate together is that the peak location of the droplet data rate is always in the close neighborhood of the location where the shear stress is maximum, except at $X/D = 48$. In other words, the location of the peak of the data rate “envelope” is always close to the upper boundary of the perceived cooling film, where Reynolds shear stress is high. The values of the locations acquired corresponding to the peak of the data rate curve, i.e., the upper boundary of the cooling film layer, are summarized in Table 3.4.

As for the bottom boundary of the coolant film layer, it cannot be easily measured due to the fact that the gap between the coolant film and the test surface is in the scale of millimeters. The droplet density in this area is very low and not enough measurement points are available. Furthermore, examining the shear stress and the data rate distribution curves, presented earlier, indicating some occurrence of local high Reynolds shear stress near the wall. Thus, if the cooling film is not lifted off, but is attached to the wall, Reynolds shear stress will not be a good indicator of the bottom boundary of the cooling film boundary because there will be no such lower boundary. However, rough estimates could be made by defining the lower film boundary at a certain low data rate, say, 2 HZ. Based on that rough estimate, the locations of the lower boundary of the jet stream obtained are presented in Table 3.4.
Figure 3.38 Distributions of data rate and Reynolds shear stress at different Y and X/D locations for Case 14 (M=0.66) [Cylindrical Holes, Extended Section]
Looking at the droplet size distribution curves in Fig. 3.37, the droplet sizes for the X/D = 1 cases are around 7μm. However, the droplet size actually increases to 10 μm downstream from X/D = 48 to 60. Based on this initial observation, this trend of increasing droplet diameter in the downstream direction seems against intuition because as the droplets travel downstream, they should absorb heat, evaporate gradually, and become smaller. The answer can be found in the detailed droplet size distribution histograms presented in Fig. 3.39 through 3.45. Comparing the size distributions and the data rate at different downstream locations, it can be seen that as droplets move downstream of the injection holes, two phenomena are observed: disappearing of small size droplets and decreasing in data rate. Since small size droplets evaporates faster than the bigger droplets, and, as a result, the total number of droplets decreases and the average droplet size weights more towards the bigger droplets.

The distribution of droplet sizes for the Case 14 with M = 0.6 is shown in Figs. 3.39-3.45 at different streamwise locations. Within each figure, the droplet size distribution histogram at different elevations (Y-axis) from the surface is plotted. The X-axis is the droplet size and the Y-axis is the droplet count. Those figures are intended to give the information of the droplet size distribution over the flow field. Droplet size information is critical in understanding the droplets’ behavior and the evaporation process in mist cooling. It has close interactions with the thermal-flow field. Also, the variation of droplet size reveals the heat transfer characteristic of mist cooling.

The droplet data rate is the number of droplets that are captured and counted per unit time at the measurement location. It is noted that not all of the droplet signals are counted in the measurement: only the signals that meet certain threshold criteria. For example, the intensity of the light that is reflected by a droplet is theoretically proportional to its size, i.e. bigger droplets refract higher intensity light while smaller droplets refract lower intensity light. Thus, if a droplet's signal bears very low light intensity, but still appears quite large in size based on the frequency measurement, the signal will be dropped because it is very likely that the light signal is from wall reflection or is simply noise.

The droplet data rate is determined by a combination of two factors: droplet velocity and droplet density. As droplet velocity increases, more droplets pass the measurement volume per unit time, and, as a result, the droplet data rate is higher. Similarly, if the droplet distribution is dense, more droplets will be captured per unit time, which also results in higher data rates.
Figure 3.39 Droplet size distribution of various Y locations at X/D=1, M=0.66 (Case14: cylindrical holes, extended section, with intensity validation)

In the area very close to the injection hole, X/D = 1, as shown in Fig. 3.39, the droplet distribution of different y-elevations is plotted. Notice that the droplet distribution patterns at Y = 4 mm and 5 mm are quite similar. The droplet size distribution is centered at about 8 μm and the biggest droplets are sized at 22 μm. The difference is that the droplet data rate at Y = 5 mm is more than twice that of Y = 4 mm. As the measurement elevates to Y = 8 mm, the droplet distribution changes greatly with the most frequently captured droplet size increasing to about 16.73 μm with the largest droplets sized at 36 μm. As the Y-coordinate further increases to 9 mm, the most frequently captured size increases to 22.18 μm and the biggest droplet can be as large as 45 μm or larger. In the mean time, the droplets smaller than 10 μm only account for less than 5% of the total population. It is also noted that the droplet data rate is very low too; 12 Hz. In summary, at an area very close to the injection hole, X/D = 1, the droplet size distribution moves towards larger sizes away from the surface. The droplet data rate increases first from 4 mm to 8 mm away from the surface, and then decreases to zero at higher Y-elevations.
As X/D further increases (i.e. moving downstream), the droplet size distributions at the same elevations at X/D = 13 are plotted in Fig. 3.40. At Y = 4 mm, compared to the same plot at X/D = 1, the size distribution is very close, with the most frequently captured size being 9 μm with the largest size reaches 25 μm. At Y = 5 mm, the size distribution seems identical to those at Y = 4 mm with the most frequently captured size at 9.93 μm. When the comparison is made between the distributions at Y = 5 mm, 8 mm, 10 mm, and 13 mm the diameter distribution is very similar, with the mean droplet diameter decreasing from 9.93 μm to about 8.06 μm and then increases again to 8.41. Finally, at Y = 15 mm, the cooling jet becomes wider and has a higher elevation with large droplets pushed to the upper end. The droplet size distribution is unique with a Bi-modal droplet size distribution, which is caused by the existence of large number of small size droplets along with the large size droplets, as shown in Fig. 3.41 (a). It is clear that the arithmetic mean diameter, D_{10}, increases from 9.93 μm at Y= 15 mm to 22.16 μm at Y=16 mm. That sudden increase in D_{10} is a result of the combined effect of the droplet interactions with the main flow and temperature fields. For the first while, it is mind boggling trying to explain why the droplet diameter increases in such a short distance. By looking carefully to Fig 3.41 (a) and (b) one can see that the number of small size droplets (~10 μm) decreases by 50% while the number of bigger size droplets (~15 μm) slightly increases. This in turn increases the D_{10} and explains the mathematical reason (not yet the physical reason) for that increase. Before accept this mathematical explanation, it needs to further examine whether or not the selection of the lower and upper bounds of light intensity filter could contributes to this bi-modal distribution and whether or not the intensity distribution can verify the bi-modal histogram. To this end, on the right side of the same figure, the light intensity reflected from the droplets is plotted vs. the diameter for both cases. This Intensity Validation Graph is an important tool in optimizing the size data. There is a relatively sharp upper bound on the intensity for any given droplet size. This bound should follow a roughly diameter squared relationship. The Intensity Validation plot is used to help selecting the PMT voltage. The goal is to pick a PMT voltage so that larger droplets have intensities that are very near saturation (intensity of 1000). If too high of a voltage is picked, even at small droplet sizes intensities will reach saturation. The role of thumb is to select a voltage that gives a saturation intensity of 1000 at a droplet diameter equals to (D_{max}/3), where D_{max} is the maximum measurable diameter under the current optical settings. This helps provide a rough guide to a range of voltages that will produce the best data. It can be clearly
seen that in Fig. 3.41 (b) that sufficient number of droplets have intensities between the upper and lower voltage bounds and two areas with concentrated intensities around 10 μm and 25 μm. This implies that the bi-modal droplet distribution is valid and not caused by the uncertainty in setting the upper and lower voltage bounds.

After confirming that Intensity Validation Graph doesn't falsely contribute to the bi-modal droplet distribution, the physics that leads to this bi-modal droplet distribution is explained below. Due to the long distance that the droplets have traveled through in the mixing chamber, it is believed that the droplets have reached their terminal speed (i.e. the droplets maintain a constant slip velocity with the coolant air.) before entering the film hole. As a result, bigger droplets will have higher initial momentum shooting into the main chamber because the droplets are accelerated though the film hole channel from the coolant supply plenum, which is pressurized by the compressed air source. Bigger droplets have higher mass, and, thus, higher momentum. Therefore, the droplets that are able to acquire enough momentum to penetrate through the coolant film must be “big” droplets. This is supported by the droplet size distribution near the injection holes in the first streamwise location (X/D = 1). Thus, combining the information from the size curves and the data rate curves, and the droplet and flow physics, it is concluded that the location where the data rate starts to decrease is close to where the droplet size begins to increase as the upper boundary of the film stream (between Y=8 and 9 mm in Fig. 3.39 and between Y= 13 and 15 mm in Fig. 3.40). This increase in the droplet mean diameter should not be confused with the increased evaporation rate at the film layer edge. Small size droplets evaporate quickly which increases the population of bigger size droplets downstream. This in turn increases the mean size of the droplets. As been discussed in the previous section, the initial droplet size distribution near the injection hole is determined mainly by fluid mechanics rather than heat transfer. The bigger the droplets are, the higher their momentums are, and, in turn, the further up those big droplets can travel to. This explains why the droplet size increases as Y elevates and also explains the existence of the bi-modal size distribution at these higher elevations. This means that the larger droplets observed near the cooling film upper boundary at Y=15 mm and X/D=13 (Fig. 3.40) is associated with the group of larger droplets from outside the cooling film at Y= 9 mm and X/D=1.
Figure 3.40 Droplet size distribution of various Y locations at X/D=13 (cylindrical holes, extended section, M=0.66)
Figure 3.41 Clarification of the Bi-Modal size distribution through laser light intensity strength maps at M= 0.66 at X/D =13 (a) Y=15 mm (b) Y=16 mm

In the region far away from the injection hole, X/D=28, shown in Fig. 3.42, the droplet size distributions at Y = 4 mm is similar to that at X/D=13 but with lower D_{10}. The decrease in D_{10} is attributed to the reduction of large size droplet numbers due to partial evaporation in streamwise direction. The size distributions at Y = 6, 8, and 10 is almost identical with D_{10} around 9 \mu m. This increase in D_{10} from 6.8 \mu m at Y = 4 mm to 9 \mu m further away from the wall is attributed to the full evaporation, and thus disappearing of small size droplets after contact with hot main flow, as can be seen from the droplet distributions. What happens is a change of the proportions of big and small size droplets, which leads to the final D_{10} distributions shown in Fig. 3.42. The data rate increases at Y=8 mm and then decreases again. Finally, the droplets at Y=15 mm has a slight bi-modal diameter distribution similar to that obtained at Y=13 mm at the X/D=13 streamwise location. This bi-modal distribution is a result of the existence of the large droplets flying at the same elevation at X/D=13. These large droplets are those possess higher
initial momentum outside the cooling film upper boundary near the cooling hole, as been
discussed earlier.

Figure 3.42 Droplet size distribution of various Y locations at X/D=28 (Cylindrical Holes, Extended
Section, M=0.66)
As X/D further increases (i.e., moving downstream), the droplet size distributions at X/D = 48 are plotted in Fig. 3.43. At Y = 5 mm, compared to the same plot at X/D = 28 (Fig. 3.42), the size distribution is almost identical, with the most frequently captured size being 8.9 μm with the largest size reaching 20 μm. At elevations Y = 10, 15, and 20 mm, the size distribution seems identical to those at Y = 5 mm with the most frequently captured size being 8.71, 8.38, and 9.17 μm, respectively. At X/D = 48, the cooling jet has expanded wider, so droplets present at higher elevation at Y = 30 mm and 40 mm, where no meaningful droplet information was collected earlier upstream at X/D=28. Again, larger droplet mean diameters of 10.17 μm and 11.7 μm are obtained away from the wall at Y = 30 and 40 mm is interpreted to be caused by vanishing of small droplets. The mean droplet diameter maintains constant around 8.5 μm in the area between 3 mm and 15 mm from the wall and then increases to 11.7 μm at Y = 40 mm.

As X/D further increases, the droplet size distributions at X/D = 64 are plotted in Fig. 3.44. At Y = 5 mm, compared to the same plot at X/D = 48, the size distribution is almost identical, with the most frequently captured size being 10.44 μm with the largest size reaching 22 μm. At elevations Y = 10, 15, 20, 30 and 40 mm, the size distribution seems identical to those at Y = 5 mm with the most frequently captured size being 13.18, 14.12, 12.04, and 11.89 μm, respectively. Unlike the previous X/D=48 location, the mean droplet diameter increases from 10.44 μm to about 14.12 μm and then decreases again to 11.7 μm. This final decrease in the droplet mean diameter indicates effective consumption of larger droplets near the upper cooling film, meaning very few larger droplets existing away from the wall near upper boundary of the cooling film.

As X/D further increases, the droplet size distributions at X/D = 80 are plotted in Fig. 3.45. At Y = 5 mm, compared to the same plot at X/D = 64, the size distribution is almost identical, with the most frequently captured size being bigger at 12.54μm with the largest size reaching 25 μm. At elevations Y = 10, 15, 20, 30 and 27 mm, the size distribution seems similar to those at Y = 5 mm with the most frequently captured size being 12.91, 14.17, 11.94, 9.21 and 8.99 μm, respectively. Unlike previous locations, the mean diameter decreases rapidly beyond after Y=20 mm. Again, this implies a gradual depletion of larger droplets near the upper cooling film, meaning almost of large droplets are all evaporating into small droplets away from the wall.
Figure 3.43 Droplet size distribution of various Y locations at X/D=48 (Cylindrical Holes, Extended Section, M=0.66)
Figure 3.44 Droplet size distribution of various Y locations at X/D=64 (Cylindrical Holes, Extended Section, M=0.66)
Figure 3.45 Droplet size distribution of various Y locations at X/D=80 (Cylindrical Holes, Extended Section, M=0.66)
3.3.1.2 Effect of blowing ratio, M

For other blowing ratios M=1.0 and M=1.4, the same size figures are plotted in Fig. 3.46 and Fig. 3.47(a)-(f). Generally, similar D_{10} size distribution and data rate shapes are obtained with slight differences about the locations of maximum data rate points, or the upper boundary location, as summarized in Table 3.4. Based on the previous analysis, the “envelope” is wider in Y direction at all streamwise locations for these two higher blowing ratios than in the M=0.66 case. This reflects the lift-off jet condition to push the droplets to higher elevations at higher blowing ratios cases. Again, note that the equivalent blowing ratios for real gas environment when large density difference presents between the cooling flow and the hot gas are 1.5, 2, and 2.5, respectively.

In order to further investigate the relationship between the droplet size distributions and cooling film boundaries, the Turbulence Reynolds Shear Stress (\overline{uv}) is plotted on the secondary Y-axis in the same figure of the size distribution curve in Figs. (3.48)- (3.49) for blowing ratios of 1.0 and 1.4, respectively. Results similar to the M=0.66 case is obtained suggesting a reasonable agreement between the locations of peak data rate and maximum shear stress, where the cooling film's upper boundary is proposed. Since the cooling film is lifted off in the two higher blowing ratio cases, the lower boundary layer could be proposed at the location where a local high Reynolds shear stress occurs. As can be seen in Figs. 3.48 and 49, more definite local high values of Reynolds shear stress occur than in the low blowing case shown in Fig. 3.38. Nonetheless, it is still more certain to use the location of 2 Hz data rate as the reference to identify the lower boundary of the cooling film than using the Reynolds shear stress value. Both estimated upper and lower cooling film boundaries are listed in Table 3.4.

In order to simplify the analysis, it was first assumed that the droplet velocity does not significantly affect the droplet data rate. With this assumption made, the droplet data rate was interpreted to be directly proportional to the droplet's density. Now, the validity of this assumption is examined. The average droplet velocity (V) is plotted on the secondary Y-axis in the same figure of the size distribution curve for different Blowing Ratios in Figs. (3.50)-(3.52). Within the envelope, in the circled Y-range, the change in most of the velocities is less than ± 20%, with few locations reaching ± 30%. The effect of this velocity variation on data rate is approximately linearly. Therefore, the data rate value may vary ± 20% accordingly, but the data rate distribution curves in Figs. 3.46 and 47 will not change much.
Figure 3.46 Distributions of droplet size and data rate at different Y and X/D locations for Case 14 (M=1.0) [Cylindrical Holes, Extended Section]
Figure 3.47 Distributions of droplet size and data rate at different Y and X/D locations for Case 14 (M=1.4) [Cylindrical Holes, Extended Section]
Figure 3.48  Distributions of data rate and Reynolds shear stress at different Y and X/D locations for the Cylindrical holes case with M=1.0.
Figure 3.49  Distributions of data rate and Reynolds shear stress at different $Y$ and $X/D$ locations for the Cylindrical holes case with $M=1.4$. 
Figure 3.50 Distribution of droplet size and the droplets mean velocity at different Y and X/D locations at M= 0.66 with Cylindrical holes.
Figure 3.51 Distribution of droplet size and the droplets mean velocity at different Y and X/D locations at M= 1.0 with Cylindrical holes.
Figure 3.52 Distribution of droplet size and the droplets mean velocity at different Y and X/D locations at M= 1.4 with Cylindrical holes.
3.3.1.3 The Proposed coolant air/droplet spreading profile in the cooling film

The physics of the aforementioned observations is summarized below:

1. The coolant film, mixed with the mist droplets, keeps its own identity traveling in the main flow channel in the streamwise direction. The coolant film with mist travels as a layer with finite thickness, which is discernible through the combined information of both the the data rate curves and the Reynolds shear stress ($\overline{uv}$).

2. The location of maximum Reynolds shear stress ($\overline{uv}$) indicates the upper boundary of the coolant film layer. This upper boundary can also be identified by the location of the peak value from the data rate curve. A reasonable agreement was obtained between the two locations.

3. The wedge shape of the data rate distribution curve represents the region that "envelope" the droplets. Since the boundaries of the "envelope" are not clear because they gradually fade away, the lowest data rate approximately at 2 Hz are defined as the envelope's boundaries.

4. The region enclosed by the “envelope” (or droplet layer), as shown by the data rate curve, is the range where the droplets can reach and be statistically measured. It is noted that this envelope size is larger than the film layer width. This is because large droplets penetrate through the air coolant film layer and travel further into the main flow. This phenomenon was also predicted in CFD simulations by the studies of Li and Wang (2007, 2008).

Table 3.4  Y location for coolant film upper (peak data rate) and lower (2 HZ data rate) boundaries acquired from the data rate curve for different Blowing Ratios with cylindrical holes.

<table>
<thead>
<tr>
<th>X/D</th>
<th>M = 0.66 Upper (mm)</th>
<th>Lower (mm)</th>
<th>M=1.0 Upper (mm)</th>
<th>Lower (mm)</th>
<th>M=1.4 Upper (mm)</th>
<th>Lower (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>4</td>
<td>2</td>
<td>1</td>
<td>7</td>
<td>4</td>
</tr>
<tr>
<td>13</td>
<td>13</td>
<td>9</td>
<td>4.5</td>
<td>13</td>
<td>13</td>
<td>5</td>
</tr>
<tr>
<td>28</td>
<td>28</td>
<td>9</td>
<td>3</td>
<td>28</td>
<td>9</td>
<td>3</td>
</tr>
<tr>
<td>48</td>
<td>48</td>
<td>8</td>
<td>3</td>
<td>48</td>
<td>10</td>
<td>3</td>
</tr>
<tr>
<td>64</td>
<td>64</td>
<td>8</td>
<td>2</td>
<td>64</td>
<td>10</td>
<td>3.5</td>
</tr>
<tr>
<td>80</td>
<td>80</td>
<td>9</td>
<td>4</td>
<td>80</td>
<td>16</td>
<td>7</td>
</tr>
</tbody>
</table>

116
3.3.1.4 Linking the droplet data with the heat transfer data

Now that the droplet dynamics has been thoroughly analyzed, efforts will be dedicated to finding the connection between the droplets behavior and the adiabatic mist film cooling effectiveness occurring on the test wall surface.

Based on the previous analysis, the shape of the coolant film layer is outlined in Fig. 3.53 by plotting its upper and lower boundaries from the information summarized in Table 3.4. It is noted that the plot actually represents the 2-D film layer's center-plane on which the PDPA measurements were made. In the experiment, this film layer is actually three dimensional rather than the two-dimensional plot as shown. The X-axis is the streamwise location in terms of X/D while the Y-axis is the elevation (mm) from the plate surface. As shown in Fig. 3.53, a “bending back” pattern for the mist/air coolant film layer is apparently found at X/D = 13 on the upper boundary of the film and on the lower boundary also for all cases except for M=0.66. The initial momentum for the jet flow enables the coolant mixture to shoot into the main flow chamber but is bent by the main flow at a certain inclined angle. Since the injection momentum for this case M = 0.6 is not strong enough for the film to be completely lifted off from the surface, the film of air and mist mixture is “bent back” towards the surface again. The turning point for the film layer is at around X/D = 13 as shown in Fig. 3.53. **It must be noted here that this detailed description of the film layer shape is only made available with the help of the droplet measurement data.** Motivated by the goal of finding the link between the droplet behavior and the mist cooling heat transfer, the net enhancement of cooling effectiveness for this case is also plotted in the same figure of the mist/air coolant film sketch in Fig. 3.53. It is noticed that the enhancement reduces as the jet is lifted (blown off) immediately downstream of the holes. Then, it is immediately noticed that the starting point of the sudden increase in net enhancement is in the close proximity of the “bending back” location of the mist/air film. Finally, it is noticed at X/D = 80 that both the upper and bottom boundaries of the film layer tend to detach from the wall, which means that the film diffuses and loses its identity. That is why the enhancement curve flattens and tends to reduce downstream of X/D =80. This reemphasizes the fact that the enhancement of mist cooling effectiveness is better achieved when the film is attached to the wall than when it is blown off from the surface. So, “bending back” film pattern is critical in keeping the mist droplets close to the surface, thus improving the cooling effectiveness.
Figure 3.53 The relationship between the mist film cooling enhancement and the mid-plane shape of the mist/air coolant film layer for \( M = 0.6 \) Case

The effect of increasing the Blowing Ratio on the jet profile is shown in Figs. 3.54 (a) and 3.54(b). It is noticed that increasing the Blowing Ratio causes the film layer to be more lifted from the surface as it is clearly shown from the elevations of the lower and upper boundaries of the mist film. This in turn reduces the net film cooling enhancement. The “bending back” film pattern is noticed in both high blowing ratio cases with the corresponding increase in the net enhancement. Finally, the detachment of film layer from the surface occurs earlier downstream at higher blowing ratios. The corresponding reduction of the cooling enhancement can be seen after the mist film detachment in Fig. 3.54 (b).

Figure 3.54 The mid-plane shape of the mist/air coolant film layer at the hole centerline with the Net Enhancement of cooling effectiveness (a) \( M = 1.0 \) (b) \( M = 1.4 \)
Based on the description of the thermal-flow physics, the droplet size distribution curves, the droplet data rate distribution curves, and the previous discussions, an illustration of the droplet distribution patterns at different locations are drawn in Fig. 3.55 for the M=0.66 case. The drawing represents a 2D profile for the centerline distribution of the film layer. Some specific remarks may be noted as follows:

1. Bigger droplets always shoot to the upper edge of the envelope.
2. Bigger droplet outside the film layer reduces in size downstream due to contact with the hot main flow.

Droplets count inside the film layer decreases due to dry out of small droplets which is accompanied with average droplet diameter weighted toward surviving larger droplets.

Figure 3.55  A qualitative illustration of droplet size distribution in reference to the proposed mist film profile for low blowing ratio M=0.6
3.3.2 Fan-Shaped (Diffusion) holes

This section reports and analyzes the effect of the fan-shaped cooling holes geometry on droplet data and behaviors. Then, these droplet data are used to interpret the heat transfer results obtained in section (3.2). Fan-Shaped holes geometry with Blowing Ratios of 0.66, 1.0, and 1.4 are the controlling parameters studied here with corresponding Cases 20, 22, and 24. Refer to Table 3.3 for more detailed description about the studied cases.

3.3.2.1 Size distribution with droplet data rate plots

In order to give more detailed information about the droplet behavior, the average droplet size distribution is plotted together with the droplet data rates in Fig. 3.56. By putting them together in the same plot, some correlation patterns could be more conveniently discovered. The average droplet size as a function of the vertical distance from the surface is plotted in Fig. 3.56 for different streamwise locations. On the chart, the abscissa (X-axis) is the vertical distance (Y) from the plate surface. Six streamwise locations, X/D = 1, 13, 28, 48, 64, and 80 are plotted for the M=0.6 case in Fig. 3.56 (a)-(d), respectively. The data rates are plotted on the secondary ordinate (Y-axis) on the right.

Looking at the averaged droplets size (D_{10}) distribution for X/D = 1, the first measurement point is 2 mm away from the surface (compared to 3.75 mm for the cylindrical holes). It is noted that below this point, the data rate is too low to produce a meaningful measurement. This means that there is a very minimal number of droplets present under a certain elevation from the surface. From a physical point of view, this result shows that the droplets are actually lifted away from the surface at the area very close to the injection hole (X/D=1). This is understandable because mist droplets are carried into the main flow test section by the coolant. With the help of the initial coolant jet's injection momentum, droplets are shooting higher than the cooling air. The average droplet size is slightly decreasing (8-7\mu m) from 2 mm to 5 mm away from the surface and then increases to 8.84 \mu m at Y = 6mm. The average droplet size starts to decrease again to reach 7.22 \mu m at Y = 8 mm and then increases to 8.34 \mu m at Y= 9 mm. Again, it is noted that beyond Y = 9 mm, the data rate is too low for any meaningful measurements (but the burst efficiency is high).

Examining Fig. 3.56 (b) at X/D = 13, the general pattern shows that the droplet average diameter, D_{10,zig-zags}. The average droplet size first decreases to 8.32 \mu m at Y=8 mm and then
increases to 10.76 µm at Y=10 mm and then repeats the same trend. At X/D = 28, the D\textsubscript{10} distribution pattern shown in Fig. 3.56 (c) is similar to that of Fig. 3.62 (b) with the peak value of D\textsubscript{10} being about 7.4µm at Y=3 mm. The size (D\textsubscript{10}) distribution at X/D = 48 is a bit different from the previous cases. Droplets can be detected at higher elevations, Y=18 mm. Further downstream, the size (D\textsubscript{10}) distributions at X/D = 64 and 80 are very similar to that at X/D = 28 but with smaller mean diameters due to the continuous evaporation of mist droplets.

Again, similar to what has been done in the cylindrical holes cases, the droplet data rate is plotted on the second Y-axis on the right in Fig. 3.56 (a)–(f) for different X/D locations at 1, 13, 28, 48, 64, and 80, respectively. The X-axis is the vertical distance (mm) from the plate surface. Examining Fig. 3.56 for data rate plot for different X/D locations, it is found that a roughly “wedge” shape is observed in all six plots. Similar to the cylindrical holes cases, the wedge shape represents an “envelope” containing the droplets within the flow. In contrast to the cooling film layer, this enveloped region is named "droplet or mist layer" in this study. All the measurable droplets are distributed within this droplet layer, which is characterized by an apex showing the highest droplet density and bounded by upper and lower boundaries of near-zero droplet rates. Outside this envelope, the droplet density quickly drops to almost zero.

One common trend in the size plots discussed above is the existence of a small “hump” (local maximum) followed by a reduction in the size distribution (minimum value). Although not clear in all streamwise locations, this hump has some physical significance. The relatively large droplet size in the hump region is thought due to the cold air film surrounding the droplets, resulting in less evaporation. After the droplet leaves this region, diameter starts to decrease as a result of contact with the hot main air flow. So there is some reason to conjecture that the width of this hump represents the thickness of the film layer. As having been done in the cylindrical holes case, the Reynolds shear stress measurements will be used to further investigate the relationship between the droplet size distribution and coolant stream boundaries. The Turbulence Reynolds Shear Stress (\text{\overline{uv}}) is plotted on the secondary Y-axis in the same figure of the data rate curve in Fig. 3.57 for the M=0.66 case (Case 20). The turbulence Reynolds shear stress is expected to be higher in the shear layer between the coolant film layer and the main flow. Looking at these figures, the locations of the peak of the Reynolds shear stress shows some kind of agreement with the location of the peak data rate. However, unlike the cylindrical case, the agreement is weak and only few places show that. This may be attributed to the diffusive
nature of the flow produced by the fan-shaped holes, which effectively spreads the cooling film laterally in the spanwise direction. Thus, the strength of the cooling film's shear layer is reduced, resulting in a weaker turbulent mixing on the upper boundary of the cooling film than in a cylindrical hole case. This weaker turbulent mixing is favorable for protecting the surface cooler, but in the meantime also reduce its connection with Reynolds shear stress. Borrowing from the previous experience in the cylindrical case, the location of the peak of the data rate curve is again assumed to be close to the upper boundary of the cooling film in the diffusion hole cases. Based on this approach, the values of the the upper boundary of the cooling film layer, are summarized in Table. 3.5.

Table 3.5 The Y locations for coolant film upper and lower boundaries acquired from the data rate curve (Fan-Shaped holes) for different Blowing Ratios.

<table>
<thead>
<tr>
<th></th>
<th>M = 0.66</th>
<th></th>
<th>M = 1.0</th>
<th></th>
<th>M = 1.4</th>
</tr>
</thead>
<tbody>
<tr>
<td>X/D</td>
<td>Upper</td>
<td>Lower</td>
<td>X/D</td>
<td>Upper</td>
<td>Lower</td>
</tr>
<tr>
<td></td>
<td>(mm)</td>
<td>(mm)</td>
<td></td>
<td>(mm)</td>
<td>(mm)</td>
</tr>
<tr>
<td>1</td>
<td>4</td>
<td>2</td>
<td>1</td>
<td>6</td>
<td>2</td>
</tr>
<tr>
<td>13</td>
<td>9</td>
<td>4.5</td>
<td>13</td>
<td>9</td>
<td>6</td>
</tr>
<tr>
<td>28</td>
<td>9</td>
<td>3</td>
<td>28</td>
<td>8</td>
<td>3</td>
</tr>
<tr>
<td>48</td>
<td>8</td>
<td>3</td>
<td>48</td>
<td>9</td>
<td>3</td>
</tr>
<tr>
<td>64</td>
<td>8</td>
<td>2</td>
<td>64</td>
<td>9</td>
<td>2</td>
</tr>
<tr>
<td>80</td>
<td>9</td>
<td>4</td>
<td>80</td>
<td>8</td>
<td>4.5</td>
</tr>
</tbody>
</table>
Figure 3.56  Distributions of droplet size and data rate at different X/D locations for Case 20 (M=0.6) [Fan-Shaped Holes]
Figure 3.57 Distributions of data rate and Reynolds shear stress at different X/D locations for Case 20 (M=0.6) [Fan-Shaped Holes]
As in the cylindrical holes, the same procedure for analysis and discussion of detailed droplet distribution from droplet's histograms is followed in the diffusion holes. The distribution of droplet sizes for Case 20 where \( M = 0.6 \) is shown in Figs. (3.58)-(3.63) at different streamwise locations. Those figures are intended to give the information of the droplet size distribution over the flow field. Droplet size information is critical in understanding the droplets’ behavior and the evaporation process in mist cooling.

In the area very close to injection hole, \( X/D = 1 \), as shown in Fig. 3.58, the droplet distribution of different \( Y \)-elevations is plotted. At \( Y = 2 \) mm, the droplet size distribution is centered at about 8.78 \( \mu \)m and the biggest droplets are sized at 30 \( \mu \)m. Notice that the droplet distribution patterns at \( Y = 5 \) mm and 8 mm are quite similar. The difference is that the droplet data rate at \( Y = 5 \) mm is much higher than that at \( Y = 8 \) mm. Very few droplets are captured at \( Y = 9 \) mm and beyond as evidenced from the small data rate, 1hz.

In summary, in the area very close to the injection hole, \( X/D = 1 \), the droplet size distribution moves towards smaller sizes as elevation increases, i.e., away from the surface. Meanwhile the droplets smaller than 5 \( \mu \)m account for 25% of the total droplet population. These two observations are opposite to what was previously obtained for the cylindrical holes at the same \( X/D \) location. Like cylindrical holes, the droplet data rate increases first from 2 mm to 5 mm away from the surface, and then decreases to zero at higher \( Y \)-elevations. The mean diameter is around 8 \( \mu \)m at all elevations.

As \( X/D \) further increases (i.e., moving downstream), the droplet size distributions at the same elevations at \( X/D = 13 \) are plotted in Fig. 3.59. At \( Y = 5 \) mm, compared to the same plot at \( X/D = 1 \), the size distribution is very close, with the most frequently captured size being 8.76 \( \mu \)m (compared to 7.74 \( \mu \)m at \( X/D = 1 \)) and the largest size reaching 25 \( \mu \)m. At \( Y = 8 \) mm, the size distribution seems identical to those at \( X/D = 1 \) with the most frequently captured size at 8.32 \( \mu \)m (compared to 7.22 \( \mu \)m at \( X/D = 1 \)). The jet becomes wider (lifted up) and droplets could be captured at higher elevations at \( Y = 10 \) and 13 mm with D10 reaching 10.79 \( \mu \)m and 9.68 \( \mu \)m, respectively. At \( Y = 13 \) mm, the normal trend of bigger size droplets is restored again. Droplets as big as 30 \( \mu \)m are captured. The count of smaller size droplets reduces, due to evaporation, at all elevations.

125
Figure 3.58 Droplet size distribution of various Y locations at X/D=1 (Fan-Shaped Holes, M=0.66)

Figure 3.59 Droplet size distribution at various y locations at X/D=13 (Fan-Shaped Holes, M=0.66)
In the region far away from the injection hole, \( X/D = 28 \), shown in Fig. 3.60, the droplet size distributions at all elevations are almost identical. The average size is around 6.7 \( \mu m \) with a slight decrease as elevation increases. The data rate increases at \( Y=5\) mm and then decreases again. The size distributions show that the average droplet diameters are relatively uniformly distributed throughout the film at all elevations.

As \( X/D \) further increases (i.e. moving downstream), the droplet size distributions at \( X/D = 48, 64, \) and 80 are plotted in Fig. 3.61 - Fig 3.63. All the distributions are similar to that at \( X/D =28 \), with the most frequently captured size being around 7.5 \( \mu m \) and the largest size reaching 20 \( \mu m \). The only difference is that the jet becomes wider at \( X/D = 64 \) and \( X/D = 80 \) and reaches higher elevations. This relatively more uniform and similar droplet distribution can be attributed to the diffusive nature of the fan-shaped holes.

![Figure 3.60 Droplet size distribution of various Y locations at X/D=28 (Fan-Shaped Holes, M=0.66)](image)

Figure 3.60 Droplet size distribution of various Y locations at X/D=28 (Fan-Shaped Holes, M=0.66)
Figure 3.61 Droplet size distribution of various Y locations at X/D=48 (Fan-Shaped Holes, M=0.66)

Figure 3.62 Droplet size distribution of various Y locations at X/D=64 (Fan-Shaped Holes, M=0.66)
It is noted that after the peak value in the data rate curve, the data rate decreases sharply for all locations in Fig. 3.56. Based on the assumption that the data rate is proportional to the droplet density, this implies that the droplet density decreases sharply after the peak. From the previous assumption for easier analysis, the coolant film still keeps its identity and travels in a layer of spreading finite thickness. Also, the peak of the data rate curve is close to the upper boundary of the coolant film layer where the shear stress is maximum. This was captured from the Reynolds shear stress-data rate curves presented earlier (Fig. 3.57). Although the agreement between the location of the two peaks was not that strong in the fan-shaped hole case, the same conclusion may be drawn from the well established cylindrical case results. Thus, the right side of the peak of the data rate curve is actually outside the span of the coolant film. Noting that the mist is injected into the main flow channel carried by the coolant film, it is reasonable to expect that the number of droplets will decrease sharply outside the coolant film layer.

Figure 3.63 Droplet size distribution of various Y locations at X/D=80 (Fan-Shaped Holes, M=0.66)
On the other hand, if the droplets can travel outside of the coolant film layer, they must possess momentum that is high enough to break through the coolant film layer. Due to the long distance that the droplets have traveled through the mixing chamber, it is believed that the droplets have reached their terminal speed (i.e. the droplets maintain a constant slip velocity with the coolant air.) before entering the film hole. As a result, bigger droplets will have higher initial momentum shooting into the test section because the droplets are accelerated through the film hole channel from the coolant supply plenum. Bigger droplets have higher mass, and, thus, higher momentum. Therefore, the droplets that are able to acquire enough momentum to break through the coolant film must be on the “larger” side of the droplet size distribution.

As for the bottom boundary of the coolant film layer, it cannot be easily measured due to the fact that the gap between the coolant film and the test surface is in the scale of millimeters. The droplet density in this area is very low and not enough measurement points are available. Also, the refraction and reflections of laser beams due to the Acrylic surface and plate joints exacerbate the droplet measurement difficulty. As a result, there is no significant pattern from either the droplet size curve or the data rate curve that can clearly identify the jet stream lower boundary. However, rough estimates could be made by extrapolating the data rate curve to the location where low values, i.e. 2 hz would occur. The locations of the lower boundary of the cooling film stream obtained by using this approach are presented in Table 3.5.

Some noticeable features of the film/mist jet that can be withdrawn from the previous size distribution

1- Very interesting size distributions are noticed in the case of Fan-Shaped holes. Big size droplets (>20 μm) only exist at close distances from the hole, as shown at X/D =1 and X/D=13. For X/D>13, such large size droplets are absent. This may be attributed to the evaporation of such droplets assisted by the diffusive nature of the fan-shaped holes.

2- Also, for X/D>13, there are relatively similar droplet size distributions at all elevations, with average droplet diameter ranging between 7 and 8 μm.

3- The coolant film, mixed with the mist droplets, keeps its own identity traveling in the main flow channel in the area of 1 < X/D < 64 in the streamwise direction.
3.3.2.2 The Proposed coolant air/droplet spreading profiles in the cooling film

Based on the description of the thermal-flow physics, the droplet size distribution curves, the droplet data rate distribution curves, and the previous discussions, an illustration of the droplet distribution patterns at different locations are drawn in Fig. 3.64 for the M=0.66 case. The drawing represents a 2D profile for the centerline distribution of the film layer. Some specific remarks may be noted as follows:

1. Bigger droplets always shoot to the upper edge of the envelope.
2. Bigger droplet outside the film layer reduces in size downstream due to contact with the main flow. Droplets count inside the film layer decreases due to dry out of small droplets.
3. Large size droplets are completely absent for X/D>13, results in a uniform size distribution.

Figure 3.64 Illustration for droplet distribution envelope (or droplet layer) versus cooling film layer for low blowing ratio (M=0.6) mist film cooling [Fan-Shaped hole]
3.3.2.3 Linking the droplet data with the heat transfer data

Now that the droplet physics has been well examined and explained, efforts will be dedicated to using the knowledge gained for detailed droplet behavior to understand the overall mist film heat transfer process. The goal is to find the connection between the droplets behavior and the heat transfer pattern of mist film cooling.

Based on the previous analysis, the shape of the coolant film layer is outlined in Fig. 3.65 by plotting the upper and lower boundaries of the film layer. It is noted that the plot actually represents the 2-D film layer's center-plane on which the measurements were made. In the experiment, this film layer is actually three dimensional rather than the two-dimensional plot as shown. The lower boundary is plotted according to the data rate curve. The X-axis is the streamwise location in terms of X/D while the Y-axis is the elevation (mm) from the plate surface. As shown in Fig. 3.65, a “bending back” pattern for the mist/air coolant film layer is apparently found at X/D = 13 on the upper boundary of the film. The initial momentum for the jet flow enables the coolant mixture to shoot into the main flow chamber but is bent by the main flow at a certain inclined angle. Since the injection momentum for this case M = 0.6 is not strong enough for the film to be completely lifted off from the surface, the mist/air cooling film is bent back towards the surface. The turning point for the film layer is at around X/D = 13 as shown in Fig. 3.65. It must be noted here that this detailed description of the film layer shape is only made available with the help of the droplet measurement data.

![Figure 3.65 The mid-plane profile of the mist/air coolant film layer for M=0.6 Case at the hole centerline with the Net Enhancement of cooling effectiveness plotted on the right Y-axis](image-url)
Motivated by the goal of finding the link between the droplet behavior and the mist cooling heat transfer, the net enhancement of cooling effectiveness for this case is also plotted in the same figure of the mist/air coolant film sketch shown in Fig. 3.65. It is noticed that the enhancement reduces as the jet is slightly lifted (blown off) immediately downstream of the holes. Then, it is noticed that the starting point of a quick increase in the cooling enhancement is near the “bending back” location of the mist/air film. Finally, at X/D = 80 that both upper and bottom boundaries of the film layer tend to detach from the wall, indicating that the film has been diluted through a long-journey of diffusions and starts to lose its identity. So, the “bending back” film pattern is a critical mechanism in keeping the mist droplets close to the surface.

3.3.2.4 The effect of the Blowing Ratio

For other blowing ratios M=1.0 and M=1.4, the same data are plotted in Fig. 3.66 and Fig. 3.67(a)-(f). Generally, the similar D₁₀ size and data rate distribution are obtained with slight differences of the locations of maximum data rate as shown in Table 3.5. Also, increasing the blowing ratio causes the mean diameter D₁₀ to decrease at all X/D locations. This trend is clear at Fig. 3.66 and 3.67 where the D₁₀ is approximately 5-7 µm, compared to 6-8 µm at M=0.66. Unlike the cylindrical holes, increasing the blowing ratio in the fan-shaped cases reduces the data rate at the hole centerline. This is attributed to the increased lateral diffusion which spread more droplets to both sides of the hole leaving the centerline with lower data rate, see Figs. 3.66 and 3.67. The opposite trend—data rate increasing with increased blowing ratios— happens in the middle between the two adjacent holes. Based on the previous analysis, the “envelope” of the droplet layer is wider than that in the M=0.66 case. This means that the upper edge of the film is lifted due to the increase of the jet momentum as expected, compared to the M=0.66 case.

In order to further investigate the relationship between the droplet size distribution and cooling film boundaries, the Turbulence Reynolds Shear Stress (\(\overline{uv}\)) is plotted on the secondary Y-axis in the same figure of the size distribution curve shown in Figs. (3.68)-(3.69) for blowing ratios of 1.0 and 1.4, respectively. Results similar to the M=0.66 case is obtained suggesting a reasonable agreement between the locations of peak data rate and maximum shear stress, where the cooling film's upper boundary is proposed. As can be seen in Figs. 3.68 and 69, more definite local high values of Reynolds shear stress occur than in the low blowing case.
Nonetheless, it is still more certain to use the location of 2 Hz data rate as the reference to identify the lower boundary of the cooling film than using the Reynolds shear stress value. Both estimated upper and lower cooling film boundaries are listed in Table 3.5 above.

Figure 3.66  Distributions of droplet size and data rate at different X/D locations for Case 22 (M=1.0) [Fan-Shaped Holes]
Figure 3.67 Distributions of droplet size and data rate at different X/D locations for Case 24 (M=1.4) [Fan-Shaped Holes]
Figure 3.68  Distributions of data rate and Reynolds shear stress at different X/D locations for the Fan-shaped holes case with M=1.0
Figure 3.69 Distributions of data rate and Reynolds shear stress at different X/D locations for the Fan-shaped holes case with M=1.4.
The effect of increasing the blowing ratio on the jet profile is shown in Figs. 3.70 (a) and 3.70(b). It is noticed that increasing the blowing ratio causes the film layer to move (higher in the Y-direction, which tends to lift the film further from the surface. This stronger lift-off in turn reduces the cooling enhancement. The “bending back” film pattern is noticed in both higher blowing ratio cases. The corresponding increase in the net enhancement near the bending-back location is not as clear as in the low blowing ratio (M=0.6) case, but the increasing trend is still visible. Finally, the film layer starts to detach from the surface at a location at about same location as in M=0.6 case, accompanied with a reduction in cooling enhancement.

![Figure 3.70](image)

**Figure 3.70** The mid-plane profiles of the mist/air coolant film layer at the hole centerline with the cooling enhancement plotted on the right Y-axis (a) M=1.0 (b) M= 1.4

### 3.4 Some 3D Aspects of the Film Layer [Cylindrical Vs. Fan-Shaped Holes]

As mentioned earlier, all the measurements are made in the centerline plane, which gives a 2D picture of the film layer profile as been depicted above. In order to fully understand the 3D nature of the film flow, some measurements have been taken in the lateral direction at selected streamwise and elevation locations.

For cylindrical holes, Fig. 3.71 (a)-(d) shows the droplet diameter and the data rate in the lateral (spanwise) direction (Z) from the middle hole centerline (Z=0) to the next hole centerline (Z=20) at several selected elevations (Y) mm from the plate surface. Figure 3.71 (a) shows the results at X/D = 1 and elevation Y = 6 mm. The result shows that the data rate is maximum at the middle hole centerline (4655 HZ) and decreases to zero half the distance between the holes (at Z=10 mm), indicating low cooling enhancement in this region. It is noticed that the data rate in
the middle hole center (Z=0) is almost three times as high as the data rate of the adjacent hole (Z=20). This droplet rate difference indicates a non-uniform distribution of the droplets between holes at this elevation (Y=6mm). Moving downstream of the injection hole to X/D=13 at Y=10 mm, Fig 3.71 (b) shows that the diameter and the data rate distributions become more uniform with an average value of $D_{10} = 11 \mu m$. The data rate is small at Z=10, yet it is not Zero, and the $D_{10}$ overshoots above 10µm due to the entrainment of bigger size droplets near the cooling film’s upper boundary. Moving further downstream, at X/D=28, Fig. 3.71 (c) shows more uniform distributions of diameter and data rates. Finally, at X/D = 64, Fig. 3.71 (d) shows that the data rate and the diameter distributions are very uniform. This is a clear indication of the complete merge of the two adjacent cooling film layers, signified by a better and more uniform lateral coverage.

Figure 3.71 Lateral distributions of diameter and data rate at selected X/D and Y locations downstream of the cylindrical holes for Case 14 (M=0.66)
For fan-shaped holes, Fig. 3.72 (a)-(j) shows the droplet diameter and the data rate in the lateral (spanwise) direction (Z) from the middle hole centerline (Z=0) to the next hole centerline (Z=20) at selected elevations (Y) mm from the plate surface. Figure 3.72 (a) shows the results at X/D = 1 and elevation Y = 5 mm. In contrast to the results of the cylindrical holes, the data rate unexpectedly reaches maximum between holes (2571 Hz at Z=9) and decreases to much lower values at holes' centers. Apparently, this phenomenon of more droplets presenting between the holes is caused by the diffusion hole’s specific geometry. It implies that the droplets have somehow gained lateral momentum when they are transported thorough the diffusion hole passage. Again, if this lateral component of momentum is gained in the diffusion hole, larger droplets are supposed to gain a higher inertia and be thrown off farther from the hole’s center plane. Surely, the droplet size data in Fig. 3.72 verifies that larger droplets are indeed accumulated near the region half-way between the holes. This clearly explains the reason for excellent lateral coverage and the very high cooling effectiveness between jet injections, especially at X/D locations close to the holes.

It is again noticed that the data rate in the middle hole center (Z=0) is more than the data rate of the adjacent hole (Z=20). Like the cylindrical hole cases, this behavior indicates a non-uniform distribution of the droplets between holes from the mist supply chamber. Inside the mixing chamber where the droplets hitting the chamber walls tend to stick to the wall and leave more droplets enter the middle hole than the side holes. Moving downstream of the injection hole, Figure 3.72 (c) - (d) at X/D = 13 shows that the diameter and the data rate distributions become more uniform with an average value of $D_{10} = 7.5 \, \mu m$. Moving further downstream, at X/D = 48 and 80, Fig. 3.72 (g)-(j) show very uniform distributions of diameter and data rates. This is a clear indication of the complete merge and mixing of the two injections. It can be concluded that for X/D>48, the fan-shaped holes provide a nearly 2D uniform cooling film coverage of the entire surface.
Figure 3.72 Lateral distributions of diameter and data rate at selected X/D and Y locations downstream of the fan-shaped holes for Case 20 (M=0.66)
CHAPTER 4
MIST TRANSPORT TO HIGH PRESSURE TURBINE COMPONENTS

4.1 INTRODUCTION

Gas turbines play a vital role in today’s industrialized society. As the demands for power increase, the power output and thermal efficiency of gas turbines must also increase. One method of increasing both the power output and thermal efficiency of the engine is to increase the turbine inlet temperature. In the modern advanced gas turbines, the turbine inlet temperature can be as high as 1500°C; however, this temperature exceeds the melting temperature of the metal airfoils. Therefore, it is imperative that the blades and vanes are cooled so they can withstand these extreme temperatures. Cooling air around 600K is extracted from the compressor and passes through the airfoils. With appropriate implementation of cooling schemes, the temperature of the blades can be lowered to approximately 1000K, which is permissible for reliable operation of the engine.

The film cooling technique has been applied in modern gas turbines since the 1980s to protect the hot turbine components, such as turbine blades and vanes, from hot flue gases. As there is a need to continuously increase the turbine inlet temperature to improve gas turbine performance, continuous improvement of film cooling effectiveness is essential. There have been numerous studies that have focused on air film cooling over flat surfaces or turbine airfoil surfaces with streamwise coolant injection in the past decades (Goldstein, 1971; Mayhew et al., 2003; Walters and Leylek, 1997); others have studied film cooling in airfoil cascade environments to better simulate the flow and heat transfer mechanisms at engine conditions (Zhang and Pudupaty, 2000; Drost and Bolcs, 1998; Medic and Durbin, 2002). While most of the above studies were conducted at the stationary cascade blades, studies on rotating turbine are also abundant. Dunn et al. (Dunn et al., 1986 I, II) studied the heat transfer on the vane, end walls, and rotors in a full-stage rotating turbine using a shock-tunnel facility and thin-film heat flux gages.

As the working gas temperature continuously increases to augment thermal efficiency, new cooling techniques are needed to surpass incremental improvements of convectional gas turbine cooling technologies. A promising technology to enhance film cooling is to inject water
mist into the coolant flow. Each droplet acts as a cooling sink and flies over a distance before it completely vaporizes. This “distributed cooling” characteristic allows controlled cooling by manipulating different sizes of injected water droplets. The enhanced cooling is attributed to many factors: (a) The flow temperature is reduced mainly due to droplet evaporation and partially due to larger specific heats of water and water vapor; (b) the droplets’ interactions with the flow augments turbulent mixing; (c) the sudden expansion of water vapor volume (about 900~3600 %) from fast liquid evaporation when liquid droplets touch a hot wall introduces a expulsive momentum thrust that also enhances mixing and convective heat transfer; and (d) the brief period that the liquid droplet is in contact with the hot surface provides an enhanced wall heat transfer through direct heat conduction. Another important merit of employing mist film cooling is that some larger droplets can fly longer and evaporate farther into the downstream region where the single phase air film cooling becomes less effective.

Numerous experimental and numerical investigations have been conducted on the field of air/mist cooling. One of the early experimental studies was done by Takagi and Ogasawara (Takagi and Ogasawara, 1974) to investigate mist/air flow and heat transfer inside a vertical rectangular tube. In this study, compressed air was mixed with water droplets generated by an atomizing nozzle before it went through the test section. Their results showed that the heat transfer coefficient decreased as the wall temperature increased. Moreover, the heat transfer coefficient increased as the droplet concentration or the air flow-rate increased or as droplet size decreased.

Mori, et al. (1982) performed an experimental study of the mist/air cooling of a highly heated vertical tube of 1.8 mm ID. It was found that the heat transfer along the tube axis could be divided into three typical regions namely, liquid film region, dry-out region, and gas-phase forced convection region. In the liquid film region, the heat transfer coefficient is almost ten times higher than that without mist. In the gas-phase forced convection region, the heat transfer coefficient follows the single-phase convective heat transfer correlations. Janssen, et al. (Jansen et al., 1986) performed another study on mist/air cooling of a very hot tube. The experimental setup was similar to that of Mori, et al. (1982). Experimental results demonstrated that a mixture of hot air and cold water droplets would cool a hot surface more effectively than the same mixture introduced to the same surface after the droplets have totally evaporated. These experimental results actually suggested that the droplet dynamics play an important role in the
heat transfer enhancement. A heat transfer coefficient 10 times higher than that in the gas only region was also observed in the liquid film region.

Guo et al. (2000 I, II) studied the mist/steam flow and heat transfer in a straight tube under highly superheated wall temperatures. It was found that the heat transfer of steam could be significantly improved by adding mist into the main flow. An average enhancement of 100% was achieved with less than 5% mist. In another experimental study with a horizontal 180-degree tube bend, Guo et al. (2000) found that the outer wall has better cooling than the inner wall. However, the inner wall can achieve better cooling enhancement. The overall cooling enhancement ranged from 40 to 300 percent with some local maximum enhancement being over 500 percent. Nazarov et al. (2009) conducted experimental work investigating the heat transfer processes in cooling surfaces by a pulse gas-droplet stream. It has been shown that depending on the time parameters of the pulse spray, the integral heat transfer can effectively be controlled. They also showed that a concurrent air supply with jets, leads to an efficacious intensification of the heat transfer between the spray and a vertical obstacle. It needs to be noted that the physics of sprayed liquid jets is different from mist flow as been discussed in Chapter1. Recently, Pakhomov et al. (2010) performed a numerical simulation using the Eulerian/Eulerian method to study the flow structure and heat transfer of impact mist jet with low concentration of droplets (liquid mass flow ≤1%). Their results showed that the introduction of droplets causes a substantial increase of heat transfer rate when compared with a one-phase air impact jet.

Wang, et al. (2005) conducted an experimental study of a mist/steam cooling system consisting of three rows of circular jet impingement in a confined channel. The experiment results indicated that an average cooling enhancement of 200-300% was achieved with a local maximum enhancement over 800%. Nirmalan, et al. (1996) applied mist/air cooling in a gas turbine vane. They found that by using mist/air cooling, the cooling airflow can be reduced more than 50% to reach the same overall cooling levels of air-only cooled vanes, but they also found that the leading-edge area is overcooled.

Because of the difficulty and cost of conducting an experiment at high Reynolds number and under elevated pressure and temperature conditions, CFD simulation has been implemented to provide preliminary flow and heat transfer physics. Many numerical studies have been performed to show the effectiveness of air/mist cooling technique. Li and Wang (2006, 2007) simulated mist/air film cooling and showed that a small amount of mist injection (2% of the
coolant mass flow rate) could increase the adiabatic film cooling effectiveness by about 30% - 50% under low temperature, velocity, and pressure conditions similar to those in the laboratory. They also investigated the effects of different flow parameters, injection hole configurations, and coolant supply plenum on the cooling effectiveness. Both 2D and 3D film cooling geometries were simulated. Under the GT operating conditions with high temperature and high pressures, Wang and Li (2008) found the mist cooling enhancement was less attractive in terms of “enhancement percentage” (10-20%) than the cases with low pressure, velocity, and temperature conditions. However, due to high surface temperature in the real gas turbine condition, a relatively smaller percentage of cooling enhancements can result in a larger wall temperature reduction, which is critical to significantly extend the life expectancy of gas turbine airfoils. To further simulate more closely to the actual GT operating conditions, Li and Wang (2008) presented the mist/air film cooling heat transfer coefficient under conjugate condition by employing internal channel cooling beneath the blade surface. The results of conjugated 2-D cases indicated that reverse heat conduction from downstream to upstream along the solid wall was strong within a distance of 5 slot widths. Recently, Dhanasekaran and Wang (2009) studied the effect of using mist film cooling for rotating gas turbine blades under lab and elevated (real) operating conditions. Their results showed that the average mist cooling enhancement of about 15% and 35% are achieved on the laboratory and elevated conditions, respectively. This can translate into a significant blade surface temperature reduction of 100-125 K with 10% mist injection at elevated condition.

The previous reviewed work shows that mist cooling is a promising and efficient technique based on the assumption that the mist can be transported and delivered to the inlet of the cooling sites. However, the following questions and challenges have yet to be answered. Where is the mist generated? Can the mist survive the extremely hot conditions inside the flow passages in the gas turbine and be successfully delivered to the needed sites? The objective of the present part of the study (Chapter 4) is to answer these questions. Section 4.1 will be dedicated to mist transport in the internal cavities of the vane. Section 4.2 will be dedicated to the compressed water transport to the injector inside the pre-swirl chamber. Finally, mist transport to the rotating components (blades and rotating disk cavities) will be covered in Section 4.3.
4.2 AN INVESTIGATION OF APPLICABILITY OF TRANSPORTING WATER MIST FOR COOLING TURBINE VANES

The **objective** of the present part of the study (Sec 4.2) is to use a CFD scheme to investigate the feasibility of transporting the mist through the internal cooling air passage to the film cooling hole sites on the **vanes**. This is the first part of a series of studies to investigate how to transport mist to the needed sites, which include vanes, blades, pre-swirlers, and rotating disk cavities.

**4.2.1 Studied Configuration**

A schematic of a typical film cooled high-pressure turbine is shown in Fig. 4.1. The relatively cold compressor air (600K) is bled from the compressor discharge through the cooling air passage where it enters the cooling channels inside the turbine vanes to cool the vanes' walls, which are toasted by extremely hot flu gas at about 1300K (2350°F). Eventually, the air exits the film cooling holes to form a protective layer around the external vane surface to minimize direct contact with the hot gases. Figures 4.2a and 4.2b provide a detailed view of the air flow path starting at the cooling air duct near the casing and ending at the film cooling holes at the vane surface. This path represents the computational domain of interest.

![Figure 4.1 Schematic of mist cooling of high pressure turbine using water mist injected through the Cooling air passage. (Modified from (Kurzke, 2007))](image-url)
The atomizer is proposed to be located at the compressor bleed location, which is a short distance just upstream of the vane's leading edge. The water mist is to be injected at the inlet of the cooling air passage along the casing. The tubing of the high-pressure water is directly inserted through the turbine casing. For simplicity, only one vane sector is selected for study. A more complicated arrangement or optimization strategy could be arranged in the future to reduce engineering cost and improve mist-distribution effectiveness.
4.2.2 Zero-Dimensional Model

A quick and reasonably accurate estimate of the droplet's *residence time* can be obtained with a simple zero-dimensional model. The residence time is the time during which the droplet starts to evaporate from the droplet surface, then boil from within the droplet, and finally become all vapor. The following correlation (Kuo, 1986) is used to calculate the residence time ($\tau$) of the droplet:

$$
\tau = \frac{\rho_f h_{fg} d_r^2}{\lambda_1 (1+0.23\sqrt{\text{Re}})(T_1-T_f)}
$$

Equation 4.1 was derived assuming steady flow at constant pressure with the free stream temperature greater than the boiling temperature of the droplet in order for boiling to occur. It also assumes that radiation is inactive and that the droplet remains at fixed temperature (boiling temperature) throughout the boiling process. In this study, the water temperature $T_f$ =300K, air temperature $T_1$ = 600K, latent heat $h_{fg}$=2256 kJ/Kg, thermal conductivity of air $\lambda_1$=0.048 W/m·K, and droplet Reynolds number, based on droplet diameter and assumed slip velocity of 10 m/s, $\text{Re}_d = 55$.

Based on this data, the droplet's residence time for diameters of 10 and 20 µm is calculated and shown in Table 4.1. The distance travelled during this period of residence time is estimated based on the cooling air velocity at 50 m/s. The results tabulated in Table 4.1 are very informative.

**Table 4.1 Droplet residence time and potential traveling distance for two different sizes of water droplets**

<table>
<thead>
<tr>
<th></th>
<th>D =10 µm</th>
<th>D = 20 µm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time (s)</td>
<td>0.0072</td>
<td>0.0233</td>
</tr>
<tr>
<td>Distance Travelled (m)</td>
<td>0.358</td>
<td>1.169</td>
</tr>
</tbody>
</table>

They state that if the initial water droplets are 20 µm in diameter, their residence time is about 23 ms; and during this flashing instant, they can fly a distance greater than 1 m before they completely disappear. In a typical 7-frame GT, the distance over 1 m can cover the distance from the mist injection point to the film cooling hole base and most of the vane's surface. It is
understood that, once the mist exits the film cooling holes, the droplets will face hot gas temperatures as high as 1300K. Therefore, the portion of residence time outside the vane will be shortened by about 70%. However, this quick estimate is meant to determine if the droplets can survive and reach the film cooling holes. Once they reach the cooling holes, the external film cooling behavior has been intensively studied by (Guo et al., 2000 I, II; Wang et al., 2005; Li and Wang, 2006 and 2007; Wang and Li, 2008; Li and Wang, 2008; Danasekaran and Wang, 2009; Wang and Danasekaran, 2010; Danasekaran and Wang, 2008, 2010, and 2011). It is important to emphasize that many think that the mist would disappear "instantaneously" when it is injected into the hot gas environment. Here, the above calculation informs us that this "instantaneous time" is not zero seconds, but is about 23 ms for a droplet initially in 20 µm and is exposed to 600 K gas. Even if the air temperature is raised to 1300 K, the calculation shows that a 20 µm droplet will take 4 ms to completely evaporate, which is still not zero seconds.

Of course, this zero-dimensional analysis and associated correlation have some uncertainties, but they provide quick and positive feedback to our questions. This positive result motivates a continuous study to employ a more sophisticated multi-phase CFD scheme.

4.2.3 CFD Calculations

The zero-dimensional model results showed that the application of water mist/air cooling technique is feasible. To obtain more reliable results, comprehensive CFD calculations are performed for flow under real gas turbine operating conditions.

4.2.3.1 Numerical method

A feasible method to simulate cooling with air/mist injection is to consider the droplets as a discrete phase since the volume fraction of the liquid is small (less than 0.1%). The trajectories of the dispersed phase (droplets) are calculated by the Lagrangian method (Discrete Phase Model, DPM). The impacts of the droplets on the continuous phase are considered as source terms to the governing equations of mass, momentum, energy, and species. The following are the governing equations of mass, momentum, energy and species, which are based on time averaged steady state conditions:

\[
\frac{\partial}{\partial x_i} (\rho u_i) = S_m
\]  

(4.2)
\[ \frac{\partial}{\partial x_i} (\rho u_i u_j) = \rho g_j - \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_i} (\tau_{ij} - \rho \bar{u}_i \bar{u}_j) + F_j \] (4.3)

\[ \frac{\partial}{\partial x_i} (\rho c_p u_i T) = \frac{\partial}{\partial x_i} \left( \lambda \frac{\partial T}{\partial x_i} - \rho c_p \bar{u}_i \bar{T} \right) + \mu \Phi + S_h \] (4.4)

\[ \frac{\partial}{\partial x_i} (\rho u_i C_j) = \frac{\partial}{\partial x_i} \left( \rho D_j \frac{\partial C_j}{\partial x_i} - \rho \bar{u}_i \bar{C}_j \right) + S_j \] (4.5)

where \( \tau_{ij} \) is the symmetric stress tensor. The source terms \( (S_m, F_j \text{ and } S_h) \) are used to include the contributions from the dispersed phase. \( \mu \Phi \) is the viscous dissipation, and \( \lambda \) is the heat conductivity. \( C_j \) is the mass fraction of the species \( (j) \) in the mixture, and \( S_j \) is the source term for this species. \( D_j \) is the diffusion coefficient. The diffusion term is used for bi-diffusion between water vapor and air mass. When the liquid evaporates into water vapor, it surrounds the liquid droplet. Then the water vapor will be transported away through convection and mass diffusion. Two species (air and water vapor) are simulated in this work. The terms of \( \rho \bar{u}_i \bar{u}_j \), \( \rho c_p \bar{u}_i \bar{T} \), and \( \rho \bar{u}_i \bar{C}_j \) in the equations above represent the Reynolds stresses, turbulent heat fluxes, and turbulent concentration (or mass) fluxes, which should be modeled properly for a turbulent flow as seen in the cooling passages in gas turbines.

Generally, the Reynolds Stress Model (RSM) turbulence model performs better and is more consistent with experimental data in mist/air or mist/steam cooling applications as reported in (Wang and Danasekaran, 2010; Danasekaran and Wang, 2008). Also, the standard k-\( \varepsilon \) turbulence model has been proven to be robust with good results only next to the RSM model (Wang and Danasekaran, 2010; Danasekaran and Wang, 2008), but k-\( \varepsilon \) model required almost an order of magnitude less computational time than the RSM model does. Since the primary goal of this study is to give an estimate of the droplet residence time, the standard k-\( \varepsilon \) model is used with standard wall functions to model the near-wall turbulence structure. The mesh is generated with the first near-wall cells located \( y^+ \approx 30 \).

The equations for the turbulent kinetic energy \( (k) \) and its dissipation rate \( (\varepsilon) \) are as follows.

\[ \frac{\partial}{\partial x_i} (\rho u_i k) = \frac{\partial}{\partial x_i} \left( \mu + \frac{\mu_k}{\sigma_k} \right) \frac{\partial k}{\partial x_i} + G_k - \rho \varepsilon \] (4.6)

\[ \frac{\partial}{\partial x_i} (\rho u_i \varepsilon) = \frac{\partial}{\partial x_i} \left( \mu + \frac{\mu_\varepsilon}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} + C_{1\varepsilon} G_k \frac{\varepsilon}{k} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \] (4.7)
The term $G_k$ is the generation of turbulence kinetic energy due to the mean velocity gradients. The turbulent viscosity, $\mu_t$, is calculated from the equation

$$\mu_t = \rho C_\mu \frac{\varepsilon^2}{k} \quad (4.8)$$

and the effective heat conductivity ($\lambda_{\text{eff}}$) and the effective diffusion coefficient are calculated by the following two equations, respectively.

$$\lambda_{\text{eff}} = \lambda + C_p \mu_t / P_{\text{rt}} \quad (4.9)$$
$$D_{\text{eff}} = D + \mu_t / S_{\text{ct}} \quad (4.10)$$

The constants $C_{1\varepsilon}, C_{2\varepsilon}, C_\mu, \sigma_k$, and $\sigma_\varepsilon$ used are: $C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3$ (Launder and Spalding, 1972). The turbulence Prandtl number, $P_{\text{rt}}$, is set to 0.85, and the turbulence Schmidt number, $S_{\text{ct}}$, is set to 0.7.

**Discrete Phase Model (Water Droplets).** As the injected water mass flow is small (about 10% of the cooling air) and the droplet diameter is also small, the volume fraction of water is small (<1%): much less than the upper limit condition (10%) to use the DPM. Following the governing equations for the droplet flow:

Droplet Flow and Heat Transfer- The droplets are tracked by the Lagrangian method by applying Newton’s 2nd Law with the following equation of motion:

$$m_p \frac{dV_p}{dt} = \sum F = F_D + F_g + F_{th} + F_S + F_o \quad (4.11)$$

where $m_p$ is the droplet mass and $V_p$ is the droplet velocity (vector). The right-hand side is the combined force acting on the droplets, including $F_D$ (drag force), $F_g$ (gravity and buoyancy force), $F_S$ (Saffman lift force), and $F_{th}$ (thermophoretic force). $F_o$ includes all other forces like rotational forces. For more details about these forces see the manual for ANSYS V12.0 (2009).

The energy equation for any individual droplet can be given as the following equation:

$$m_p c_p \frac{dT_p}{dt} = \pi d^2 h(T_\infty - T) + \frac{dm_p}{dt} h_g + \pi d^2 c_p \sigma \left( \frac{\theta_p^4}{\theta_R^4} - T_p^4 \right) \quad (4.12)$$
where \( h_{fg} \) is the latent heat, \( \theta_R = (I/(4\sigma))^{1/4} \) is the radiation temperature, and \( I \) is the radiation intensity. The convective heat transfer coefficient (\( h \)) can be obtained with an empirical correlation (Ranz and Marshal, 1952), and the radiation heat transfer term can be reasonably neglected because the extracted compressed air doesn't include hot, combusted gases.

The evaporated mass is calculated by two modes: evaporation and boiling. During the evaporation mode, the evaporated mass change rate or vaporization rate is affected by the relative humidity in the air and is shown in Eq. (4.13) as being governed by the concentration difference between droplet surface and the air stream,

\[
- \frac{dm_p}{dt} = \pi d c_k (C_s - C_\infty) + \frac{dm_p}{dt} h_{fg} \tag{4.13}
\]

where \( c_k \) is the mass transfer coefficient and \( C_s \) is the vapor concentration at the droplet surface, which is evaluated by assuming that the flow over the surface is saturated. \( C_\infty \) is the vapor concentration of the bulk flow, which is obtained by solving the transport equations. When the droplet temperature reaches the boiling point, the following equation can be used to evaluate its evaporation rate (Kuo, 1986):

\[
- \frac{dm_p}{dt} = \pi d^2 \left( \frac{\lambda}{d} \right) \left( 2.0 + 0.46 \text{Re}_d^{0.5} \right) \ln \left( 1 + c_p(T_\infty - T)/h_{fg} \right)/c_p \tag{4.14}
\]

where \( \lambda \) is the gas/air heat conductivity and \( c_p \) is the specific heat of the bulk flow.

Stochastic method (ANSYS Version 12.0, 2009) is used to consider turbulence dispersion effect on droplets tracking. The droplet trajectories are calculated with the instantaneous flow velocity \( (u + u') \), and the velocity fluctuations are then given as:

\[
u' = \zeta \left( \frac{\overline{u^2}}{2} \right)^{0.5} = \zeta \left( \frac{2k}{3} \right)^{0.2} \tag{4.15}\]

where \( \zeta \) is a normally distributed random number. This velocity will apply during the characteristic lifetime of the eddy \( (t_e) \), a time scale calculated from the turbulence kinetic energy and dissipation rate. After this time period, the instantaneous velocity will be updated with a new \( \zeta \) value until a full trajectory is obtained. Since the results are sensitive to the time scale, an appropriate selection of the time scale is critical. In this study, the time scale is selected as 0.009 \( (k/\varepsilon) \). More detailed study about the effect of time scale on computational results and an appropriate selection of time scale is referred to (Dhanasekaran and Wang, 2008).
4.2.3.2 Computational domain

The computational domain is a part of the cooling passage that feeds the vane with the cooling air besides the cooling cavity inside the vane. As the Frame 7FA engine contains 48 vanes in the first turbine stage, the computational domain is a sector of \(360/48 = 7.5^\circ\) in the circumferential direction. The part simulated from the air cooling passage is 25 cm in the axial direction, which represents 1.25 the vane axial chord length with a height of 7 cm. The vane cooling cavity (channel) is approximated to be a cylinder with a 5 cm diameter and a height of 18 cm equal to that of the vane. The exit of the domain is a row of film cooling holes that contains 28 holes in the radial direction (z-direction) with 0.15 cm in diameter for each. The computational domain is shown in Fig. 4.3 with red arrows representing the cooling air flow direction. The water mist is injected at the inlet of the cooling passage and moves with the air until the droplets reach the exit holes.

![Computational Domain Diagram](image)

**Figure 4.3 The computational domain**

4.2.3.3 Boundary conditions

4.2.3.3.1 Airflow

The main flow is assumed to be dry air (zero humidity). The mist cooling is investigated at real working conditions of a Frame 7FA gas turbine engine. As a single row of film cooling
holes is used, the mass flow rate is calculated as a 1.6 % of the engine total mass flow rate. This mass flow is divided among the 48 nozzles to give an inlet mass flow rate of 0.1483 kg/s, which is imposed on the inlet boundary. The inlet static temperature is 600 K, reasonably assumed equal to the compressor discharge temperature. Inlet turbulence conditions for $k$ and $\varepsilon$ are defined according to the following correlations:

$$k = \frac{3}{2} (u_{avg} I)^2 \quad (4.16)$$
$$\varepsilon = C_\mu \frac{k^{3/2}}{\ell} \quad (4.17)$$
$$\ell = 0.07 D_h \quad (4.18)$$

where the turbulence intensity $I = 5\%$ is assigned to simulate a fully-developed duct flow; the turbulence length scale, $\ell$, is based on the inlet hydraulic diameter of the air cooling duct $D_h = 9.12 \, \text{cm}$; $C_\mu = 0.09$; $u_{avg}$ is the average inlet air velocity.

For the cooling passage along the outer casing and feeding the vane cavity, the side walls in the circumferential direction are assumed symmetric with zero normal gradients; the bottom wall has a constant temperature of 1100 K as it is adjacent to the hot flue gases about 1300 K; and the upper and end walls are assumed adiabatic. For the cooling cavity walls, the temperature is taken as 900 K in the baseline case. All walls are assumed to be no-slip walls. At exit of the domain, which is a vertical row containing 28 holes, a constant static pressure of 16 atm is imposed along the boundary. This pressure represents the actual pressure of the flue gases in the main stream felt by the cooling holes. These boundary conditions are summarized in Table 4.2.

**Table 4.2 Boundary conditions**

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Values Assigned</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Mass flow rate = 0.01483 kg/s, $T = 600 , \text{K}$, $I = 5 %$, $D_h = 9.12 , \text{cm}$</td>
</tr>
<tr>
<td>Exit</td>
<td>Static Pressure = 16 atm</td>
</tr>
<tr>
<td>Sides</td>
<td>Symmetric with zero normal gradients</td>
</tr>
<tr>
<td>Walls</td>
<td>No-Slip, Lower passage wall at 1100 K, Cavity wall temperature = 900 K</td>
</tr>
</tbody>
</table>

155
4.2.3.3.2 Droplet injection

The uniform droplet size of 30μm is considered in the base case (Table 4.3) with injection velocity of 1.05 m/s (equal to the mass weighted average velocity of the air flow at inlet); and the mist ratio, the mass ratio of mist over cooling airflow, is 10% (about 0.01483 kg/s) in the base case. The number of mist injection points at the coolant inlet depends on the number of computational elements (cells) at the inlet surface. In the present case, about 8160 injection points are placed. The trajectory number for stochastic tracking is chosen to be 10.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Air (Dry base Case)</strong></td>
<td></td>
</tr>
<tr>
<td>Air inlet mass Flow</td>
<td>0.1483 kg/s</td>
</tr>
<tr>
<td>Air inlet temperature</td>
<td>600 K</td>
</tr>
<tr>
<td>Exit static pressure</td>
<td>16 atm</td>
</tr>
<tr>
<td>Cavity wall temperature</td>
<td>900 K</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>Standard k-ε</td>
</tr>
<tr>
<td><strong>Droplets (Wet base case)</strong></td>
<td></td>
</tr>
<tr>
<td>Droplet initial diameter</td>
<td>30 μm</td>
</tr>
<tr>
<td>Droplet initial velocity</td>
<td>1.05 m/s</td>
</tr>
<tr>
<td>Droplet initial temperature</td>
<td>300 K</td>
</tr>
<tr>
<td>Mist Ratio</td>
<td>10 %</td>
</tr>
<tr>
<td>Droplet wall boundary cond.</td>
<td>Reflect</td>
</tr>
</tbody>
</table>

4.2.3.3.3 Discrete-phase wall boundary condition

When the droplet reaches the wall, its trajectory is determined from the discrete-phase wall boundary condition. There are many possible trajectories that a droplet can take; and each droplet, as it approaches the wall, has a particular trajectory (ANSYS Version 12.0, 2009) based on whether the wall is dry or flooded. In the case of a dry wall, the droplets have three major regimes, including reflect, break-up, and trap. According to Watchers et al. (Watchers and Westerling, 1966), the regimes depend on the incoming Weber number of the droplet. Here, the Weber number is the ratio of kinetic energy of the droplet to its surface tension energy (\(We = \frac{\rho DV^2_d}{\sigma}\)). It was shown from their experimental results that the droplet with an incoming Weber
number \( \text{We}_{\text{in}} \) less than 10 reflects elastically with a nearly equal outgoing Weber number \( \text{We}_{\text{out}} \). As the incoming We increased further to \( \text{We}_{\text{in}} > 80 \), the droplet falls into disintegration region which leads to breakup of the droplet to several small droplets. In the transition region of \( 30 < \text{We} < 80 \), the droplet has the possibility of either reflecting or breaking-up. Apart from the above two facts, the droplets can be instantaneously vaporized by the superheated wall also. In this case, the trajectory calculations are terminated and particles' entire mass passes into the vapor phase and enters the cell adjacent to the boundary. On the other hand, in the presence of a flooded wall condition, a droplet has the chance for four different regimes, including splashing, spreading, rebounding, or sticking. For the spread regime, the arriving drops are assumed to coalesce to form a local film. The wall-film boundary condition used in this study is based on the work of Stanton et al. (Stanton and Rutland, 1996) and O’Rourke et al. (O’ Rourke and Amsden, 2000). The four regimes — stick, rebound, spread, and splash — are based on the impact energy and wall temperature. Below the boiling temperature of the liquid, the impinging droplet can stick, spread, or splash; however, while above the boiling temperature, the particle can only either rebound or splash. The impact energy is defined by

\[
E^2 = \left( \rho \frac{V_r^2 D}{\sigma} \right) \left[ \frac{1}{\min\left(h_0/D,1\right)} + \frac{\delta_{bl}}{D} \right] \tag{4.19}
\]

where \( V_r \) is the relative velocity of particle in the frame of the wall, \( \delta_{bl} \) is the boundary layer thickness, \( D \) is the droplet diameter, \( \sigma \) is the droplet surface tension, and \( h_0 \) is the film thickness.

The sticking regime is applied when the value of \( E \) becomes less than 16. Splashing occurs when the impingement energy is above a critical \( E \) value of \( E_{cr} = 57.7 \). More details about the wall-film model can be found in (Stanton and Rutland, 1996; O’ Rourke and Amsden, 2000). In addition to the reflect and the wall-film boundary conditions, there is a commonly used boundary condition known as the wall-jet model and it is based on the work of Naber and Reitz (Naber and Reitz, 1988). The wall-jet boundary condition is appropriate for high-temperature walls where no significant liquid film is formed, and in high-Weber number impacts where the spray acts as a jet. The wall-jet boundary condition assumes an analogy with an inviscid jet impacting a solid wall where the outcome of impingement (the direction and velocity of the droplet) depends mainly on the droplet Weber number and on its impingement angle. According to the value of Weber number, the droplet may reflect \( \text{We} < 80 \) or slide along the wall.
A more detailed description of the underlying theory of that model is available in (Naber and Reitz, 1988). The boundary condition of droplets at walls in the base case is assigned as “reflect,” which means the droplets elastically rebound once reaching the wall.

The droplets are injected with a velocity equal to that of the inlet air and a temperature of 300 K. The details of base case conditions are given in Table 4.3. To further understand the effect of some important operating parameters, a parametric study is performed as shown in Table 4.4. The parameters chosen for study are droplet initial diameter ($D_i$), the mist ratio, the cavity wall temperature ($T_w$), and the droplet boundary condition at the wall. The results of the parametric study are shown in the results section of this work. At the outlet, the droplets just simply escape from the computational domain (exit holes).

<table>
<thead>
<tr>
<th>Case</th>
<th>$D_i$ ($\mu$m)</th>
<th>Mist ratio (%)</th>
<th>$T_w$ (K)</th>
<th>Droplet BC</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Base case</td>
<td>30</td>
<td>10</td>
<td>900</td>
<td>Reflect</td>
</tr>
<tr>
<td>2</td>
<td>40</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Base case</td>
<td>30</td>
<td>10</td>
<td>900</td>
<td>Reflect</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>15</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Base case</td>
<td>30</td>
<td>10</td>
<td>900</td>
<td>Reflect</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td></td>
<td>800</td>
<td></td>
</tr>
<tr>
<td>Base case</td>
<td>30</td>
<td>10</td>
<td>900</td>
<td>Reflect</td>
</tr>
<tr>
<td>6</td>
<td></td>
<td></td>
<td>1000</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td></td>
<td></td>
<td></td>
<td>Wall-jet</td>
</tr>
<tr>
<td>Base case</td>
<td>30</td>
<td>10</td>
<td>900</td>
<td>Reflect</td>
</tr>
<tr>
<td>8</td>
<td></td>
<td></td>
<td></td>
<td>Wall-film</td>
</tr>
</tbody>
</table>

### 4.2.3.4 Meshing and simulation procedure

The computational domain is constructed by unstructured hexahedral elements as shown in Fig. 4.3. This type of cell falls between the structured hexahedral and the unstructured tetrahedral in terms of the accuracy of the solution obtained. The computational domain geometry is decomposed into 5 sub-regions before being meshed with the Cooper scheme that yields this kind of mesh. More intensive meshes are used near the walls, especially the cavity walls and the film holes, in order to give a value of $y^+ \approx 30$ which is reasonable for the turbulence.
model selected. Four mesh densities are tested for grid independence, and the base case is solved for every mesh. The results are compared and summarized in Table 4.5 for selection of the appropriate mesh size.

Table 4.5 Grid independency analysis

<table>
<thead>
<tr>
<th></th>
<th>50K Cells</th>
<th>100K Cells</th>
<th>211K Cells</th>
<th>460K Cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dry Exit Static Temperature (K)</td>
<td>621.9</td>
<td>609.7</td>
<td>611.8</td>
<td>608.2</td>
</tr>
<tr>
<td>Max. Predicted Residence Time (s)</td>
<td>0.398</td>
<td>0.378</td>
<td>0.427</td>
<td>0.434</td>
</tr>
<tr>
<td>D_{10} (μm)</td>
<td>18.96</td>
<td>18.99</td>
<td>18.93</td>
<td>18.32</td>
</tr>
<tr>
<td>D_{32} (μm)</td>
<td>21.03</td>
<td>21.06</td>
<td>20.98</td>
<td>20.94</td>
</tr>
</tbody>
</table>

Actually, all solution variables give the same trend but only selected ones are presented here. For the dry case, the most relevant solution variable is the static temperature of the air at the film holes, so mass weighted average value over all holes is selected as a criterion for the comparison. For the wet case (with droplets), residence time, arithmetic mean diameter (D_{10}), and Sauter mean diameter (D_{32}) are the selected variables to be monitored. Comparing the variable values at the last two columns of Table 4.5, it can be seen that the solution has achieved variations within 0.1%-3% between the last two mesh sizes. This suggests that the grid of 211,000 cells gives the cost-effective choice and hence, it is used for the remaining parametric study.

The simulation is carried out using the commercial CFD software ANSYS12.0 from Ansys, Inc. The simulation uses the segregated solver, which employs an implicit pressure-correction scheme and decouples the momentum and energy equations. The SIMPLE algorithm is used to couple the pressure and velocity. Second order upwind scheme is selected for spatial discretization of the convective terms and species. The computation is conducted for the main and coolant flow field (continuous phase) first. After obtaining an approximate converged flow field of the air, the dispersed phase of droplet trajectories are calculated. At the same time, drag, heat, and mass transfer between the droplets and the air are calculated. Iterations proceed alternatively between the continuous and discrete phases. Converged results are obtained after the residuals satisfy mass residual of 10^{-3}, energy residual of 10^{-6}, and momentum and turbulence
kinetic energy residuals of $10^{-4}$. These residuals are the summation of the imbalance for each cell, scaled by a representative of the flow rate.

4.2.4 Results and Discussion

4.2.4.1 Model validation

The mist/steam cooling CFD scheme and model have been validated by the same research group members Dhanasekaran and Wang (Dhanasekaran and Wang, 2008, 2010, and 2011; Wang and Dhanasekaran, 2010) with the experimental data in conditions like flow in heated tubes (Guo et al., 2000 I, II), 180-degrees tube bend (Guo et al., 2000), and impinging jets (Wang et al., 2005). The mist/air cooling CFD models have been qualified by Li and Wang (2006, 2007, and 2008). The same CFD scheme and model are used in this study, hence no validation is repeated here.

4.2.4.2 Base case results

In the current work, a numerical simulation is performed with the base case conditions shown in Table 4.3. The flow field in the dry base case is solved first without droplets until a converged solution is obtained then the droplets are injected according to specifications in Table 4.3 for the mist base case. Figures 4.4 and 4.5 show the static temperature contours at a mid-plane for the dry and mist base cases, respectively.

Figure 4.4 Contours of the static temperature [K] at the mid-plane in the dry base case (No water mist is injected)
Due to the droplets evaporation and boiling, latent heat of vaporization is absorbed from the main flow resulting in a remarkable reduction, approximately 200 degrees, in the air temperature. This is a very important result as it implies a high cooling potential for the mist cooled cavity air. As the air temperature is reduced, the amount of air required for cooling the engine components, which is bled from the compressor, can be notably reduced and the net output power can be augmented. Also, saving the precious compressor air helps to increase the thermal efficiency of the gas turbine engine. Another factor that leads to savings in the compressor air is that the specific heat of air/water vapor mixture is higher than that of air alone. This means a higher cooling capacity of the mist-air and hence a lower compressed air consumption. The final contribution from the mist comes from its higher film cooling effectiveness which can help extend the life of the turbine components.

Figure 4.6 shows the contours of water vapor mass fraction from the evaporation and boiling of the droplet streams under the base case conditions.
Figures 4.7 and 4.8 show the droplet traces colored by droplet diameter (cm) and droplet residence time (s), respectively. It can be noticed that trajectories close to the wall vanish early in the domain. A thin layer of water vapor with high Cp value is therefore formed and shields the core flow from the hot wall, thus leaving the core flow cooler to sustain the water droplets longer. *This key result supports the zero-dimensional result that droplets can survive up to the exit cooling holes.* Again, the main goal of the current study is to prove that droplets with reasonable diameters and loading can survive in this very hot environment until they reach the cooling holes. Moreover, the droplets that exit the cooling holes should be of sufficient mass (or diameter) to sustain the mist cooling over the external vane surface. Please note that not all of the droplet streams are shown in Figs. 4.7 and 4.8. The CFD solution helps show the flow behavior in the cooling cavity up to the cooling holes. It is noticed that the number of droplet trajectories is low at the upper 7 holes. This happens as a result of the flow separation in the vicinity of the inlet region of the cooling cavity. This flow separation happens due to the presence of the stagnation zone near the “end wall” at the end of the air cooling duct. This separation region deflects the droplets away from the cavity wall for the upper 7 holes. This problem can be mitigated by designing a tapered end wall if so designed.
Figure 4.7 Droplet traces colored by droplet diameter [cm] in the base case (10% Mist Ratio, 30 µm initial diameter, 900 K cavity wall temperature)

Figure 4.8 Droplet traces colored by residence time [S] in the base case (10% Mist Ratio, 30 µm initial diameter, 900 K cavity wall temperature)

Figure 4.9 shows the droplet diameter distribution at exit holes in the base case. The mean droplet diameter can be evaluated in various ways. The most commonly used mean diameters are the Arithmetic Mean Diameter, $D_{10}$, and the Sauter Mean Diameter, $D_{32}$. Both diameters can be defined as follows:

$$D_{10} = \frac{\sum_{i=1}^{n} d_i}{n}, \quad D_{32} = \frac{\sum_{i=1}^{n} d_i^3}{\sum_{i=1}^{n} d_i^2}$$

(4.20)
Where $n$ is the total number of droplets. The Sauter mean diameter is of interest in applications where the active surface area is important, as in spray evaporation applications. Droplets with arithmetic mean diameter of 18.93 $\mu$m and mean Sauter diameter of 20.98 $\mu$m are obtained. These diameters are suitable for mist/film cooling over the external vane surface as previously studied by References (Takagi and Ogasawara, 1974; Mori et al., 1982; Jansen et al., 1986; Guo et al., 2000 I, II).

![Figure 4.9 Diameter [m] distribution histogram at exit holes in the base case (10% Mist Ratio, 30 $\mu$m initial diameter, 900 K cavity wall temperature)](image)

**Figure 4.9 Diameter [m] distribution histogram at exit holes in the base case (10% Mist Ratio, 30 $\mu$m initial diameter, 900 K cavity wall temperature)**

### 4.2.4.3 Parametric study

The required outcome of this work is to prove the existence of droplet streams with reasonable diameters (in the order of 15-20 $\mu$m) at film holes. To understand well the effect of changing operating parameters on the droplet distribution at the exit, a parametric study is performed. Mist ratio (ratio of the mass flow rate of water mist to the air mass flow), initial mean diameter of injected droplets, cavity wall temperature, and wall droplet boundary condition are the studied parameters. Although the real droplet diameters are polydispersed, all the cases are performed assuming a uniform (monodispersed) droplet diameter distribution. This simplification is necessary for exercising a controlled study to track change of droplet's size. To show the effect of injected diameter distribution, one case is calculated with injected droplet diameters follow the Rosin-Rammler distribution (Rosin and Rammler, 1933).
4.2.4.3.1 Effect of changing injection mist ratio

Particle loading is an important parameter in any two-phase flow application. Mist ratio, defined as the ratio of mass flow rate of water mist at injection location to that of air, is used here to express the effect of particle loading. As shown in Figs. 4.10 (a) and (b), reducing the mist ratio from 10% to 5% causes almost all the droplets to evaporate leaving only 2 droplet streams out of 8160 to reach the exit holes. This is clearer in Fig. 4.11, where the mean droplet diameter at exit is reduced to 1.71 µm compared with 21.26 µm at 15 % mist ratio. This suggests that at least 10 % mist ratio is required for Frame 7FA gas turbine vanes to provide the necessary amount of mist that can be used for film cooling.

![Figure 4.10 Diameter [m] distribution histogram at exit holes with (a) 5% Mist Ratio (b) 15% Mist Ratio](image)

*Figure 4.10 Diameter [m] distribution histogram at exit holes with (a) 5% Mist Ratio (b) 15% Mist Ratio*

![Figure 4.11 Effect of changing Mist Ratio (%) on droplet exit diameter.](image)

*Figure 4.11 Effect of changing Mist Ratio (%) on droplet exit diameter.*
4.2.4.3.2 Effect of changing injection (initial) diameter

As shown in Figs. 4.12(a) and 4.12(b), injecting 10% mass ratio of droplets with initial diameters 30µm (or larger) is necessary to get 20 µm (or larger) at exit holes. This trend is clearer in Fig. 4.13, where the mean droplet diameter at exit increases to 25.77 µm with initial droplet diameter of 40 µm compared with 12.65 µm in final average droplet diameter with 20 µm initial diameter. Also, the relation between the initial and final diameters seems to be linear, as shown in Fig. 4.13 with a linearity constant of approximately 0.628 for the arithmetic mean.

![Diameter distribution histogram at exit holes with (a) 20 µm initial diameter (b) 40 µm initial diameter.](image)

Figure 4.12 Diameter [m] distribution histogram at exit holes with (a) 20 µm initial diameter (b) 40 µm initial diameter.

![Effect of changing droplet injection (Initial) diameter on droplet exit diameter](image)

Figure 4.13 Effect of changing droplet injection (Initial) diameter on droplet exit diameter
A similar situation happens for the mean Sauter diameter. This can help in estimation of the required inlet diameter to achieve a certain cooling effect associated with a certain exit diameter. These results suggest that droplets with diameters ranging from 30-20 µm and a mist mass ratio of 10% are suitable to achieve the droplet sizes in the range of 10-20 µm that are more effective in mist/film cooling for Frame 7FA gas turbine vanes. This diameter range is suitable from heat and mass transfer point of view as it enables the droplets to survive until they reach the exit holes and complete their trip around the vane external surface. From the aerodynamics point of view, this diameter range has a small Stokes number, which means that it will follow the main stream and reduce its chances of impinging at walls. This in turn would reduce the concerns of erosion and localized thermal stresses.

4.2.4.3.3 Effect of changing cavity wall temperature

In real gas turbines, the blade cooling process is a complex conjugate heat transfer problem. Hence, the cavity wall temperature depends on both the internal flow cooling and heating from the external flows along with the wall material. A more precise way is to calculate the flow fields and heat transfer both inside and outside the walls. However, since the present study is only focused on the droplet evaporation in the internal flow, the wall temperature has been assigned as a boundary condition. Since the conjugate heat transfer problem is not solved, a range of wall temperature between 1000 K and 800 K is hereby assigned to bracket the wall temperature that could occur in the real condition. The results of droplet diameter distribution at exit holes are shown in Fig. 4.14. It can be concluded that the droplet diameter reduces within 1µm with the increase of cavity wall temperature from 800 K to 1000 K.

Figure 4.14 Effect of changing cavity wall temperature on droplet exit diameter
4.2.4.3.4 Effect of changing wall droplet boundary condition

Generally, wall boundary condition plays a significant role in droplet trajectory calculation in wall-bounded flows. As the droplet hits the wall, the droplet encounters a change in velocity and/or direction according to the wall boundary condition selected. In the base case, the simple reflect boundary condition was selected. This boundary condition calculates the reflection angle of the droplet based on its incidence angle assuming that velocity remains unchanged, i.e., elastic impaction. The results of the base case are presented earlier in Fig. 4.9. In the parametric study, another two boundary conditions are used, namely, wall-film and wall-jet boundary conditions with the details of those models discussed earlier. The results show that the droplets’ diameter distribution at exit in the wall-film and wall-jet boundary conditions cases are almost identical to that of using the reflect wall condition as shown in Fig. 4.9. The averaged diameters of the three conditions are almost identical, as compared in Fig. 4.15. It can be concluded that the droplet boundary condition at the wall has a negligible effect on the droplet diameter distribution at exit which is expected in the current work. As the wall is highly superheated and at a much greater temperature than the water boiling temperature, it is unlikely to have a wall-film forming on it due to the deposition or sticking of water droplets (Stanton and Rutland, 1996; O’Rourke and Amsden, 2000). Therefore, it is not unexpected that different droplet wall boundary conditions do not render results that differ by very much.

![Figure 4.15 Effect of changing wall droplet boundary condition on droplet exit diameter distribution](image)

168
4.2.4.3.5 Effect of droplet initial diameter distribution

The effect of droplet diameter distribution at inlet is studied by first injecting uniform diameters of 20, 30, and 40 µm, respectively, to gain a clear understanding of the droplet size effect under a controlled condition. This is followed by injecting distributed droplet diameters to simulate more closely the actual atomized droplet condition. The Rosin-Rammler distribution function (Rosin and Rammler, 1933) is used based on the assumption that an exponential relationship exists between the droplet diameter, \( d \), and the mass fraction of droplets with diameter greater than \( d \) as follows:

\[
Y_d = e^{-\left(\frac{d}{d_m}\right)^n}
\] (4.21)

where \( d_m \) refers to the mean diameter 30 µm and \( n \) refers to the spread parameter. From the relationship, the spread parameter of 5.94 is calculated and used to fit the size distribution into the CFD model. Basically, five diameter sets are used to express the whole range of droplets, and a spread parameter, \( n \), is calculated for each set from using Eq. (4.21). An average value of 5.94 is obtained from Eq. 4.21 as the spread parameter in the current study. The result of using the Rosin-Rammler distribution as the initial droplet size distribution in Fig. 4.16 shows that droplet distribution at exit tends to be closer to the larger diameter case. This happens because smaller diameters in the distribution (from 0 to 30 µm) tend to evaporate quickly leaving only the droplets with larger diameters to survive the journey.

![Figure 4.16 Diameter [m] distribution histogram at exit holes with Rossin-Rammler initial diameter distribution](image)

**Figure 4.16 Diameter [m] distribution histogram at exit holes with Rossin-Rammler initial diameter distribution**
4.2.5 Conclusions

From the current investigation, some conclusions can be drawn as follows:

- The mist cooling for high-pressure gas turbine vane is feasible under real operating conditions.
  
  For example, under real Frame 7FA operating conditions, 50% of the mist can survive with an average droplet diameter of 10 – 20 µm if mist with 10% mass ratio and 20-30µm in initial diameter is injected.

- Due to the large superheated wall temperature, a thin layer of water vapor with higher Cp value than air is formed near the wall. This vapor layer shields the core flow from the hot wall, thus leaving the core flow cooler to sustain the water droplets flying longer.

- The relation between the initial and final injected diameters is found to be linear, while the relation between mist ratio and final diameter is not. This gives some control of the operating conditions of the mist cooling technique.

- The effect of different wall droplet boundary conditions was found inconsequential.

The results obtained in this study are very encouraging and warrant a continuous study to find a method to transport the mist to the rotating blade. Furthermore, the mist-induced cooling enhancement may introduce local cold spots of overcooling. Therefore, development of a coupled multi-phase CFD model and finite-element stress analysis will be a future task.
4.3 INVESTIGATION OF APPLICABILITY OF USING WATER MIST FOR COOLING HIGH-PRESSURE TURBINE COMPONENTS VIA ROTOR CAVITY FEED CHANNELS

4.3.1 Background

There have been numerous studies in the past decades that have focused on mist film cooling over flat surfaces or turbine airfoil surfaces with streamwise coolant injection (Takagi and Ogasawara, 1974; Mori, et al., 1982; Jansen et al., 1986; Guo et al., 2000 I, II; Nazarov et al., 2009; Pakhomov et al., 2010; Wang, et al., 2005; Nirmalan et al., 1996; Li and Wang, 2006; Dhanasekaran and Wang, 2009, 2011, and 2012). While most of the above studies were conducted at lab conditions of low temperatures and pressures, their results showed that the mist film cooling is a promising and effective cooling technique. However, all the previous studies focused only on investigating the mist cooling performance, none of them studied how to transport the mist to the target component sites or answered the question of whether is it possible to transport water mist to different engine components at real engine conditions. In other words, can the mist survive the extremely hot conditions inside the flow passages in the gas turbine and be successfully delivered to the needed sites? A series of CFD research has been undertaken by the authors to answer this question. The first study (Ragab and Wang, 2012) was performed using CFD (computational fluid dynamics) to investigate the possibility of transporting water mist to vanes of a high pressure turbine stage under real operating conditions, as been detailed in Sec 4.1. The droplets could travel from the injection location, located in the air cooling duct, coming from the compressor discharge, all the way to the film holes at the vane surface. The results showed that water droplets of 30 µm and 10% mass loading ratio of the cooling air were able to survive and reach the film cooling holes of Frame 7 turbine vanes with an average diameter of 10 µm. Furthermore, calculations showed that these survived droplets will be able to complete the external film cooling around the vane surface. Although the research is only numerical, the results are very encouraging and, to a great extent, are sufficient to encourage experimental validation of the CFD results.

Figure 4.1 shows a schematic diagram of a typical two stage high pressure film cooled turbine. The relatively cold compressor air (600 K) is bled from the compressor discharge through the cooling air passage where it enters the cooling channels inside the turbine vanes to cool the vanes' walls, which are toasted by extremely hot flu gas at about 1300 K (2350°F).
Eventually, the air exits the film cooling holes to form a protective layer around the external vane surface to prevent it from direct contact with the hot gases. To retrofit this existing turbine with the mist cooling techniques, it is proposed that atomizers be located at a short distance just upstream of the vane's leading edge. The water mist is to be injected at the inlet of the cooling air passage along the casing. The tubing of the high-pressure water is directly inserted through the turbine casing. For the rotor, the situation is a bit difficult because of the rotation. The exact path of the compressed water tubing depends on the engine design and how the cooling air is fed to the rotor film holes (Ragab and Wang, 2012; Snowsill and Young, 2006; Kurzke, J., 2007, Gupta et al., 2008; El-Sadi et al., 2007).

Basically, there are two main designs. In one of the designs, the cooling air is supplied to the first rotor blades through the second stage vane through a diaphragm box inboard of the airfoil inner endwall. This Outboard Air Supply route is shown in Figure 4.17 marked with the blue arrow. Although this passage is not considered in this study, it is important to highlight it as an alternative passage and is currently investigated separated for another study.

![Figure 4.17 Heavy frame turbine secondary flow regions (Dennis, 2006)](image)

In the other design of the Onboard Air Supply passage, the cooling air, which is bled from the compressor exit, is introduced to a preswirl nozzle where it is accelerated tangentially in the direction of the rotor rotation. This acceleration reduces the relative total temperature of the
cooling air and hence increases the film cooling efficiency and components lifetime. The performance of this preswirl nozzle is crucial for the engine components’ life expectancy. It is believed that a 20° K increase in relative total temperature of the air fed to the rotor is sufficient to reduce component lives by as much as 50% (Snowsill and Young, 2006). This motivates the current study by routing the mist through preswirl nozzle and rotor disk cavity. In this second design, the atomizer is proposed to be located somewhere in the preswirl nozzle chamber, as shown in Fig. 4.1. To deliver the compressed water to that proposed location, a feed channel is inserted through the turbine casing and passes through the engine stationary part (first vane) all the way to the atomizer, as shown in Fig.4.1.

To deliver the compressed water to the proposed location, a feed channel is inserted through the turbine casing and passes through the engine stationary part (first vane) all the way to the atomizer. Now, the compressed water is introduced to the atomizer through the feed channel, and the atomized water droplets will mix with the compressor bleed air to form a mixture of air/mist. This mixture will serve in cooling the blade disc and the seals and will travel through the rotor cover-plate cavity to, hopefully, reach the rotor blades. Based on this description of the mechanism used to deliver the compressed water to the atomizer location, the geometry of the channel along with the operating boundary conditions are introduced in Fig. 4.19.

After the previous clarification and description of the mechanism of the mist cooling for the rotor blades, some new questions arise. Will boiling occur as the feed channel passes through the extremely hot vane? Typically, the boiling phenomenon is undesirable in gas turbine engines for many reasons: (a) Boiling is an unstable phenomenon which causes vibrations and generates structural stresses. The problem is further exacerbated in the case of high speed rotating machinery like turbines; (b) Boiling converts a fraction of the compressed liquid water into vapor, which will adversely affect the atomization process in the atomizer and lead to reduced mist cooling.

Boiling is a complex phenomenon and its modeling is a challenging task. Figure 4.18 shows a conventional textbook schematic for the basic phenomenon occurring in forced convection boiling, which exemplifies what would happen when a liquid is forced over a heated surface like that in the feed channel studied in this work. Onset of Nucleate boiling (ONB) and Onset of Significant Boiling (OSB) are two important physical phenomena that the CFD code could reasonably predict, although not always successfully. According to the temperature of the
liquid, convective boiling can be classified as **Saturated boiling** and **Subcooled boiling**. Saturated boiling occurs on surfaces immersed in a liquid which is at the saturation temperature. In subcooled boiling, the average liquid temperature stays below the saturation value, producing local boiling at the wall (surface- or micro-boiling) with subsequent condensation of the vapor as it departs from the wall and moves into the colder bulk of the fluid. The physics of boiling implies the formation of bubbles on the hot surface, **surface boiling**, which happens when the surface temperature is slightly above the saturation temperature of the single phase liquid medium. Vapor bubbles form at the nucleation sites on the wall, and they continue to increase in size. With the continuous addition of heat, bubbles detach and separate from the wall surface. As the degree of subcooling decreases along the channel and, consequently, so also does the rate of condensation, the steam bubbles formed will be able to penetrate further into the subcooled flow core, increasing the turbulent transport of heat and changing the pattern of temperature distribution over the cross section of the flow core. After more bubbles are generated, a **bubbly flow regime**, with velocity higher than the single phase liquid velocity, will form. As depicted in Fig. 4.18, the **bulk boiling** evolves until all the liquid changes to superheated vapor. Again, different physics in each regime requires different sub-models to be adequately predicted.

**Figure 4.18** Conceptual picture of forced convective boiling (a) different phase-change regimes with a uniform heat flux boundary condition (thermopedia.com) (b) subcooled boiling schematic
It is more challenging, in the meantime, to have a generalized multiphase CFD model for boiling flow and heat transfer. Most of the sub-models proposed so far account only partially for the relevant physics such as the onset of nucleate boiling (ONB), departure from nucleate boiling (DNB), critical heat flux (CHF) and post dry-out. This situation is further complicated when considering critical heat flux conditions as flow regime transitions need to be taken into consideration (Bestion, D., 2007; Anglart et al., 1997; Lu et al., 2008).

Over the last decade or so, the Eulerian multifluid method with the so-called RPI wall boiling model, developed by Kurul & Podowski in Rensselaer Polytechnic Institute (Kurul and Podowski, 1991), has been established as a recognized modeling approach for mechanistic prediction of boiling phenomena. Attempts have since been made to extend this approach towards modeling CHF and post-dryout. Indeed, such boiling models have been implemented in designing heat exchangers and nuclear equipment (Kurul and Podowski, 1991; Burns et al., 2007; Lavieville et al., 2005; Tentner et al., 2006; Ioilev, 2007; Li, H., 2010; Macek and Vyskocil, 2008; Troshko and Hassan, 2001). However, it is commonly recognized that though promising, all the CFD multiphase boiling modeling approaches are still at the stage of development and validation when dealing with the prediction of practical boiling flow and heat transfer. Efforts to further improve the model accuracy and numerical robustness are ongoing.

4.3.2 Objective and Scope

The purpose of the current work is to use CFD to predict whether boiling will occur in the feed channel. If it does, the inception of boiling phenomenon will be examined and the design parameters will be customized to minimize, or suppress, the boiling to ensure that the feed water will be transported to the atomizer in a state suitable for effective atomization. Hence, the mist/air cooling technique will work for cooling the rotating components and seals. Since the two-phase boiling phenomena and CFD predication capability have been well developed in heat transfer community with applications in heat exchanger design and nuclear industry, it is not the purpose of this work to thoroughly study the boiling phenomena and its modeling issues. Thus, the methodology is to use the available capabilities of existing CFD codes to reasonably predict boiling. This will help to identify potential technical issues that could challenge implementation of the mist/cooling technique in real gas environment.
In the sections below, the general governing equations and physical models are first presented. They are followed by a brief description of the numerical approach to deal with the boiling flow. The Rensselaer Polytechnic Institute's (RPI) model is then employed through a systematic study of a benchmark case with boiling flow in an axisymmetric circular channel. After validation of the benchmark case, the model is applied to predict boiling in the current problem, which is very close in physics and operating conditions to the benchmark case. Parametric study is then performed to include changing factors that affect boiling. Before conducting numerical simulations, it is customary to use simple 1-D empirical correlations to provide an estimate of the boiling phenomenon that may occur in the feed channel under the benchmark operating condition. The 1-D result also provides a guidance for parametric study in CFD simulation. The 1-D model selected is a well known empirical correlation frequently used in convection heat transfer calculations.

Figure 4.19 shows a schematic of the feed channel to be used and represents the studied geometry, for the numerical model, in this work. The feed channel is proposed to be of ¼ inch ID and 75 cm long with a pressure of 1000 Psi (68 atm). Under the operating conditions of a Frame 7 engine, as will be detailed later, the flow conditions are specified in Fig. 4.19.

**Figure 4.19 Geometry and flow conditions in the simulated pressurized liquid water feed channel**

### 4.3.3 One-Dimensional Model

A quick estimate of single phase water temperature in a heated channel can be done using a one-dimensional model equation. The purpose is to have an estimate of the conditions of
water at the end of the feed channel. A simple and well documented model of Dittus-Boelter (Incropera and DeWitt, 2002) will be used. For the present study of a typical Frame 7 engine the total air mass flow rate is approximately 445 kg/s with 3% of that mass flow directed for cooling the 70 rotors in the first stage of the high pressure turbine. Air mass flow rate for each rotor cooling channel can be calculated as 0.1907 kg/s. Liquid water equivalent of 10% of the cooling air mass flow, i.e., 0.01907 kg/s, is added for mist cooling for each feed channel. For the ¼ inch feed channel with 1000 psia operating pressure, the water velocity, Reynolds number, and Prandtl number are 0.696 m/s, 27,300, and 0.948, respectively. Nusselt number can be calculated using Dittus-Boelter Equation 4.22a (Incropera and DeWitt, 2002) as follows:

\[
\text{Nu}_D = 0.023 \text{Re}^{4/5} \text{Pr}^n
\]

(Valid for Turbulent, fully developed flows, \(0.6 \leq \text{Pr} \leq 160; \text{Re}_D \geq 10,000; L/D \geq 10; \) and \(n = 0.4\) for heating \((T_w > T_m)\))

Finally, the water mean temperature distribution inside the channel, without boiling, can be calculated from the following equation (Incropera and DeWitt, 2002):

\[
\frac{\Delta T_o}{\Delta T_i} = \frac{T_w - T_{m,x}}{T_w - T_i} = \exp \left( - \frac{P \rho h}{m_w c_p} \right)
\]

(4.22b)

Where \(T_w = 650\) K is the wall temperature (assumed), and \(P\) is the channel perimeter.

Figure 4.20 shows the mean water temperature distribution inside the feed channel using the one-dimensional single phase model for different inlet water temperatures. Although it is not correct to use a single phase model to calculate the average liquid temperature for a flow that may undergo boiling, it serves as a good tool to indicate if boiling will occur by examining if the liquid temperature will rise above the saturation temperature. Based on this principle, three water inlet temperatures (473 K, 350 K, and 300 K) are assigned; the single phase liquid temperature distributions along the channel are shown in Fig. 4.20. The exit temperatures of these cases are 609 K, 578 K, and 566 K, respectively. Comparing them with the saturation temperature 558 K at 1000 psia, this means that phase change will happen in the first two cases, but only mild phase change will happen in the end of the channel. This also implies that an inlet water temperature less than 300 K could avoid boiling.
4.3.4 CFD Calculations

The simple one-dimensional calculation results showed that the phase change process is likely to happen. To obtain more detailed results, comprehensive CFD calculations are performed with the RPI wall boiling model included.

4.3.4.1 Mathematical modeling
4.3.4.1.1 Basic governing equations

In case of boiling, the vapor volume fraction \( V_f \) increases to a limit exceeding the underlying assumptions of being a discrete phase \( (V_f > 0.1) \). This requires that both vapor bubbles and liquid are modeled as separate phases. The present Eulerian/Eulerian multiphase boiling model has been developed to predict boiling within the CFD solver ANSYS/FLUENT 14.0. A set of governing equations for n-phase multi-fluid flows is solved. Wall boiling phenomenon is modeled using the mechanistic RPI boiling model. A range of sub-models are considered to model the interfacial momentum, mass and heat transfer, and turbulence-bubble interactions. The detailed governing two-phase governing equations are presented here.

Introducing \( q \) to represent the \( q^{th} \) phase, the generalized phase governing equations have the following forms, respectively:

**Phase mass conservation**

\[
\frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q) = \sum_{p=1}^{n} (\dot{m}_{pq} - \dot{m}_{qp}) + S_q \tag{4.23}
\]
Where $\alpha_q$ is the volume fraction, $\vec{v}_q$ is the velocity of phase $q$ and $m_{pq}$ characterizes the mass transfer from the $p^{th}$ to $q^{th}$ phase. By default, the external mass source term (exerted on the phase $q$) $S_q = 0$.

**Phase momentum conservation**

$$\frac{\partial}{\partial t} (\alpha_q \rho_q \vec{v}_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q \vec{v}_q) = -\alpha_q \nabla P + \nabla \cdot \bar{\tau}_q + \alpha_q \rho_q \vec{g} + \sum_{p=1}^{n} \left( \bar{R}_{pq} + m_{pq} \vec{v}_{pq} - m_{qp} \vec{v}_{qp} \right) + \left( \bar{F}_q + \bar{F}_{\text{lift},q} + \bar{F}_{\text{vm},q} \right) \quad (4.24)$$

Where $\bar{\tau}_q$ is the $q^{th}$ phase stress-strain tensor,

$$\bar{\tau}_q = \alpha_q \mu_q \left( \nabla \vec{v}_q + \nabla \vec{v}_q^T \right) + \alpha_q \left( \lambda_q - \frac{2}{3} \mu_q \right) \nabla \cdot \vec{v}_q \vec{I} \quad (4.25)$$

Here $\mu_q$ and $\lambda_q$ are the shear and bulk viscosity of phase $q$, $\bar{F}_q$ is an external body force, $\bar{F}_{\text{lift},q}$ is a lift force, $\bar{F}_{\text{vm},q}$ is a virtual mass force, $\vec{v}_{pq}$ is the interphase velocity (relative velocity vector), $\bar{R}_{pq}$ is an interaction drag force between phases, and $P$ is the pressure shared by all phases. Equation 4 must be closed with appropriate expressions for the interphase force $\bar{R}_{pq}$ (will be introduced later). This force depends on the friction, pressure, cohesion, and other effects, and is subject to the conditions that $\bar{R}_{pq} = -\bar{R}_{qp}$ and $\bar{R}_{qq} = 0$.

**Energy conservation equation**

$$\frac{\partial}{\partial t} (\alpha_q \rho_q h_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q h_q) = \alpha_q \frac{\partial h_q}{\partial t} + \bar{\tau}_q \cdot \nabla \vec{v}_q - \nabla \cdot \vec{q}_q + S_q + \sum_{p=1}^{n} (Q_{pq} + m_{pq} h_{pq} - m_{qp} h_{qp}) \quad (4.26)$$

where $h_q$ is the specific enthalpy of the $q^{th}$ phase, $\vec{q}_q$ is the heat flux, $S_q$ is a source term that includes sources of enthalpy (e.g., due to chemical reaction or radiation), $Q_{pq}$ is the intensity of heat exchange between the $p^{th}$ and $q^{th}$ phases, and $h_{pq}$ is the interphase enthalpy (e.g., the enthalpy of the vapor at the temperature of the droplets, in the case of evaporation). The heat exchange between phases must comply with the local balance conditions $Q_{pq} = -Q_{qp}$ and $Q_{qq} = 0$. 179
4.3.4.1.2 RPI wall boiling model

One of the widely recognized wall boiling models is the multi-dimensional, multifluid RPI wall heat flux portioning model developed by Kurul & Podowski (Kurul and Podowski, 1991). According to the RPI model, the total heat flux from a wall to liquid is partitioned into three components: liquid phase convective heat flux, \( q_C \); quenching heat flux, \( q_Q \); and evaporation heat flux, \( q_E \):

\[
q_W = q_C + q_Q + q_E
\]  
(4.27)

Assuming that the heated wall surface is subdivided into a portion \( A_b \) covered by nucleating bubbles and the remaining part \((1 - A_b)\) occupied by fluid, the RPI model gives the following expressions for the three heat flux components, see Fig. 4.21:

\[
q_C = h_c(T_w - T_l)(1 - A_b)
\]  
(4.28)

Where \( h_c \) is the liquid phase heat transfer coefficient, and \( T_w \) and \( T_l \) are the wall and liquid temperature near the wall, respectively.

\[
q_Q = \frac{2k_l}{\sqrt{\pi\lambda_lT}}(T_w - T_l)A_b
\]  
(4.29)

This term models the cyclic averaged transient energy transfer related to liquid filling the wall vicinity after the bubble detachment with a period of \( T \). Where, \( k_l \) and \( \lambda_l \) are the heat conductivity and diffusivity in the liquid phase.
\[ q_E = V_d N_w \rho_v h_v f_{bw} \]  
(4.30)

Where \( f_{bw} = 1/T \) is the frequency of bubble departure; \( N_w \) is the active nucleate site density; \( \rho_v \) is the vapor density; and \( h_v \) is the latent heat of evaporation. \( V_d \) is the volume of the bubble based on the bubble departure diameter \( (V_d = \frac{\pi}{6} D_{bw}^3) \). The equations (4.28-4.30) need closure with the following parameters:

**Area of influence**

\[ A_b = \min \left( 1, K \frac{N_w \pi D_{bw}^2}{4} \right) \]  
(4.31)

\( K \) varies between 1.8 and 5 can be computed by the relation proposed by Del Valle and Kenning (Del Valle and Kenning, 1985)

\[ K = 4.8 \exp \left( - \frac{J_{a_{sub}}}{80} \right) \]  
(4.32)

And \( J_{a_{sub}} \) is the subcooled Jacob number, which is defined as:

\[ J_{a_{sub}} = \frac{\partial [\rho_v / T]^2}{\rho_v h_v} \]  
(4.33)

**Frequency of bubble departure:** is calculated as (Cole, 1960)

\[ f_{bw} = \frac{1}{T} = \frac{4 g (\rho_l - \rho_v)}{3 \rho_l D_{bw}} \]  
(4.34)

**Bubble departure diameter:** is calculated based on empirical correlations by Tolubinski and Kostanchuk (1970)

\[ D_{bw} = \min \left( 0.0014, 0.0006 \exp \left( - \frac{\Delta T_{sub}}{45.0} \right) \right) \]  
(4.35)

**Nucleate site density:**

\[ N_w = C^n (T_w - T_{sat})^n \]  
(4.36)

where \( n=1.805 \) and \( C=210 \), which are empirical parameters from Lemmert and Chawla (1977)

**Interfacial area,**

\[ A_i = \frac{6a_d (1-a_d)}{D_{d}} \]  
(4.37)

where subscript “\( d \)” indicates a dispersed phase, which is the vapor phase in bubbly flows.

**Bubble diameter**

In the bubbly flow regime, the bubble diameter, \( D_b \), can be given by Unal correlation (Unal, 1976) as a function of local subcooling
Interfacial Momentum Transfer

In boiling flows, the most important interfacial momentum transfers may include drag, lift and turbulent drift forces (Li, H., 2010; ANSYS, 2010).

**Interfacial drag force** – For dispersed bubbly flows, the interfacial drag force has the general form:

\[
\sum_{p=1}^{n} \mathbf{R}_{pq} = \sum_{p=1}^{n} \frac{A_i}{g} \rho_i C_D |\mathbf{V}_l - \mathbf{V}_v| (\mathbf{V}_l - \mathbf{V}_v) \tag{4.39}
\]

where the drag coefficient \(C_D\) can be computed by various models (ANSYS, 2010).

**Interfacial lift force** – The interfacial lift force has the general form (acting on a secondary phase \(P\) in a primary phase \(q\)):

\[
\mathbf{F}_{lift} = -C_l \rho_q \alpha_p (\mathbf{v}_q - \mathbf{v}_p) \times (\mathbf{v} \times \mathbf{v}_q) \tag{4.40}
\]

In most cases, the lift force is insignificant compared to the drag force. The coefficient for the interfacial lift force is calculated using a correlation proposed by Moraga et al. (1999)

**Turbulence drift force** – The turbulent drift force is calculated as (Troshko and Hassan, 2001)

\[
\mathbf{F}_{TD} = -C_{TD} \rho_c k_c \mathbf{v} \tag{4.41}
\]

\(k_c\) is the turbulent kinetic energy of a continuous phase (liquid). The turbulent disperse coefficient \(C_{TD}\) is, by default, set to 1.0.

**Interfacial Heat and Mass Transfer**

As bubbles departure from a wall and move towards subcooled regions, there is heat and mass transfer from the bubbles to the liquid. The interface (vapor)-liquid heat transfer is defined as

\[
\dot{q}_{lt} = A_i h_{lt}(T_{sat} - T_l) \tag{4.42}
\]

Where \(h_{lt}\) is based on the Ranz-Marshall correlation (Ranz and Marshall, 1952 I, II)

\[
h_{lt} = \frac{k_l}{D_d} (2 + 0.6 \text{Re}^{0.5} \text{Pr}^{0.33}) \tag{4.43}
\]
The interface (liquid)-vapor heat transfer is calculated using the method proposed by Lavieville et al. (Lavieville et al., 2005). It is assumed that the vapor retains the saturation temperature by rapid evaporation/condensation. The formulation is as follows:

\[
q_{vt} = \frac{\alpha_v \rho_v c_{pv}}{\delta_t} (T_{sat} - T_v)
\]

\(\delta_t\) is the time scale and is by default set to 0.05s.

Wall–vapor mass transfer – The evaporation mass flow is applied at the cell near the heated wall and it is derived from the evaporation heat flux, as follows:

\[
\dot{m}_E = \frac{q^*_E}{h_{lv} + c_p(T_{sat}-T_l)}
\]

Interfacial mass transfer – The interfacial mass transfer depends directly on the interfacial heat transfers. Assuming that all the heat transferred to the interface is used in mass transfer (i.e. evaporation or condensation), the interfacial mass transfer rate can be written as:

\[
\dot{m} = \dot{m}_{lt} + \dot{m}_{vt} = \frac{q^*_l + q^*_v}{h_{lv}}
\]

4.3.4.1.3 RPI model validation

In the gas turbine arrangement, the inward channel shown in Fig. 4.1 is directed in all the directions circumferentially. For the model validation benchmark case, an upward flow of subcooled water through a heated vertical pipe is selected, experimentally studied by Bartolemei and Chanturiya (1967). The validation case is very similar to that studied in this work, and the validation presents a good foundation for the upcoming investigations. The pipe has a diameter of 15.4 mm and a length of 2m. The experiments were carried out under the operating pressure of 45 bar. The subcooled water, with a subcooling of 60 K, enters from the bottom side and travels upwards through the tube. The heat flux applied on the tube surface is uniform and has the value of 570 kW/m². The inlet mass flux is 900 kg/m²-sec, and the Reynolds number is about 104,210. A 2D axisymmetric computational model of 7,815 cells has been constructed for the purpose of this validation. Three different turbulence models were selected for the computations—namely, The RNG K-ε Model, the Reynolds Stress Model (RSM), and the Shear Stress Transport K-ω Model (SSTK-ω). The liquid temperature distribution at the pipe axis is used for comparison as shown in Figure 4.22.
Although all the models under-predicted the liquid temperature near the middle of the pipe (≈ 0.8 m), the SST k-ω turbulence model was the most accurate, as shown in Fig 4.22. This departure at 0.8 m happens at the point where bulk boiling starts. The departure depends mainly on the near wall treatment method. This indicates that SST k-ω turbulence model offers the best performance in this case. One possible explanation is that in the SST k-ω model, the wall approximation approach is scaled with local near-wall grids, while in all the other turbulence models, the wall treatment method is chosen in advance, which may not be suitable for the two-phase flow development in this study. More information about the SST k-ω model and its transport equations can be found in (Li, H., 2010; ANSYS, 2010). From this validation, it can be stated that the SST k-ω model seems to give an overall better solution. The grid sensitivity study shows that the results between using 7815 cells and 22,872 cells are within 1.7 %. This validation serves as a good foundation for further investigations in this study.

4.3.4.2 The Computational domain

The computational domain consists of a feed channel proposed to transport the compressed liquid water from the turbine outer casing through the vane to the atomizer located below the vane pedestal to achieve the mist generation, as shown in Fig. 4.19. The channel is 0.00635 m (¼ in) inside diameter and 0.75 m long. The channel is vertical with water at a pressure of 68 atm flowing upward. As the problem is axisymmetric, the domain simulated is only a 2-D slice with a width equal to the channel radius. 2-D axisymmetric solutions are known.
to be a cost effective alternative for analyzing full 3-D simulations. The computational domain and the mesh used are depicted in Fig.4.23.

**Figure 4.23 The computational domain, boundary types, and meshes**

### 4.3.4.3 Boundary conditions

The primary phase is water with properties varies with temperature, and the secondary phase is vapor with properties fixed at saturation temperature (558 K) and pressure at 1000 psi (68 atm). The boiling phenomenon is studied at comparable real working conditions of a Frame 7 gas turbine engine. As explained before in the 1D analysis, the water velocity, Reynolds number, and Prandtl number can be calculated as 0.696 m/s, 27300, and 0.948, respectively.

Based on the previous calculations, the velocity of 0.696 is imposed as a boundary condition at inlet for the liquid phase. Liquid water is preheated to 473 K before entering the channel to avoid potential "thermal shock" on the thermal stresses on the casing or vane walls. This pre-heat condition of water can be relaxed if thermal shock is not an issue. Mist cooling will be more effective if no pre-heat is imposed. For the vapor phase, zero vapor is assigned at the inlet. The turbulence parameters at inlet are specified in terms of the turbulence intensity, TI, of 4 % and the hydraulic diameter, HD, equal to channel diameter of 0.00635m. At exit, the static pressure of 68 atm is imposed (design parameter). The outer wall is assumed isothermal with a temperature of 650 K (an estimate in Frame 7 engines), while the inner wall is assigned as a conjugate wall (separating the fluid and the solid zones). The parameters used in base case are summarized in Table 4.6. A parametric study is performed to investigate the effect of wall temperature, water pressure, inlet water subcool temperature, and channel length on the boiling characteristics in the feed channel as summarized in Table 4.7. As mentioned earlier, the purpose
of the parametric study is to seek available means to suppress boiling by customizing the design parameters of the feed channel like the four parameters listed in Table 4.7.

### Table 4.6 The base case values

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water (Primary Phase)</td>
<td></td>
</tr>
<tr>
<td>Water inlet velocity</td>
<td>0.696 m/s</td>
</tr>
<tr>
<td>Water inlet temperature</td>
<td>473 K</td>
</tr>
<tr>
<td>Operating pressure</td>
<td>68 atm ($T_{sat}=558$ K)</td>
</tr>
<tr>
<td>Inlet subcooling</td>
<td>85 K</td>
</tr>
<tr>
<td>Channel length</td>
<td>0.75 m</td>
</tr>
<tr>
<td>Channel wall temperature</td>
<td>650 K</td>
</tr>
<tr>
<td>Vapor (Secondary Phase)</td>
<td></td>
</tr>
<tr>
<td>Vapor inlet velocity</td>
<td>0 m/s</td>
</tr>
<tr>
<td>Vapor Temperature</td>
<td>558 K</td>
</tr>
</tbody>
</table>

### Table 4.7 The matrix of parametric studies

<table>
<thead>
<tr>
<th>Case</th>
<th>$T_w$ (K)</th>
<th>P (atm)</th>
<th>$\Delta T_{sub}$</th>
<th>L (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>600</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>625</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Base case</td>
<td>650</td>
<td>68</td>
<td>85</td>
<td>0.75</td>
</tr>
<tr>
<td>3</td>
<td>665</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>60</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Base case</td>
<td>650</td>
<td>68</td>
<td>85</td>
<td>0.75</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td>80</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td></td>
<td>110</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td></td>
<td></td>
<td>15</td>
<td></td>
</tr>
<tr>
<td>Base case</td>
<td>650</td>
<td>68</td>
<td>85</td>
<td>0.75</td>
</tr>
<tr>
<td>8</td>
<td></td>
<td></td>
<td>110</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td></td>
<td></td>
<td>208</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td>0.50</td>
</tr>
<tr>
<td>Base case</td>
<td>650</td>
<td>68</td>
<td>85</td>
<td>0.75</td>
</tr>
<tr>
<td>11</td>
<td></td>
<td></td>
<td></td>
<td>1.00</td>
</tr>
</tbody>
</table>
4.3.4.4 Meshing and simulation procedure

The computational domain is constructed by structured hexahedral elements as shown in Fig. 4.23. The computational domain geometry is decomposed into 2 sub-regions: one for the solid zone and the other is for the fluid zone. As boiling flow is strongly affected by local mechanisms in the turbulent boundary layer near the heated wall, more intensive meshes are used near the walls, especially on the fluid side. A value of $y^+ \approx 30$ is considered reasonable for the standard wall function approach selected. Wall function approach was selected as a cost effective alternative to Enhanced Wall Modeling approach, which is extremely computationally demanding. Four mesh densities from 6 K to 41K are tested for solution independence, and the base case is solved for every mesh. The results are compared in Fig. 4.24. The most important solution variable in the current study is the vapor volume fraction, so it is selected as a criterion for comparison. Figure 8 shows that the variation in the values of the calculated vapor volume fraction is negligible when the mesh size is doubled from 20 K cells to 41 K cells. Therefore, this mesh of 41 K cells is selected for the subsequent analyses.

![Figure 4.24 Mesh sensitivity analyses](image)

The simulation is carried out using the commercial CFD software ANSYS14.0 from ANSYS, Inc. The simulation uses the segregated solver, which employs an implicit pressure-correction scheme and decouples the momentum and energy equations. The phase coupled SIMPLE algorithm is used to couple the pressure and velocity. Second order upwind scheme is selected for spatial discretization of the convective terms. The computation is conducted for each
phase simultaneously, and the interphase exchange terms (lift, drag, and heat and mass transfer) are used to couple the two phases.

The problem is treated as a transient, liquid-vapor two-phase turbulent flow with boiling occurring on the heated wall. The liquid phase properties are assumed to vary with the liquid temperature, while the saturation properties are used for the vapor phase. The saturation temperature is around 558 K. Heat conduction equation is solved in the conjugate solid region to give the temperature distribution inside the channel wall. Time periodic solution is obtained after 20 seconds, approximately 666 time steps but the solution is continued to 1000 time steps to ensure full convergence. Time periodic solution is judged by the variation of local solution variables at exit of the domain as well as by the values of residuals. Variations of vapor volume fraction and mass flow rate imbalance, with time, are taken as convergence criteria.

For numerical accuracy, the second order upwind scheme is used for spatial discretization. A time step size is calculated based on the reasonable Courant number criterion. The Courant Number can be roughly estimated for 1D case as \( u \frac{\Delta t}{\Delta x} \), where \( u \) is the air velocity, \( \Delta t \) is the time step size, and \( \Delta x \) is the length interval for the computational cell. As the numerical scheme employed is implicit, which is unconditionally stable, a high Courant number of 40 can be used to achieve faster convergence. Accordingly, a time step size of \( 0.03 \) s is calculated and used for the calculations (based on the velocity and mesh size intervals in the base case). To make sure that the solution is independent of the size of the time step, a sensitivity study is performed. Different time steps are used to reproduce the base case solution, and the results are compared with the original time step size results. Results of the time step size analysis are shown in Table 4.8. It is clear that the solution is time step size independent because increasing or decreasing the time step size an order of magnitude has a negligible effect on the solution variables (\( \approx 0.1 \% \) change).

<table>
<thead>
<tr>
<th>Table 4.8 Time step size [s] sensitivity study results</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Time Step Size (s)</strong></td>
</tr>
<tr>
<td>Exit ( V_f )</td>
</tr>
<tr>
<td>Exit liquid static temp.</td>
</tr>
</tbody>
</table>
4.3.5 Results and Discussion

4.3.5.1 Baseline case results

In the current work, a numerical simulation is performed with the baseline case conditions shown in Table 4.6. The flow field is solved for both phases until a time periodic solution is obtained. Results are shown here for the baseline case.

Figure 4.25 shows the contour of vapor volume fraction, liquid static temperature, and vapor static temperature, respectively. As the channel is too long compared to its diameter, only three selected parts of the tube representing inlet, middle, and exit sections are displayed for clarity. As shown in Fig. 4.25(a), volume fraction is zero at the inlet, as there is no vapor entering the domain, and then starts to increases in the middle of the channel until it reaches the highest value at the channel exit. Of course, the vapor volume fraction starts to increase near the wall as the boiling starts from the wall where the nucleation sites exist until bulk boiling happens. Figure 4.25(b) shows the mass flow averaged liquid static temperature increasing from the inlet value of 473 K to the saturation value of 558K at exit. The effect of wall superheat is clear at the exit section of the channel where a thin layer of liquid is slightly above the saturation temperature. As we solve the conjugate heat transfer problem, the temperature distribution inside the channel wall is also shown. Figure 4.25(c) shows that the vapor temperature is fixed at the saturation value of 558K, which is a characteristic and a basic feature of subcooled boiling.

Figure 4.26 shows the variation of temperature and volume fraction with position through the channel (axial variation). A wealth of physics related to boiling is presented in this figure which warrants a detailed description. The nucleation sites formed on the inner wall are the places where bubbles originally form (where surface- or micro-boiling happens). This happens when the inner wall temperature exceeds the saturation temperature, as marked with (ONB). Bubbles detach from the surface with continuous addition of heat and start to form the bubbly flow regime, as marked with (OSB) in Fig. 4.26. Penetration of vapor bubbles into the channel core and the accompanied heat and mass transfer exchange cause the Bulk Boiling to happen.

First of all, it is important to mention that the outer wall temperature of 650 K, not shown in figure, is the main driver for boiling as it provides the energy of vaporization. The continuous consumption of heat, used in vaporization, prevents the inner wall temperature from increasing dramatically (burning or boiling crisis). Thus, it is clear that the static temperature of the inner wall \(T_{s\text{-wall}}\), as calculated from the liquid layer attached to the wall, starts to increase from the
inlet value of 473K to 10 degrees of superheat \((T_w-T_{\text{sat}})\), which is a unique feature of subcooled boiling, as shown schematically in Fig. 4.19. The point at which the inner wall temperature exceeds the saturation temperature \((x \approx 0.05 \text{ m})\) is considered the ONB where micro-boiling occurs and where nucleation sites start to appear. It takes a while until bubbles grow and detach from the surface declaring that significant boiling has started. At \(x \approx 0.12 \text{ m}\), the vapor volume fraction at the wall \((V_{f\text{-wall}})\) starts to increase which is considered the OSB point. At \(x \approx 0.52 \text{ m}\), boiling increases and droplets travel to the core of the channel without condensation. As the degree of subcooling decreases along the channel and, consequently, so also does the rate of condensation, the steam bubbles formed will be able to penetrate further into the subcooled flow core, increasing the turbulent transport of heat and changing the pattern of temperature distribution over the cross section of the flow core. The static temperature distribution at the channel axis \((T_{\text{axis}})\) and the bulk liquid temperature \((T_b)\) starts to coincide, declaring that bulk boiling has started. Accordingly, vapor bubbles start to appear at the core of the channel causing the vapor volume fraction at the channel axis \((V_{f\text{-axis}})\) to increase dramatically due to the bulk boiling.

Also, Fig. 4.26 shows a comparison between the single phase 1D calculation of mean temperature \((T_{m\text{-SP}})\) and the CFD computed average liquid temperature at the channel axis \((T_{\text{axis}})\) and the bulk liquid temperature \((T_b)\). The single phase calculation tends to under-predict the mean liquid temperature because no near-wall boiling model is included in the 1-D calculation. Finally, at the point where bulk boiling starts \((x \approx 0.52 \text{ m})\), the axis and the bulk liquid temperatures coincide, as mentioned earlier. From this point on, the vapor volume fraction increases dramatically and the fluid velocity greatly increases due to the rapid expansion of overall volume flow rate. The liquid portion is dragged forward by faster moving vapor bubbles as shown in Fig. 4.27(a). Figure 4.27(b) shows the distribution of heat flux supplied to the water during the boiling process in terms of three components—namely, Liquid phase, Quenching, and Evaporation heat fluxes.
Figure 4.25 Contours of (a) vapor volume fraction (b) average liquid static temperature (c) vapor static temperature at different sections of the channel (Baseline Case)

Figure 4.26 Variation of temperature and volume fraction (right axis) with position along the channel
4.3.5.2 Parametric Study

The result of the baseline case indicates noticeable nucleate boiling starts as early as 10 cm into the channel and bulk boiling occurs at about 2/3\textsuperscript{rd} of channel length. The main goal of this study is to examine the variables that may affect the baseline case result and seek means to minimize or avoid boiling. This section presents the study of effect of four parameters on boiling phenomena in the studied channel.

4.3.5.2.1 Effect of changing wall temperature

Wall temperature is a vital important parameter in any phase change analysis. In the baseline case, the wall temperature is assumed constant at 650 K. Figures 4.28-4.29 show the effect of varying wall temperature between 665 K and 600 K. Figure 4.28 shows that if the wall temperature is lower than 625 K, it will be safe enough to avoid nucleate boiling in the feed channel and hence ensure the proper operation of the atomizer. On the other hand, if the wall temperature reaches 665 K, the vapor volume fraction will reach 0.817 at the exit. This 665 K wall temperature provides an upper limit beyond which the boiling will deviate from the equilibrium nucleate boiling phenomenon to the unstable Departure of Nucleate Boiling (DNB) regime. Also, it is noted that the OSB happens at earlier locations at higher wall temperatures.
Figure 4.28 Variation of the vapor volume fraction along the conjugate wall for different channel wall temperatures

Figure 4.29(a) shows the variation of vapor volume fraction and vapor velocity (m/s) at the channel exit plane for different wall temperatures. Mass weighted averages are used in both cases. It is clear that increasing the wall temperature produces a significant linear change in the vapor volume fraction by producing more vapor bubbles. These vapor bubbles move through the channel core with velocities faster than the liquid phase velocity. Vapor velocity at exit is almost linearly increasing with the wall temperature as well. Due to the momentum exchange between the liquid and the vapor phases, the liquid phase velocity is accordingly increasing with wall temperature, as shown in Fig. 4.29(b). In the same figure, the exit liquid phase temperature (K), at the secondary axis, also increases until it reaches the saturation temperature.

Figure 4.29 Variation at the channel exit plane for different wall temperatures. (a) Vf and vapor velocity V (secondary axis) (b) Liquid velocity V and liquid static temperature $T_{s,l}$ (secondary axis)
4.3.5.2.2 Effect of changing the channel length

The effect of changing the channel length is studied to account for the location of the atomizer for different engines designs. The results in Fig. 4.30 show that the longer the channel length, the higher the vapor volume fraction is. This is expected as longer channel provides a longer residence time for the liquid phase to change into vapor. One interesting note is that the OSB is independent of the channel length as it occurs at $x \approx 0.12$ m for the three cases. It is interesting to notice that the vapor volume fraction at a specific axial location is proportional to the channel length (from $x=0.25$ to $x=0.5$). This proportionality reflects an aspect of the elliptic behavior of the solved system of governing equations. This elliptic behavior is responsible for transmitting the disturbances further upstream of the channel, against the flow direction, in this subsonic flow case. Similar results are obtained for exit velocity and temperature. These results are important as it gives a way to control the exit conditions of the channel to match the desired atomizer design.

![Figure 4.30 Variation of vapor volume fraction along the conjugate wall for three different channel lengths.](image)

4.3.5.2.3 Effect of inlet subcooling

Subcooling temperature indicates the temperature below the saturated temperature; therefore, the higher the subcooling temperature, the lower the preheating being employed. In real gas turbines, it is a matter of caution by feeding cold water directly to a highly heated engine component, like the proposed feed channel in Fig. 4.1 due to the concern on potential "thermal shock" that could cause large thermal stress near the feeding entrance. Thus, in the baseline case the water is preheated to 473 K (or 85 K subcooling form $T_{\text{sat}} = 558$K) to minimize the potential thermal shock by assuming that the water is introduced after the gas turbine reaches full load to help reduce compressed air extraction for cooling. However, a smart-control algorithm for controlling water feeding rate can be implemented during gas turbine start-up process from cold.
In this way, the feed water can be fed at room temperature during the ramping process and the material temperature surrounding the feed channel will gradually adjusted, not reaching as high as the current operating condition without the feed channel. Without going through the complexity of conducting a transient study from start-up engine from cold, the analysis is performed by simply reducing the water inlet temperature but keeping the wall temperature at 650K. Figure 4.31(a) shows that inlet subcooling of 15 K (or high preheating) greatly accelerates the boiling process, which occurs directly at the channel inlet. As the degree of subcooling decreases along the channel and, consequently, so also does the rate of condensation, the steam bubbles formed will be able to penetrate further into the subcooled flow core, increasing the turbulent transport of heat and changing the pattern of temperature distribution over the cross section of the flow core. Also, Fig. 4.31(a) shows that the vapor exit volume fraction is inversely proportional to the degree of subcooling. This result is also important as it gives a way to control the exit conditions of the channel to match the desired atomizer design. Finally, Fig. 4.31(a) shows that the inlet subcooling of 208 K (Inlet temperature = 350 K) will completely suppress the boiling process. This helps to indicate a threshold for the boiling process to start under the current operating conditions and serves to introduce a boiling-free flow. However, the boiling results for highly subcooled cases ($\Delta T_{\text{sub}} > 100$) should be taken with care because the RPI model is not validated in these cases (Kurul and Podowski, 1991; ANSYS, 2010). Comparing with the result of 1-D model, again, it shows that 1-D model over-predicts the water temperature distribution. The 1-D model predicts that the temperature inside the channel will rise above the saturation and boiling would occur. This is due to the lack of near-wall boiling model in the 1-D model as discussed earlier when Fig. 4.26 was examined.

4.3.5.2.4 Effect of changing operating pressure

Operating pressure is one of the important parameters that will directly affect not only the boiling process, but also the atomizer performance. Normally, a higher pressure is desirable for both processes as it suppresses boiling and enhances the atomization process.

Figure 4.31(b) shows, as expected, that increasing the pressure to 110 bar is sufficient to suppress the boiling process with a subcool temperature at 85 K ($T_{\text{in}} = 473$ K) as it gives an average vapor volume fraction of 0.0025 at exit. The case of 60 bar is very close to the unstable DNB regime.
4.3.5.2.5 Effect of gravity

Since the gravity can be in any direction in the actual gas turbine geometry, it is important to investigate its effect on the results. As the channel is vertical, the problem is solved with flow in the upward and the downward directions. The effect of gravity direction on the vapor volume fraction generated is shown in Figure 4.32. The vapor volume fraction at exit is slightly higher at exit in case of the downward flow.

Figure 4.32 Effect of gravity on the vapor volume fraction at the contact wall

4.3.6 Practical Issues

It is commonly known that boiling can be suppressed by reducing the inlet water temperature, increasing the pressure, or reducing the wall temperature. The above analyses provide the actual values for the pressure, wall temperature, and threshold inlet water temperature for suppressing the boiling.
The analyses indicate that keeping the wall temperature less than 600 K is an effective means in suppressing boiling, yet it is also the only parameter that cannot be modulated because it is subjected to the engine operating conditions. Thus, in order to keep the wall temperatures less than 600 K, it is important to position the channel in a suitable place. Before the positioning of the channel is determined, it will be interesting to study the effect of the change in the number of feed channels on boiling. The baseline case is repeated with two additional numbers of channels and the results are shown in Fig. 4.33. Figure 4.33 shows that decreasing the number of channels from 70 to 16, will suppress boiling completely (with \( T_w = 650 \text{ K} \) in the base case). The reason of choosing 16 channels is that 16 is a factor of 48 (the number of vanes). Thus, every channel serves 3 vanes.

![Figure 4.33 Effect of reducing number of channels on the vapor volume fraction at the contact wall](image)

The following list of some practical issues and proposed solutions follows from the above results:

1. To modulate the wall temperature to lower than 600 K, the feed channel may pass through the vane cavity all the way to the atomizer. The reason for this is that the vane cavity carries the compressor cooling air at 600 K. Knowing that Frame 7FA has 48 vanes in the first stage of the HP turbine, the number of feed channels reduces to 48 instead of 70. This situation will increase the mass flow rate and velocity in the channel to 0.0278 kg/s, 1.077 m/s, respectively. As shown in Fig. 4.33, this will produce a relatively desired (i.e., little to no boiling) working environment. Further safety can be achieved if only 16 channels are used to carry the compressed water, as this completely prevents boiling. Furthermore, if the system
will be used in conjunction with a mist cooled vane, the vane cavity temperature will be around 400 K, as investigated in (Ragab and Wang, 2012). These conditions are more certain to prevent any boiling.

2. Although operating pressures above 110 bars are recommended to suppress boiling, they may cause flash evaporation of atomized droplets. Flash evaporation may happen once the droplets are injected in the low pressure preswirl chamber (≈ 18 bar, 600 K). Further investigation is required to estimate its possibility and impact.

4.3.7 Conclusions

The present work proposes a method to implement a liquid water transport mechanism to provide mist cooling technique to the rotating engine components by inserting a feed channel in the engine body. The expected boiling is predicted by using the Eulerian multi-fluid model in the CFD solver ANSYS/FLUENT in conjunction with the RPI wall boiling model. The conclusions are summarized as follows:

• It is feasible to have highly pressurized water transported to the internal engine cooling passages and be further atomized as mist for turbine component cooling.

• Reducing wall temperature below 600K surrounding the feed channel is an effective parameter on suppressing boiling.

• Higher operating pressures (≈ 110 bar), higher inlet subcooling (ΔTsub ≈ 210K or Tin = 348 K), and shorter channel lengths (≈ 0.5 m) leads to boiling free flows. A combination of these parameter values can be used with further optimization.

• The 1-D model gives a simple and fast method to calculate the water temperature distribution through the feed channel. However, the 1-D model tends to over-predict the temperature distribution in the channel and, thus, under-predict the inlet temperature that boiling could occur due to lack of near-wall boiling model in the 1-D model.

Further investigation is required to study the potential of thermal stresses and the droplet distribution in the rotating components.
4.4 INVESTIGATION OF APPLICABILITY AND IMPACT OF USING WATER MIST FOR COOLING HIGH PRESSURE TURBINE COMPONENTS: ROTOR BLADES

4.4.1 Background

The previous work reviewed before in Sec 4.1 shows that mist cooling is a promising and efficient technique based on the assumption that the mist can be successfully transported and delivered to the inlet of the cooling sites. However, the following questions and challenges have yet to be answered. Where is the mist generated? Can the mist survive the extremely hot conditions inside the flow passages in the gas turbine and be successfully delivered to the needed sites? Partial answer was given in previous sections 4.1 and 4.2 and more will be detailed here.

The objective of the present study is to use a CFD scheme to investigate the feasibility of transporting the mist through the internal cooling air passage to the film cooling hole sites on the rotor blades. This work is the third part of a series of studies to investigate how to transport mist to the needed sites, including vanes, blades, seals, pre-swirlers, and rotating disk cavities.

4.4.2 Studied Configuration

A schematic of a typical film cooled high-pressure turbine is shown in Fig. 4.1. The relatively cold compressor air (600 K) is bled from the compressor discharge through the cooling air passage where it enters the cooling channels inside the turbine vanes to cool the vanes' walls, which are toasted by extremely hot flu gases at about 1300 K (2350°F). Eventually, the air exits the film cooling holes to form a protective layer around the external vane surface to minimize direct contact with the hot gases.

From a previous stator cavity study in Sec. 4.1, the atomizer was proposed to be located in the passage of the compressor bleed air near the outer casing a short distance just upstream of the vane's leading edge. The water mist is to be injected at the inlet of the cooling air passage along the casing. The tubing of the high-pressure water is directly inserted through the turbine casing, see Figs. 4.1 and 4.2a. For the rotor, the situation is more challenging because of the rotation. The exact path of the compressed water tubing depends on the engine design and how the cooling air is fed to the rotor film holes (Dennis, 2006; Snowsil and Young, 2006). Basically, there are two possible designs. In the first design, the cooling air is supplied to the first rotor blades through a pre-swirl nozzle, see Figs. 4.2a and 4.34, where it is accelerated tangentially in
the direction of the rotor rotation. This acceleration reduces the relative total temperature, which is the total temperature in the rotating frame of the cooling air, and, hence, increases the film cooling efficiency. The performance of this pre-swirl nozzle is crucial for the engine components life expectancy. It is estimated that a 20°C increase in relative total temperature of the air fed to the rotor is sufficient to reduce the average lifetime of the components by as much as 50% (Snowsill G.D. and Young C., 2006).

In this first design, the atomizer is proposed to be located somewhere in the pre-swirl nozzle chamber. To deliver the compressed water to that proposed location, a feed pipe is inserted through the turbine casing and passes through the engine stationery part (first vane) all the way to the atomizer, as shown schematically in Fig. 4.1. The compressed water is introduced to the atomizer through the feed pipe, and the atomized water droplets will mix with the compressor bled cooling air to form a mixture of air/mist. This mixture will serve to cool the blade disc and the seals, and will travel through the rotor cover-plate disk cavity to, hopefully, reach the rotor blades. In the second design of the Onboard Air Supply passage, the cooling air, bled from the compressor exit, is introduced to the first stage rotor blades via the second stage vane passing through a nozzle diaphragm box inboard of the airfoil inner endwall. This Outboard Air Supply route is shown schematically in Fig. 4.1 marked with the blue arrow (from 2nd vane to 1st bucket) and circled to show the domain of interest. For simplicity, only one blade sector for the rotor cavity and only 2D nozzle diaphragm passage geometry is studied. A more complicated arrangement or optimization strategy could be arranged in the future to reduce engineering cost and improve mist-distribution effectiveness.

![Figure 4.34 Schematic of Pre-Swirl nozzles, Seals, and Cover-plate disk cavity (Snowsill and Young, 2006)](image-url)
4.4.3 Zero-Dimensional Model

A quick and reasonably accurate estimate of the droplet's *residence time* can be obtained with a simple zero-dimensional model. The residence time is the time during which the droplet starts to evaporate from the droplet surface, then boil from within the droplet, and finally become all vapor. Correlation 4.1 is used to calculate the residence time ($\tau$) of the droplet. In this study, the water temperature $T_f = 300$ K, air temperature $T_1 = 600$ K, latent heat $h_{fg} = 2256$ kJ/kg, thermal conductivity of air $\lambda_1 = 0.048$ W/m-K, and droplet Reynolds number $Re_d = 55$, based on droplet diameter and assumed slip velocity of 10 m/s. Based on this data, the droplet's residence time for diameters from 10 and 20 µm is calculated according to Eqn. (4.1) and shown in Table 4.9. The distance travelled during this period of residence time is estimated based on the cooling air velocity at 50 m/s. The results tabulated in Table 4.9 are very informative.

<table>
<thead>
<tr>
<th></th>
<th>D = 10 µm</th>
<th>D = 20 µm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time (s)</td>
<td>0.0072</td>
<td>0.0233</td>
</tr>
<tr>
<td>Distance (m)</td>
<td>0.358</td>
<td>1.169</td>
</tr>
</tbody>
</table>

Table 4.9 shows that if the initial water droplets are 20 µm in diameter, their residence time is about 23 ms; and during this flashing instant, they can fly a distance greater than 1 m before they completely evaporate. In a typical 7-frame GT, the distance over 1 m can cover the distance from the mist injection point near the casing to the film cooling hole base as well as most of the vane's surface. However, the situation may differ for the rotor depending on the passage used to deliver the mist. It is understood that, once the mist exits the film cooling holes, the droplets will face the hot gas temperatures as high as 1300K. Therefore, the portion of residence time outside the vane or the rotor will be shortened by about 70%. However, this quick estimate is meant to determine if the droplets can survive and reach the film cooling holes. Once they reach the cooling holes, the external film cooling behavior has been intensively studied by (Guo et al., 2000 I, III; Wang et al., 2005; Li and Wang, 2006 and 2007; Wang and Li, 2008; Li

It is important to emphasize that it has been commonly thought that the mist would disappear "instantaneously" when it is injected into the hot gas environment. Here, the above calculation informs us that this "instantaneous time" is not zero second, but it is about 23ms for a droplet initially in 20 µm and is exposed to 600 K gas. Even if the air temperature is raised to 1300K, the calculation shows that a 20 µm droplet will take 4 ms to completely evaporate, which is still not zero second. Of course, this zero-dimensional analysis and associated correlation have some uncertainties, but the result provides a quick and positive feedback to our questions. This positive result motivates a continuous study to employ a more sophisticated multi-phase CFD scheme.

4.4.4 CFD Calculations

The zero-dimensional model results showed that the application of water mist/air cooling technique is feasible. To obtain more reliable results, comprehensive CFD calculations are performed for flow under real gas turbine operating conditions.

4.4.4.1 Mathematical Model for the Carrier Phase

A feasible method to simulate cooling with air/mist injection is to consider the droplets as a discrete phase since the volume fraction of the liquid is small (less than 0.1%). The trajectories of the dispersed phase (droplets) are calculated by the Lagrangian method (Discrete Phase Model, DPM). The impacts of the droplets on the continuous phase are considered as source terms to the governing equations of mass, momentum, energy, and species. Equations 4.2-4.5 of mass, momentum, energy and species, represent the basic governing equations in this part of the work. As in Sec 4.2, the standard k-ε model is used with standard wall functions to model the near-wall turbulence structure as presented in Equations 4.6-4.10.

To track the trajectory of droplets, the hydrodynamic drag, gravity, and forces, such as the “virtual mass” force, thermophoretic force, Brownian force, and Saffman's lift force, are combined to affect the droplet motion. The DPM model equations 4.11-4.14 are used to represent the droplet particles.
4.4.4.2 Computational domain

The computational domain is a sector of the cover-plate disk cavity that feeds the blade with the cooling air coming from the pre-swirl nozzle. As the Frame 7 engines contain 70 buckets in the first turbine stage, the computational domain is a sector of $360/70 = 5.143^\circ$ in the circumferential direction. The part simulated from the cavity is 16 cm in the axial direction with an average height of 65 cm. The receiver hole is 3 cm diameter. The computational domain and the mesh are shown in Fig. 4.35. The water mist is injected at the receiver hole, and moves with the air until the droplets reach the exit plane that leads to the blade base (blade broach).

![Figure 4.35 The computational domain](image)

4.4.4.3 Boundary conditions

4.4.4.3.1 Airflow

For the rotor cover-plate disk cavity, rotating at 3600 RPM, the main flow is assumed to be dry air (zero humidity). The rotating reference frame model is used for the simulation which only provides a moving speed to the simulated geometry, but the mesh is not actually rotating. As a single blade cavity is used as the simulated geometry with the mass flow rate is calculated as 3 % of the engine total mass flow rate. The total mass flow is divided among the 70 rotor blades to give an inlet mass flow rate of 0.1907 kg/s, which is imposed on the inlet boundary. The inlet static temperature is 600 K, reasonably assumed equal to the compressor discharge temperature. For the cooling cavity walls, the temperature is taken as 650 K. All walls are assumed to be no-slip walls. At the exit of the domain, a constant static pressure of 18 atm is imposed along the boundary. Inlet turbulence conditions for $k$ and $\varepsilon$ are defined according to the
correlations 4.15-4.17. With the turbulence intensity $I = 5\%$ is assigned to simulate a fully-developed duct flow; the turbulence length scale, $\ell$, is based on the inlet hydraulic diameter of the air cooling duct $D_h = 9.12$ cm; $C_u = 0.09$; $u_{avg}$ is the average inlet air velocity, which is calculated through the given inlet mass flow rate.

### 4.4.4.3.2 Droplet injection

For the rotor cavity, the uniform droplet size of 50 $\mu$m is considered with injection velocity of 62.8 m/s (equal to the mass weighted average velocity of the air flow at inlet); and the mist ratio is 10% (about 0.01907 kg/s). The number of mist injection points at the coolant inlet depends on the number of computational elements (cells) at the inlet surface. In the present case, about 40,000 injections are released from the inlet face based on 10 tries for stochastic tracking. The boundary conditions and the base case parameters are summarized in Table 4.10. Based on a previous study in Sec. 4.1, the wall boundary condition effect on droplet trajectories was found insignificant, under the real engine operating conditions. Therefore, the simplest boundary condition of “reflect,” is used. This means that the droplets elastically rebound once reaching the wall. To further understand the effect of some important operating parameters, a parametric study is performed as shown in Table 4.11. The parameters chosen for study are droplet initial diameter ($D_i$), and the mist ratio. These parameters were found important in a previous study as detailed in Sec 4.1.

#### Table 4.10 Base case conditions for the rotor cavity

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Air</strong></td>
<td></td>
</tr>
<tr>
<td>Air inlet mass Flow</td>
<td>0.1907 kg/s</td>
</tr>
<tr>
<td>Air inlet temperature</td>
<td>600 K</td>
</tr>
<tr>
<td>Exit static pressure</td>
<td>18 atm</td>
</tr>
<tr>
<td>Cavity wall temperature</td>
<td>650 K</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>Standard k-$\varepsilon$</td>
</tr>
<tr>
<td><strong>Droplets</strong></td>
<td></td>
</tr>
<tr>
<td>Droplet initial diameter</td>
<td>50 $\mu$m</td>
</tr>
<tr>
<td>Droplet initial velocity</td>
<td>62.8 m/s</td>
</tr>
<tr>
<td>Droplet initial temperature</td>
<td>450 K</td>
</tr>
<tr>
<td>Mist Ratio</td>
<td>10 %</td>
</tr>
<tr>
<td>Wall boundary cond.</td>
<td>Reflect</td>
</tr>
</tbody>
</table>
Table 4.11 The matrix of the parametric study for the rotor cavity problem

<table>
<thead>
<tr>
<th>Case</th>
<th>$D_i$ (μm)</th>
<th>Mist ratio (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>30</td>
<td></td>
</tr>
<tr>
<td>Base Case</td>
<td>50</td>
<td>10</td>
</tr>
<tr>
<td>2</td>
<td>70</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>90</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>5</td>
</tr>
<tr>
<td>Base Case</td>
<td>50</td>
<td>10</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td>15</td>
</tr>
<tr>
<td>6</td>
<td></td>
<td>20</td>
</tr>
</tbody>
</table>

4.4.4.4 Meshing and simulation procedure

For the rotor cavity, the computational domain is constructed by unstructured tetrahedral elements as shown in Fig. 4.35. This type of unstructured mesh is suitable for complicated geometries like that of the present cover-plate disk cavity. More intensive meshes are used near the walls, in order to give a value of $y^+ \approx 30$ which is reasonable for the turbulence model selected. Four mesh densities are tested for grid independence, and the base case is solved for every mesh. The results are compared in Fig. 4.36. Solution variables like relative total temperature ($Tr_t$), total temperature ($T_t$), static Temperature ($T_s$), relative Mach number ($Mr$), and Mach number ($M$) showed the same trend. The term “relative” implies being relative to the rotating frame. It is clear that the difference between 0.9 and 1.2 million is insignificant, so the 0.9 million mesh size is selected for the calculations to save CPU time.

![Figure 4.36 Mesh sensitivity analyses (for the rotor study)](image-url)
The simulation is carried out using the commercial CFD software ANSYS14.0 from ANSYS, Inc. The simulation uses the pressure based solver, which employs an implicit pressure-correction scheme and decouples the momentum and energy equations. The SIMPLE algorithm is used to couple the pressure and velocity. Second order upwind scheme is selected for spatial discretization of the convective terms and species. The computation is conducted for the main and coolant flow field (continuous phase) first. After obtaining an approximate converged flow field of the air, the dispersed phase of droplet trajectories are calculated. At the same time, drag, heat, and mass transfer between the droplets and the air are calculated. Iterations proceed alternatively between the continuous and discrete phases. Converged results are obtained after the residuals satisfy mass residual of $10^{-3}$, energy residual of $10^{-6}$, and momentum and turbulence kinetic energy residuals of $10^{-4}$. These residuals are the summation of the imbalance for each cell, scaled by a representative of the flow rate.

### 4.4.5 Results and Discussion

#### 4.4.5.1 Model validation

The mist/steam cooling CFD scheme and model have been validated by the same research group members Dhanasekaran and Wang (Dhanasekaran and Wang, 2008, 2010, and 2011; Wang and Dhanasekaran, 2010) with the experimental data in conditions like flow in heated tubes (Guo et al., 2000 I, II), 180-degrees tube bend (Guo et al., 2000), and impinging jets (Wang et al., 2005). The mist/air cooling CFD models have been qualified by Li and Wang (2006, 2007, and 2008). The same CFD scheme and model are used in this study; hence no validation is repeated here.

#### 4.4.5.2 Base case results (rotor cavity problem)

In the current work, a numerical simulation is performed with the base case conditions shown in Table 4.10. The flow field in the dry base case is solved first without droplets until a converged solution is obtained then the droplets are injected according to specifications in Table 4.10 for the mist base case. The base case for the rotor cavity uses 10% mist ratio and 50 µm droplet diameters.

Figures 4.37 (a) and (b) show the droplet traces colored by the droplet diameter (m) and the droplet residence time (s), respectively. It can be noticed that trajectories close to the wall vanish early in the domain due to the wall superheat. Accordingly, a thin layer of water vapor
with high $C_p$ value is therefore formed and shields the core flow from the hot wall, thus leaving the core flow cooler to sustain the water droplets longer. *This key result supports the zero-dimensional results that droplets can survive provided it has the sufficient mass and diameter.*

Again, the main goal of the current study is to prove that droplets with reasonable diameters and loading can survive in this very hot environment until they reach the blades. Please note that not all of the droplet streams are shown in Figs. 4.37 (a) and (b). The CFD solution helps show the flow behavior in the cooling cavity up to blade entry. Figure 4.37 (a) also shows that 50 µm water droplets, although big enough, completely evaporates and could not reach the blade base (under the base case conditions). Further investigations were performed to increase the diameter of the injected droplets and the mist ratio to have droplets reach the domain exit, as will be discussed later.

**Figure 4.37** Droplet traces in the rotor cover-plate disk cavity colored by (a) droplet diameter [m] (b) residence time (s) (Base Case: 10% Mist Ratio, 50 µm initial diameter, 650 K wall temperatures)

Figure 4.38 shows the temperature contours at the mid-plane of the cavity in the base case. Figures 4.38(a) and (b) show the static temperature contours in the dry and wet cases, respectively. As the cavity is rotating, the rotor only feels the relative total temperature entering to it. Contours of the relative total temperature are displayed for the wet and dry base cases in Figures 4.38(c) and 4.38(d), respectively. A local drop of 100 K in the total relative temperature is calculated. *This result is crucial for the life time expectancy of the rotor blades, as been discussed in the introduction and reported by Snowsill and Young (2006).* Again, they estimated a 50% reduction in component lives for only a 20 K increase in relative total temperature of the
cooling air. So, the 100 K drop in the cavity air relative temperature is a precious benefit. Figure 4.39 shows the relative velocity field inside the cavity (at the mid-plan).

Due to the droplets evaporation and boiling, latent heat of vaporization is absorbed from the main flow resulting in this remarkable reduction in the air temperature. This is a very important result as it implies a high cooling potential for the mist cooled cavity air. As the air temperature is reduced, the amount of air required for cooling the rotor blades, which is bled from the compressor, can be notably reduced and the net output power can be augmented. Also, saving the precious compressor air helps to increase the thermal efficiency of the gas turbine engine. Another factor that leads to savings in the compressor air is that the specific heat of air/water vapor mixture is higher than that of air alone. This means a higher cooling capacity of the mist-air and hence a lower compressed air consumption. The final contribution from the mist comes from its higher film cooling effectiveness which can help extend the life of the turbine components due to the reduction of thermal stresses.

Figure 4.38 Contours of temperature [K] at the mid-plane in the base case (a) Static temperature dry (b) Static temperature wet (c) Relative total temperature dry (d) Relative total temperature wet
4.4.5.3 Parametric study results

The required outcome of this work is to prove the existence of droplet streams with reasonable diameters (in the order of 15-20μm) at the blade section. To understand well the effect of changing operating parameters on the droplet distribution at the exit, a parametric study is performed. Mist ratio (ratio of the mass flow rate of water mist to the air mass flow) and initial mean diameter of injected droplets are the most effective parameters, as been investigated previously in Sec. 4.1. Although the real droplet diameters are polydispersed, all the cases are performed assuming a uniform (monodispersed) droplet diameter distribution. This simplification is necessary for exercising a controlled study to track change of droplet's size. Effect of injected diameter distribution is studied in Sec. 4.1.

Particle loading is an important parameter in any two-phase flow application. Mist ratio, defined as the ratio of mass flow rate of water mist at injection location to that of air, is used here to express the effect of particle loading. As shown in Fig. 4.40(a), increasing the mist ratio from the base case value of 10 % to 15 % reduces the exit relative total temperature by 40K, which means higher cooling potential for the rotor blades. Being an open loop cooling cycle, mist cooling technique has some restrictions on the water consumption rate, which is linearly proportional to the mist ratio. For the Frame 7 engine with 70 rotors in the first turbine stage, 3% of the total air mass flow of 445 Kg/s is used for cooling this row of blades. Under this condition,
a 10% mist ratio translates to 0.3% of the total air mass flow rate, or 1.335 kg/s (for the 70 rotors). The water consumption (gal/min) can be calculated for different mist ratios, for these 70 rotors, as shown in Fig 4.40(a). Figure 4.40(b) shows the relative total temperature variation at exit, for different initial droplet diameters. Compared to Fig. 4.40(a), smaller particle sizes are preferred as it completely evaporates within the domain, and, hence, give higher cooling potential. On the other hand, the larger droplets have a longer life which, from the perspective of the current work, is more desirable to achieve mist cooling through the blade and on its outer surface.

![Figure 4.40 Exit relative total temperatures and water consumption [gal/min] for different (a) mist ratios (b) initial diameters](image)

Figure 4.41 shows the droplet diameter distribution at exit holes in 15 % mist ratio case. The mean droplet diameter can be evaluated in various ways. The most commonly used mean diameters are the Arithmetic Mean Diameter, $D_{10}$, and the Sauter Mean Diameter, $D_{32}$. Both diameters can be defined as follows:

$$D_{10} = \frac{\sum_{i=1}^{n} d_i}{n}, \quad D_{32} = \frac{\sum_{i=1}^{n} d_i^3}{\sum_{i=1}^{n} d_i^2}$$  \hspace{1cm} (19)$$

Where $n$ is the total number of droplets. The Sauter mean diameter is of interest in applications where the active surface area is important, as in spray evaporation applications. Droplets with arithmetic mean diameter of 6.9 µm and mean Sauter diameter of 8.2 µm are obtained for the 15% mist case.
The droplets with diameter $6.9 \ \mu m$ at the cavity exit are too small for providing mist cooling in the blade because they need to flow through the internal cooling channels before they can reach the film cooling holes distributed on the blade surfaces. Therefore, higher mist ratios and larger initial droplets will be tested. Figures 4.42 (a) and (b) show the effect of changing the mist ratio and the initial injection diameter on the exit droplet diameter, $D_{\text{exit}}$, respectively. The exit droplet diameter is proportional to both the mist ratio and the initial diameter. Furthermore, larger initial diameter significantly increases the chance of having functional droplets at the cavity exit. "Functional" means that droplets can be large enough to be continuously transported inside the blades to the film cooling holes. Finally, mist ratios of greater than 15 %, or droplet diameters greater than 70 $\mu m$ are required to have a considerable mist cooling enhancement in the rotor region.

In conclusion, the results can help in estimating the required droplet diameter at the preswirler inlet to achieve a certain cooling effect associated with a certain droplet diameter at the exit of the rotating cavity. These results suggest that droplets with the initial diameters ranging from 50-90 $\mu m$ and a mist mass ratio greater than 15% are suitable to achieve the droplet sizes in the range of 20-40 $\mu m$ at the root (hub) of the turbine blades. According to the previous work in Sec. 4.1, this diameter range, along with the mist ratio, can produce meaningful enhancement via mist/film cooling for Frame 7FA gas turbine rotors. Also, they will be sufficient to achieve reasonable mist cooling at the film cooling injection holes. However, this range of mist ratios and droplet diameters represents a challenge for the mist cooling in rotor blades through the

Figure 4.41 Diameter [m] distribution histogram at the cavity exit (15% Mist Ratio, 50 $\mu m$ initial diameter)
cover plate cavity passage because larger droplets have higher Stokes numbers, which means that some of the droplets may deviate from the main flow and hit the hot engine walls. When large droplets hit the engine's wall, they may evaporate or agglomerate into liquid layer and stop flying. This may also give rise to erosion and localized thermal stresses. Although the above study shows that transporting mist to the rotating blade is possible through the cover-plate cavity passage, the journey seems long. A shorter passage to transport mist to the rotor blades could be more attractive and is thus investigated as an alternate approach next.

Figure 4.42 Droplets exit mean diameter [µm] results for different (a) initial diameters (b) mist ratios

4.4.6 The Alternate Cooling Passage through the Nozzle Diaphragm

The alternative cooling path, proposed earlier as shown in Fig. 4.1, is used to deliver the cooling air from the second stage vane to the rotor via the nozzle diaphragm. The details of the diaphragm are shown in Fig. 4.43 below, with arrows indicating the air flow direction from the nozzle (vane) to the bucket (blade or rotor). In between, the air passes through a secondary air passage in the rotating shaft.
Cooling air extracted from the compressor, passes through the pipe 68 which penetrates the nozzle (though the turbine casing) to reach the diaphragm 64. Within the insert 66, the air passage changes direction via elbow passage 70 and substantially straight passage 72 to direct the air tangentially (at an angle 22-23°) into an annular rotor cavity 74. From that cavity, the cooling air moves axially through multiple sets of three passages 76, and then radially outwardly at the interface of the third stage wheel 16, to an axial passage 82 between the wheel rim and the bucket shank. From here, the air travels radially outwardly in one or more passages 86, and then vents at the hot gas path at the bucket tip and the film holes, if any.

4.4.6.1 Computational domain and mesh

The computational domain, shown in Fig. 4.44, is a simplified representation of the alternative flow path through the nozzle diaphragm. As the first row of buckets is the main area of interest, the path passes from the second row nozzles to the first row of buckets carrying the cooling air extracted from the compressor. For typical dimensions of Frame 7 engines, the pipes pass though the second row of vanes are approximated to have 4 cm diameter and 20 cm long. The rotor section pipe is 4 cm diameter with only one row of film cooling holes (28 holes, 0.15
cm diameter each) simulated. To simplify this complicated configuration and give a reasonable cost effective solution, the relative motion between the stator section and the rotor assembly is neglected. Based on order of magnitude analysis performed by Wang et al. (2007), rotational forces can be neglected in comparison with the drag and gravitational forces in the droplet force balance equation. Accordingly, the three domains (the stator, the shaft, and the rotor) are stationery. Equations (4.2-4.18) are used in the same manner as in the rotor cavity problem.

The computational domain is meshed with unstructured quadrilateral elements, except for a thin layer near the walls. This thin layer is meshed with structured elements to well resolve the strong gradients in the boundary layer. Three meshes of 28k cells, 54k cells, and 99k cells were tested for solution independency. The change in the calculated static temperature at the exit of the domain was less than 0.5% when the mesh size was doubled from 54 k to 99 k (99,248 cells) cells. This indicates that the solution is almost mesh-independent. As the mesh size is crucial in droplet evaporation problems, and, as a precaution, the mesh of 99,248 cells was used in all of the calculations.

Figure 4.44 Computational domain and mesh for the nozzle diaphragm problem.
4.4.6.2 Boundary conditions

Compressed air, bled from the compressor, enters the stator pipe inlet at 600 K and 18 atm. Wall temperatures are kept constant as 700 K for the stator pipe, 600 K for the rotating shaft passage, and 800K for the rotor pipe section. Turbulence parameters at inlet are fixed with the hydraulic diameter of 4 cm and the 5% turbulence intensity. For droplet injection, the uniform droplet size of 50μm is considered with an injection velocity of 0.8 m/s in the negative y-direction and a mist ratio of 10% (about 0.01907 kg/s per rotor). A parametric study is performed as in Table 4.12.

Table 4.12 The matrix of the parametric study for the nozzle diaphragm problem

<table>
<thead>
<tr>
<th>Case</th>
<th>D₁ (μm)</th>
<th>Mist ratio (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base Case</td>
<td>50</td>
<td>10</td>
</tr>
<tr>
<td>2</td>
<td>70</td>
<td>10</td>
</tr>
<tr>
<td>3</td>
<td>90</td>
<td></td>
</tr>
<tr>
<td>Base Case</td>
<td>50</td>
<td>10</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>15</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td>20</td>
</tr>
</tbody>
</table>

4.4.6.3 Results and discussions

Although different geometries and passages are encountered, the result is still similar to that of the vane study in Sec. 4.1 or Ragab and Wang (2012), and of the rotor cavity problem discussed earlier. Similar results are expected due to the similarity in physical phenomena and boundary conditions.

Figure 4.45 shows the static temperature contours for the dry (without mist cooling) and the mist base cases. Around 150 K reduction in temperature of the cooling air is produced. This remarkable reduction in cooling air temperature produces a high cooling potential, and will significantly increase the film cooling effectiveness. This will, in turn, increase the life expectancy of the rotor blades by reducing the thermal stress. Unlike going through the rotor cavity passage, this cooling potential directly affects the blade external surface as air exits the film holes.
Figure 4.45 Static temperature contours in the base case (10% mist ratio, 50 µm initial diameter)

Figure 4.46 (a & b) shows the droplets traces colored by the droplets’ diameters and the residence time, respectively. It is clear that using 10% (wt.) mist with the 50 µm diameter droplets in the base case are unable to reach the film cooling holes. Also, from Fig. 4.46(b), this size range takes around 0.52 seconds to completely evaporate in the rotor domain. This result is in good agreement with the zero-dimensional model presented in Eq. 4.1, for the same velocity range (0.8 m/s). Figure 4.47 shows the velocity field, with emphasis on the boundary layer flow near the walls.

Figure 4.46 Droplet traces in the passage through the nozzle diaphragm colored by (a) droplet diameter [m] (b) residence time [s] (Base Case: 10% Mist Ratio, 50 µm initial diameter)
Figure 4.47 Velocity field in the base case (10% mist ratio, 50 µm initial diameter)

Figure 4.48 presents the results of the parametric study performed as summarized in Table 4.12. Figure 4.48 (a) shows that a remarkable reduction of 180 K can be achieved when the mist ratio is increased from 10% to 15% water mist with 50 µm droplets. The resulted droplets at the film holes have an arithmetic mean diameter of 25.6 µm. This result is very important as it demonstrates the potential of applying the mist cooling technique to the rotor blade outer surface through the film holes. The droplet size at the film holes is proportional to the mist ratio. Figure 4.48 (b) shows that smaller sized droplets produce better cooling of the air, but they don't last long. Also, it shows that for the 10% mist in the base case, droplets are unable to reach the film cooling holes. Figure 4.48 (b) shows that increasing the droplet size is not as effective as increasing the mist ratio since it does not produce any more droplets at the exit film holes.

These results show that mist ratio is a more convenient method to control the final characteristics of the transported mist. Also, it shows that mist cooling is more feasible following the alternative passage through the nozzle diaphragm. Please note that, although this alternate passage is shorter than the previous one, the diaphragm seal spacing is much narrower than the passage going through the pre-swirler and rotor cover-pate cavity. The potential conglomeration of droplets through this very tiny space could escape being captured by the current CFD model. However, experimental study is required to inspect the conglomeration effects though the diaphragm.
4.4.7 Conclusions

From the current investigation, the following conclusions can be drawn:

- Due to the large superheated wall temperature, a thin layer of water vapor with a higher $C_p$ value than air is formed near the wall. This vapor layer shields the core flow from the hot wall, thus leaving the core flow cooler to sustain the water droplets flying longer.

- The mist ratio is found to be a more convenient method to control the final characteristics of the transported mist.

- The mist cooling for high-pressure gas turbine rotor blades is feasible under real operating conditions with some restrictions related to the passage used to deliver the mist to the rotor blades.

- For the rotor cover-plate disk cavity study, the result indicates that using droplets with the initial diameters ranging from 50-90 $\mu$m and a mist mass ratio greater than 15% are suitable to achieve the droplet sizes in the range of 20-40 $\mu$m at the root (hub) of turbine blades.

- Unlike the pre-swirl nozzle - rotor cavity passage, the higher cooling potential produced from the nozzle diaphragm passage is directly affecting the blade external surface as air exits the film holes.

- From the range of operating conditions for the Frame 7 engines, mist cooling is more feasible following the passage through the nozzle diaphragm. Using 15% (wt.) water mist with initial droplet diameters of 50 $\mu$m, a blade wall temperature reduction of 180 K can
be achieved, and the average diameter of the droplets decrease to around 25.6 μm at the film cooling holes, which can be effectively used for film cooling.

- An experimental study is needed to validate the CFD simulation, particularly because the potential conglomeration of droplets through the tiny space through the diaphragm seals could escape being captured by the current CFD model.

The results obtained in this study are encouraging and warrant a continuous study to further investigate the mist transport to the rotating blade through experiments. Furthermore, the mist-induced cooling enhancement may introduce local cold spots or concentrated thermal stresses. Therefore, development of a coupled multi-phase CFD model and finite-element stress analysis will be a worthwhile future task.
CHAPTER FIVE

SUMMARY AND CONCLUSIONS

5.1 Summary

Film cooling scheme has been commonly used to cool the gas turbine airfoils when the turbine inlet temperature is increased to improve the gas turbine efficiency. The idea of injecting tiny water droplets to the coolant to improve the film cooling effectiveness has been investigated by the research group in the Energy Conversion and Conservation Center of University of New Orleans. This study continues the previous work by (a) adding fan-shaped holes and comparing their cooling performance with the round holes, (b) extending the length of the test section from X/D = 40 to 100 to see whether the mist film cooling enhancement prevails farther away from the injection holds, and (c) using computational simulation to investigate the feasibility of transporting mist to the film cooling holes through gas turbine inside passages.

The results show that, with an appropriate blowing ratio, the fan-shaped holes performs about 200% better than round holes in cooling effectiveness and adding 10% (wt.) mist can further enhance cooling effectiveness 170% in average. Farther downstream away from the injection holes (X/D > 50), mist cooling enhancement prevails and actually increases significantly if the cooling film doesn’t lift off from the surface. Droplet measurements from PDPA is extremely important for gaining understanding of droplets’ dynamics during evaporation and their interactions with boundary layer and shear layers of the cooling film. The simulated results show that it is feasible to successfully transport the water mist to the cooling holes on the surface of the turbine airfoils with the appropriate conditions to perform mist cooling.

5.2 Conclusions

A set of conclusions can be drawn in the following sections related to both the experimental and the CFD results.

5.2.1 Experimental work

The results of round hole cases in short section are consistent with previous study by Zhao and Wang (2012). It is found that the general pattern of the adiabatic cooling effectiveness distribution of the mist case is similar to that of the air-only case with the peak cooling
effectiveness occurring at about the same location. This implies that adding mist in the air film
does not change the general cooling pattern largely, so that knowledge acquired through film
cooling regarding the characteristics of many parameters—such as hole geometries, arrangement
of hole spacing, distance between multiple rows, inclination angles, blowing ratios, etc.—can
generally be applied to mist cooling. Also, retrofitting the old air film cooling systems with mist
cooling seems attractive.

Effect of the blowing ratio: For the blowing ratio M= 0.66 cases, the mist cooling effectiveness
is greatly enhanced in all test section setups. The net enhancement of adiabatic cooling
effectiveness, at the end of the test section, compared to the air-only film case reaches 174% in
the round-hole cases, and 170 % in the fan-shaped hole cases. It is noticed that increasing the
blowing ratios reduces the net enhancement in adiabatic cooling effectiveness and reduces the
film coverage due to the lift-off of cooling jet from the surface, resulting in a diminished
effectiveness in protecting airfoil's surface. Note that the current blowing ratio may be
converted to higher values when density difference between the coolant and the main stream
becomes higher.

Effect of mist cooling enhancement in the farther downstream region (X/D= 40~100): Extending the test section longer than X/D=40 has provided valuable information to prove the
hypothesis that mist can notably extend the cooling coverage longer downstream with decent
cooling enhancement of up to 174% for the M=0.66 case and up to 30% for M>1 cases. The
decaying rate of the adiabatic cooling effectiveness,η, for the mist case is much slower than that
of the air-only case. In other words, the cooling effectiveness of the mist film lasts longer than
the air-only film. In addition, the mist cooling generates a much more uniform surface
temperature, which is critical for reducing wall thermal stresses. This is attributed to the presence
and the gradual evaporation of the liquid droplets in the film. From the perspective of application
in gas turbine blade cooling, a longer coverage area is always favorable because it implies that
the cooling performance decays slower, the streamwise temperature gradient is lessened, and
lower thermal stresses result. Also, a lower number of film cooling holes could be used, which
means greater integrity of the blade, lower jet mixing loses, lower cooling air consumption, and
lower manufacturing costs.
**Effect of insulation**: As the insulation is added to the test section wall, the trend of the net enhancement does not change. Only the magnitude of enhancement will either change or remain constant depending on the blowing ratio. For M=0.66, the mist enhancement increases another 22%-40%; for M=1.0, the difference is negligible, and for M=1.4, the mist enhancement decreases slightly.

**Mist effect on cylindrical vs. fan-shaped holes**— For fan-shaped hole geometry, the lateral diffusion causes a remarkable enhancement in the film cooling coverage in both the streamwise and the lateral directions. For the blowing ratio M=0.66 cases, the mist cooling effectiveness is greatly enhanced in all test section setups. In the fan-shaped case, the net enhancement of adiabatic cooling effectiveness ranges from 100% in the near hole region (X/D = 25) to 170% at the end of the test section. The mist enhancement is a bit lower than that achieved by using the cylindrical holes, but the actual mist film cooling effectiveness is still (46%) higher in the fan-shaped case. As a great enhancement (200%) from the cylindrical holes has been achieved in the air-only case, when the fan-shaped hole is employed, mist cooling appears to have less room for greater improvement.

**Droplet dynamics and mist film profile**: Through the examination of droplet size distribution, data rate, velocity, and turbulent Reynolds stresses plots, the process of mist interactions with the coolant film is described, and the profile of the coolant film mixture spreading shape is obtained. Characteristics of droplet distribution due to the interactions with both the flow field and temperature field are analyzed and the results are consistent with previous observation reported by Zhao and Wang (2012). The key points of the physical model are summarized (for both the cylindrical and fan-shaped holes) as follows:

1. The coolant air film layer (mixture of coolant air and droplets) does not always coincide with the “droplet layer” (within which droplets reside). The droplets are discovered traveling outside the coolant air film.
2. The coolant air film, mixed with the mist droplets, keeps its own identity with distinguishably high Reynolds stresses at the boundaries and also with large gradients in the particle size distribution up to X/D=80. Beyond X/D=80, especially with blowing ratios greater than 0.66, the film loses its identity and diffuses with the hot main flow (detaches from the wall). This is clear from the decreasing trend of cooling effectiveness.
3. The initial droplet path and distribution are determined mainly by the initial momentum and fluid mechanics rather than heat transfer.

4. There is a cold core inside the film layer. The droplet size is bigger within the core and smaller towards both the upper and lower boundaries of the cold core layer.

5. The profile of the upper boundary of the air coolant film can be clearly identified from the data rate and droplet size curves. The bottom boundary of the film can be roughly estimated by the location where the data rate is almost zero.

6. Near the cooling holes, droplets outside the film layer (farther away from the wall) are of larger sizes due to the coolant injecting inertia and a subsequent penetration through the air film layer. For cylindrical holes, the droplet size decreases faster from the core towards the upper boundary and decays slower towards the lower boundary. In the upper boundary, the evaporation consumes the small size droplets, so the population density of bigger size droplets increases near the inside of the upper boundary. The population density starts to decrease outside the film.

7. In the cases of fan-shaped holes, it is noticed that big size droplets (>20 μm) only exist near the injection holes. For X/D>13, the droplet sizes are almost homogenously distributed at all elevations with the most frequently captured size as 7-8 μm. The absence of bigger droplets for X/D > 13 may be attributed to the diffusive nature of the fan-shaped holes and its ability to evaporate the water droplets effectively.

8. Based on the above behaviors observed from the experimental measurements, a profile of the “bending back” pattern for the air/mist coolant film is drawn. The bending back location of the mist film layer is at around X/D = 7 for both hole geometries, although, it is more distinguished in the case of the cylindrical holes. A further cross-examination of both the overall heat transfer data and the particle measurements shows that the starting point of the sudden increase in net cooling enhancement is close to the “bending back” location of the mist film. This implies that, as the mist film is bent back and approaches the surface again, the enhancement of cooling effectiveness increases. This “bending back” film pattern is critical in keeping the mist droplets close to the surface, thus improving the cooling effectiveness.
5.2.2 Mist Transport Feasibility Study

CFD mist transport study is conducted to verify the possibility of internally transporting mist, through the high pressure turbine components, to the cooling holes on the vane and rotor surface. Real Frame 7FA operating conditions were taken as an applicable example. According to the location needed to be cooled, different strategies were followed to simulate the mist transport.

1. **For Vanes:** Droplets are injected in the main cooling duct, from an atomizer attached to the engine casing wall, and then tracked in the Lagrangian frame to simulate the mist transport through the vane cavity all the way to the cooling holes on the vane surface. The results show that mist cooling for high-pressure gas turbine vane is feasible under real operating conditions. For example, under the real Frame 7FA operating conditions, 50% of the mist can survive and reach the cooling holes with an average droplet diameter of 10 – 20 µm if the mist with 10% mass ratio and 20-30 µm in initial diameter is injected. Also, the relation between the initial and final injected diameters is found to be linear, which gives a method to control the output droplet diameter knowing the initial injected droplet diameter.

2. **For Blades:** The Situation is complicated because the physical model is rotating. This rotation breaks the connection between the atomizer located at the engine casing wall and the cooling holes on the rotor blade surface. In order to reach the cooling holes on the rotor blade surface, mist has to follow one of the following long paths:

   (a) **Through the vane and rotating disk cavity.** This path is the longest, so compressed water was supplied to a nearby atomizer in the mixing chamber of the pre-swirl nozzle through a transport channel. The results are concluded separately for flow going through the feeding channel and through the disk cavity.

   – **Through the feeding channels:** 1D analysis was misleading as it showed compressed water with a temperature higher than the boiling temperature. This is due to the lack of the wall boiling model in the 1D analysis. For this reason, the 2D axisymmetric CFD study is performed. The CFD results show that it is feasible to have highly pressurized
water transported to the internal engine cooling passages, without suffering excessive boiling, and be further atomized as mist for turbine component cooling. This is feasible under operating conditions of lower wall temperatures (≈ 600 K), higher operating pressures (≈ 110 bar), higher inlet subcooling (≈ 210K), and shorter channel lengths (≈ 0.5 m). A combination of these parameter values can be used with further optimization.

Through the rotating disk cavity: It was not practical, in terms of water consumption and droplet size required, to transport mist to the rotor blades via the pre-swirl nozzle path. Although a high cooling potential is produced, the penalty is big. Being an open loop cooling cycle, mist cooling technique has some restrictions on the water consumption rate, which is linearly proportional to the mist ratio.

(b) Go through the Nozzle Diaphragm from a downstream stator – As the downstream stator is directly connected to the cooling duct flow, mist can be transported from the atomizer on the engine casing wall through the vane cavity of this stator and then reaches the upstream rotor blade through the nozzle diaphragm. Unlike the pre-swirl nozzle rotor cavity passage, the higher cooling potential produced from the nozzle diaphragm passage is directly affecting the blade external surface as air exits the film holes. Droplets of 50 microns diameter and 15 % mist ratio were able to reach the cooling hold with diameters in a range of 20-30 μm to perform the film cooling.

5.3 Future Work Recommendations

The results obtained in this study are very encouraging and warrant a continuous study to further support the feasibility of implementing the mist cooling scheme. For the mist cooling scheme to be approved in the very conservative gas turbine industry, the following tasks are recommended for future research and development:

- Conduct intermediate pressure and temperature experiments at 5 bars and 700°C before move to apply real gas turbine conditions.
- The mist-induced cooling enhancement may introduce local cold spots of overcooling. Therefore, it is essential to develop a coupled multi-phase CFD model and finite-element
stress analysis to investigate the interactions between mist cooling and thermal stresses on the blade materials.

- Due to the impurities in the combusted gases, mixture of steam and combusted gases may introduce corrosion problems on the hot turbine components. An investigation of steam introduced corrosion is recommended.
- More variable parameters can be studied such as different injection and inclination angles, different hole spacing, curved surfaces, density effect, etc.
REFERENCES

- Bestion, D., 2007, “Review of available data for validation of NURESIM two phase CFD software applied to CHF investigations,” The 12th International Topical Meeting on

Summer National Heat Transfer Conference, August 10-14, 2008, Jacksonville, FL, USA


• Ioilev, A., 2007, “Advances in the modeling of cladding heat transfer and critical heat flux in boiling water reactor fuel assembly,” NURETH-12, Pittsburgh, PA, USA.


• http://www.thermopedia.com/content/605


APPENDIX A

AN INVESTIGATION OF LIQUID DROPLET EVAPORATION MODEL USED IN MULTIPHASE FLOW SIMULATION

ABSTRACT

Modeling liquid droplet evaporation in a flow stream is very important in many engineering applications. It was discovered that the result of predicted droplet and main flow temperatures from using commercial codes sometimes presents unexplainable phenomena; for example, the droplet temperature drops too low. The objective of this study is to investigate the issues involved in the built-in droplet evaporation model by using three different approaches: (a) use the existing built-in correlations model in a commercial code, (b) use the lumped analytical analysis, and (c) actually solve the heat and mass transfer by directly using CFD without employing the built-in correlation model. In the third approach, the evaporation process is simulated by imposing water evaporation in a very thin layer at the surface of a stagnant water droplet; in the meantime, the evaporation energy is subtracted from the same place. This is performed by imposing a positive mass source term and a negative energy source term in a thin layer of cells wrapping around the droplet surface. The transport equations are then solved using the commercial CFD solver Ansys/Fluent to track the mass and energy transfer across the shell sides into the liquid droplet and out to the ambient. Unlike the built-in evaporation model in commercial codes, which assumes that all the evaporation energy (latent heat) is supplied by the droplet, in the direct CFD calculation, the evaporation energy is absorbed partly from the droplet and partly from the surrounding air according to the natural process based on the property values and the heat and mass transfer resistance inside and outside the droplet. The direct CFD result (without using evaporation correlation) is consistent with that of the lumped analytical analysis (2nd approach). During the development of the direct CFD calculation, several technical difficulties are overcome and discussed in detail in this work. A revised equation is proposed to improve the existing built-in model in the current commercial code. Both the direct CFD method and the zero-dimensional lumped method show the droplet temperature always increases.
NOMENCLATURE

A  Area [m²]
Bi  Biot number [hLc/Ks]
C_D  Drag Coefficient
C  Vapor concentration [kg/m³], Specific heat [J/kg·K]
D, d  Diameter [m], Mass diffusion coefficient [m²/s]
E  Total energy [J/kg]
h  Convective heat transfer coefficient [W/m²·K]
h_{fg}  Latent heat [J/kg]
K  Thermal conductivity, [W/m·K]
k_C  Mass transfer coefficient [m/s]
L  Length, [m]
m  Mass, [kg]
Nu  Nusselt number [hLc/Kc]
R_{cond}  Conduction Resistance [Lc/KA]
R_{conv}  Convection Resistance, [1/hA]
Re  Reynolds numbers

Greek Letters

μ  Absolute viscosity, [Pa·s]
τ  Shear stress, [N/m²], Time scale, [s]

Subscripts:

f  Fluid
I  Inlet, term number, tensor index (1, 2, 3)
J, k  Term number, tensor index (1, 2, 3)
∞  Carrier phase (away from the droplet surface)
INTRODUCTION

The evaporation of liquid droplets in a flow stream is of interest in many fields of engineering such as liquid fuel spray in a combustor, water spray cooling, gas turbine inlet fogging, mist cooling, etc.

The evaporation of liquid droplets has been studied extensively both numerically and experimentally. As known, numerical simulation, if reasonably validated, provides a fast and a cost effective tool to add the experimental studies. Numerous studies have shown the usefulness of using CFD as a tool to predict the evaporation process. Either commercial or homemade codes have helped to perform that task and have provided enlightened interpretations for the embedded physics of the evaporation process. Occasionally, some glitches appear from using the built-in evaporation models in some commercial CFD codes. For instance, in using one of the commercial codes to calculate liquid droplet evaporation, sometimes the water droplet temperature temporarily reduces significantly and sometimes even below the freezing point during a constant pressure evaporation process. Of course, this cannot happen at a constant-pressure evaporation process because, typically, the droplet temperature should not drop below the wet-bulb temperature. These findings motivate this study to investigate the built-in evaporation models used in some commercial codes.

Classical droplet evaporation theory was developed in the 1950’s by Spalding (Spalding, 1953), which yielded the well-known $d^2$-law (Williams, 1973). In real engineering applications, as in liquid fuel combustion, the collective behavior of clouds of droplets is of great interest. Sprays, created by atomizing liquid fuels, have been considered one of the most efficient ways of burning liquids. The reason for this is that, in a spray, the surface-to-volume ratio is increased greatly compared with a blob of liquid. The extra surface increases the efficiency of heat transport to the liquid, thereby promoting evaporation, ignition, and combustion of the fuel. Of course, the behavior of droplets in a spray is different from that of isolated droplets because of the interaction between droplets and each other and the interactions between drops and the surrounding gases. According to the degree of interaction, the droplet behavior in a spray differs from that of isolated droplets. However, the thermal-flow transport of multiple droplets is more complicated and difficult than of a single droplet; thus, when the literature review is conducted, it is logical to see that studies of a single isolated droplet were first conducted, followed by relating the single droplet evaporation mechanisms to dilute sprays and drop clouds.
The evaporation of isolated droplets in a stagnant air was studied by Law (Law, 1976; Law, 1977). He proposed a diffusion limit model, taking into account droplet transient conduction and giving an accurate representation of single droplet evaporation in a stagnant environment. However, this model for a stagnant ambience is inadequate for the convective droplet vaporization. The empirical correlations of the convective single droplet vaporization were formulated by Frossling (Frossling, 1938) and Ranz and Marshall (Ranz and Marshall, 1952 I, II). Their study on pure liquid droplets confirmed the analogy between heat and mass transfer at low Reynolds numbers. Independent correlations for heat and mass transfer were obtained from the experiments and these empirical correlations have been popular and widely used in most of engineering applications. Refai Ahmed, G. et al. (Refai Ahmed et. Al, 1997) presented an approximate analytical model for predicting forced convection heat transfer from stationery isothermal spheres. Waheed, et al. (Waheed et al., 2002) performed numerical simulations of the mass-transfer of droplets in a continuous phase for a combined-mode problem involving both free- and forced- convections. The effect of free and forced convection on the mass transfer was investigated by solving the complete Navier-Stokes and the convection–diffusion equations using the finite element method. The results showed that superposition of free convection on forced convection didn’t enhance mass transfer.

Smolik and Vitovec (Smolik and Vitovec, 1984) numerically analyzed the quasistationary evaporation of a water droplet into a multicomponent gaseous mixture containing a heavier component besides air. The results demonstrated the possibility of condensation of the heavier component on the surface of evaporating droplet as a result of supersaturation. Ferron and Soderholm (Ferron and Soderholm, 1987) estimated numerically the evaporation time of a pure water droplet in air with a well defined temperature and relative humidity. Their results, approximated by an equation, showed that the droplet evaporation time is primarily a function of the initial droplet diameter and the relative humidity. Then, Miller et al. (Miler et al., 1998) evaluated a variety of liquid droplet evaporation models.

High evaporation rates generate a non-uniform temperature distribution at the surface. This creates a surface tension gradient that produces a forcing in which surface fluid is pulled toward regions of higher surface tension. The viscous force then transports momentum into the interior of the droplet and a convective flow results throughout the droplet. This surface-tension-driven instability flow is called the Marangoni instability (Cloot and Lebon, 1990). A moving
surface should also drive convection exterior to the droplet increasing the exterior mass and heat transport. This increased heat and mass transport should increase the evaporation rate. On the other hand, the convection should also tend to equilibrate the temperature at the surface decreasing any surface tension gradients and, therefore, should tend to reduce the convection.

Spells (Spells, 1952) revealed experimentally the circulation patterns in drops of glycerine falling in castor oil and hence subjected to shearing force across its surface. Hegseth et al. (Hegseth et al., 1996) have experimentally shown that when a droplet evaporates quickly enough, it exhibits a vigorous interior flow. This flow is driven by surface tension gradients resulting from the non-uniform temperature distribution at the surface.

Unsteady droplet vaporization with slip and an internal circulation has been studied analytically and experimentally by Prakash and Sirignano (Prakash and Sirignano, 1978). They studied in detail the vaporization of fuel droplets flowing with high Reynolds number through a stagnant gas. Similarly, the unsteady investigations of vaporizing hydrocarbon droplets were studied by Dwyer and Sanders (Dwyer and Sanders, 1987). It was assumed that the droplet was always at the boiling point; therefore, the initial liquid heat-up process was not studied. The effects of the internal circulation were neglected.

Regarding the evaporation of a droplet in a cloud of droplets, Milburn (1957) had studied the mass and heat transfer process within finite clouds of water droplets. He developed a simple nonlinear differential equation to govern the propagation of vapor concentration, temperature, and droplet size in space and time. Kouska et al. (1978) solved the modified Maxwell equation for droplet clouds to evaluate the evaporation rate of mono-disperse water droplets. When the concentration of droplet clouds is sufficiently low, the results of the numerical calculation for droplet clouds agree well with those of a single water droplet. The equilibrated system, where a water droplet cloud is steadily mixed with unsaturated air, was also analyzed.

**PROBLEM STATEMENT AND HYPOTHESIS**

As mentioned earlier, the problem which motivated this study was originated from the observation during the application of a commercial code to calculate liquid droplet evaporation that, sometimes, the water droplet temperature temporarily reduces significantly below the wet-bulb temperature and sometimes even below the freezing point during a constant pressure evaporation process. As the problem occurred during the droplet evaporation process, it was
speculated that the built-in droplet evaporation model might have a problem. The first approach is to examine the fundamental physics involved in the droplet evaporation process. As the droplet evaporates, the surface water molecules need some energy to depart from the surface and change into the vapor phase. This energy is the latent heat of vaporization; but the question remains as to where this energy is coming from. It is hypothesized that the energy is transferred from the surrounding gas, as well as from the droplet itself. Therefore, the relative amount of heat coming from outside and inside the droplet should be weighted according to the differences in the thermal resistances of the heat transfer paths outside and inside the droplet, respectively. These relative proportions of the thermal resistances outside the droplet (convection) and inside the droplet (conduction) are determined with the evaluation of the Biot number. Hence, the droplet evaporation model should reflect the fundamental physics of liquid evaporation by including the mechanism of absorbing latent heat from both the surrounding gas, as well as from the droplet itself as controlled by the Biot number.

The current work starts with conducting a fundamental study of water droplet evaporation models and physical processes. It eventually aims at modifying the current built-in evaporation models in an existing commercial CFD code. The following approaches are taken sequentially in this study.

(a) Use Ansys/Fluent CFD code with existing built-in evaporation model and associated empirical correlations to study the evaporation process of water droplets.
(b) Use a simple zero-dimensional lump-capacitance model to analyze the problem and modify the droplet evaporation energy equation.
(c) Solve the heat and mass transfer of a single droplet during evaporation by directly using CFD without employing the built-in model. This provides an opportunity to understand the physics of the droplet evaporation process in depth and verify the hypothesis proposed in this study. (Note: This is not the direct numerical simulation, or DNS.)

(I) CFD CALCULATIONS USING BUILT-IN EVAPORATION MODEL

In this approach, the existing evaporation model available in Ansys/Fluent CFD code is used. A feasible method to simulate the evaporation of water droplets is to consider the droplets as a discrete phase since the volume fraction of the liquid is small (less than 0.1%). The trajectories of the dispersed phase (droplets) are calculated by the Lagrangian method (Discrete
Phase Model, DPM). The impacts of the droplets on the continuous phase are considered as source terms to the governing equations of mass, momentum, energy, and species. The discrete phase model (DPM) along with the standard evaporation model is used.

The Computational Domain

The computational domain is a 2D duct \((0.4 \text{ m} \times 3 \text{ m})\). The droplets are injected as a group of 4 streams equally spaced at the inlet. The ratio of the mass flow rate of the injected water to the air mass flow rate has been chosen to be small \((1 \times 10^{-6})\). This ensures that the droplet-droplet interaction effects can be neglected and makes the assumption of isolated droplet reasonable.

Mathematical Model for the Carrier Phase

Air, which is the carrier (continuous) phase, is treated as a perfect gas mixture composed of two species: air and water vapor. The governing equations include conservation of mass, conservation of momentum, conservation of energy, and conservation of species. In addition to these governing equations, other auxiliary equations are used to calculate the density, viscosity, and thermal conductivity for the mixture.

Governing equations

Since the droplet Reynolds number, based on the slip velocity, is approximately 3, laminar flow is assumed. The following are the governing equations of mass, species, momentum, and energy for unsteady laminar flow conditions.

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_1} (\rho u_i) = S_m \tag{A.1}
\]

\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_1} (\rho u_i u_j) = \rho g_j - \frac{\partial \rho}{\partial x_j} + \frac{\partial c_{ij}}{\partial x_i} + S_f \tag{A.2}
\]

\[
\frac{\partial (\rho c_p T)}{\partial t} + \frac{\partial}{\partial x_1} (\rho c_p u_i T) = \frac{\partial}{\partial x_i} \left( \frac{\partial T}{\partial x_i} \right) + \mu \Phi + S_h \tag{A.3}
\]

\[
\frac{\partial \rho Y_i}{\partial t} + \frac{\partial}{\partial x_1} (\rho u_i Y_j) = \frac{\partial}{\partial x_i} \left( \rho D_{ij} \frac{\partial Y_j}{\partial x_i} \right) + S_j \tag{A.4}
\]
where $\tau_{ij}$ is the symmetric stress tensor. The source terms ($S_m, S_j, S_f,$ and $S_h$) are used to include the contributions of the droplet evaporating species, droplet forces, and evaporation energy from the dispersed phase (water droplets). During evaporation, water vapor diffuses and is convected into the surrounding gas through the species transport equation, Eq. A.4. Here $Y_j$ is the mass fraction of species $j$ in the mixture; $S_j$ is the source term of species; and $D_j$ is the diffusion coefficient.

### Discrete Phase Model (Water Droplets).

As the injected water mass flow is very small and the droplet diameter is also small, the volume fraction of water droplet to air is expected to be small (<10 %), which is the sufficient condition to use the DPM. Following the governing equations for the droplet flow:

#### Droplet Flow and Heat Transfer

- The droplets are tracked by Lagrangian method by applying Newton’s 2nd Law with the following equation of motion

$$m_p \frac{dV_p}{dt} = \sum F = F_D + F_g + F_{th} + F_S \quad (A.5)$$

where $m_p$ is the droplet mass, and $v_p$ is the droplet velocity (vector). The right-hand side is the combined force acting on the droplets, including $F_D$ (drag force), $F_g$ (gravity and buoyancy force), $F_S$ (Saffman lift force) and $F_{th}$ (thermophoretic force). For more details about these forces see (ANSYS, 2009).

The droplet temperature change depends on convection, evaporation, and radiation. The energy equation for any individual droplet can be given as the following equation:

$$m_p c_p \frac{dT}{dt} = \pi d^2 h(T_\infty - T) + \frac{d m_p}{dt} h_{fg} + \text{Radiation} \quad (A.6)$$

where $h_{fg}$ is the latent heat. The radiation heat transfer term can be reasonably neglected because the range of temperatures is low in this study.

The convective heat transfer coefficient ($h$) can be obtained with an empirical correlation (Ranz and Marshal, 1952 I, II) as follows:

$$Nu_d = \frac{h d}{\lambda} = 2.0 + 0.6 \text{Re}_p^{0.5} \text{Pr}^{0.33} \quad (A.7)$$

where $Nu$ is the Nusselt number, and $Pr$ is the Prandtl number.
The evaporated mass is calculated by two modes: evaporation and boiling. During the evaporation mode, the evaporated mass change rate or vaporization rate is affected by the relative humidity in the air and is shown in Eq. (A.8) as being governed by the concentration difference between droplet surface and the air stream,

\[- \frac{dm_p}{dt} = \pi d^2 k_c (C_s - C_\infty) \quad (A.8)\]

where \(k_c\) is the mass transfer coefficient and \(C_s\) is the vapor concentration at the droplet surface, which is evaluated by assuming that the flow over the surface is saturated. \(C_\infty\) is the vapor concentration of the bulk flow, which is obtained by solving the transport equation in the computational cell. The value of \(k_c\) can be given from a correlation analogous to Eq. (A.7) (Ranz and Marshal, 1952 I, II) as follows:

\[Sh_d = \frac{k_c d}{D} = 2.0 + 0.6 \text{Re}^{0.5} \text{Sc}^{0.33} \quad (A.9)\]

where \(Sh\) is the Sherwood number, \(Sc\) is the Schmidt number (defined as \(\nu/D\)), and \(D\) is the mass diffusion coefficient of the water vapor in the bulk flow.

When the droplet temperature reaches the boiling point, the following equation can be used to evaluate its evaporation rate (Kuo, 1986):

\[- \frac{dm_p}{dt} = \pi d^2 \left(\frac{\lambda}{d}\right) \left(2.0 + 0.46 \text{Re}^{0.5}_d\right) + \ln\left(1 + c_p (T_\infty - T)/h_{fg}\right)/c_p \quad (A.10)\]

where \(\lambda\) is the gas/air heat conductivity and \(c_p\) is the specific heat of the bulk flow.

Theoretically, evaporation can occur at two stages: (a) when the temperature is higher than the saturation temperature (based on local water vapor concentration), water evaporates according to Eq. A.8, and the evaporation is controlled by the water vapor partial pressure until 100% relative humidity is achieved; (b) when the boiling temperature (determined by the air-water mixture pressure) is reached, water continues to evaporate according to Eq. A.10. After the droplet evaporates due to either high temperature or low moisture partial pressure, the water vapor is transported away due to convection and diffusion as described in the water vapor species transport equation A.4.
CFD Model Results

In DPM calculations, the droplet behavior and evaporation characteristics are treated globally as average properties in the injected parcels. Detailed local results, like pressure distribution or heat transfer coefficient along the surface and the temperature distribution inside the droplet, cannot be obtained. This lack of information obtained by CFD and thermodynamic calculations was the motivation to perform more detailed calculations using a single water droplet in an air stream in Approach 3. The result of droplet evaporation time and status (how much is evaporated) obtained from CFD model results will be used to compare with the result from Approach 3.

Boundary Conditions

The inlet condition of the domain is specified by a flow velocity. The outlet condition is set at the constant atmospheric pressure, and the sides of the domain are taken as periodic. Regarding the discrete phase, a mono-disperse spray composed of 4 streams is injected at the inlet of the duct. The base case parameters, according to typical droplet evaporation values, are shown in Table A1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Base Case Value (BC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet air temperature</td>
<td>310 K</td>
</tr>
<tr>
<td>Inlet air Velocity</td>
<td>1 m/s</td>
</tr>
<tr>
<td>Droplet Inlet Velocity</td>
<td>3 m/s</td>
</tr>
<tr>
<td>Droplet Re Number</td>
<td>2.63</td>
</tr>
<tr>
<td>Droplet diameter</td>
<td>20 µm</td>
</tr>
<tr>
<td>Droplet Temperature</td>
<td>300 K</td>
</tr>
<tr>
<td>Relative Humidity</td>
<td>60 %</td>
</tr>
</tbody>
</table>

Evaporation time (lifetime) of the droplets is the most favorable result sought. Figure A1a shows the variation of the droplet's diameter, velocity, and temperature. The lifetime of the
The variation in diameter also conforms with the \( d^2 \)-law of droplet evaporation which can be expressed as

\[
d_0^2 - d^2 = \lambda t
\]

(A.11)

where \( \lambda \) is the droplet evaporation constant obtained from the figure as \( 9.37 \times 10^{-10} \text{ (s/m}^2) \) for the base case.

Figure A1b shows the variation of the droplet velocity. The droplet quickly catches the main flow speed, due to its interaction with the air flow through drag, after a duration called the hydrodynamic response time which is approximately 0.004 s here. Similarly, Fig. A1c shows the variation of the droplet temperature with a thermal response time of 0.0025 s.

Figure A1 Variation of the droplet characteristics (a) Normalized droplet diameter shown in \( d^2 \)-law (b) droplet velocity (c) droplet Temperature

(II) ZERO-DIMENSIONAL MODELS

1-D Lumped Capacitance Model

A water droplet evaporates due to the heat transfer between the surrounding medium and the droplet. It is hypothesized that, when the liquid water molecules evaporate on the surface of the liquid droplet, the latent heat is supplied by both the ambient air and the interior of the droplet with the fractions being correlated to the ratio of their corresponding equivalent thermal resistances, or Biot number. The energy associated with the phase change is the latent heat of vaporization of the liquid. Evaporation occurs when liquid molecules near the liquid surface experience collisions that increase their energy above that needed to overcome the surface binding energy. The energy required to sustain the evaporation must partially come from the
internal energy of the liquid, which would then experience a reduction in temperature (The Cooling Effect). The latent energy lost by the liquid because of evaporation must be replenished by energy transfer to the liquid from its surroundings. Neglecting radiation effects, this transfer may be due to the convection of sensible energy from the gas or to heat addition by other means.

The existing built-in model in Ansys/Fluent models latent heat being absorbed only from inside the droplet and followed by transferring heat from the surrounding air to heat up the droplet, as shown in Eq. A.6. Therefore, it is speculated that the previously observed situations—when the droplet temperature becomes very low and occasionally drops to below freezing temperature intermittently during evaporation process—were caused by the model formulated as Eq. A.6. After the droplet temperature drops significantly, the subsequent heat transfer increases significantly to transport energy from the surrounding warmer air to quickly heat up the droplet. Although this behavior is transient and occurs intermittently and temporarily, it is not reasonable. This behavior should not be confused with the well-known droplet supercooling because the medium temperature (main stream), in our case, is well above the water freezing point. To investigate the proposed hypothesis, Eq. A.6 will be modified to include a term which will allow energy to be transported to the liquid droplet surface from the surrounding air based on the Biot number. The modification is based on the following model development.

Consider an equivalent thermal circuit for heat transfer path between the hot gas and the droplet in a zero-dimensional lumped capacitance model as shown in Fig. A2(a).

The Biot number, Bi, gives a simple index of the ratio of the thermal resistances inside and outside of the droplet. In the traditional definition of Biot number, conduction is treated as the dominant heat transfer mode; and no convection and associated internal flow activity are considered. This ratio also affects—although it does not necessarily determine—the corresponding fraction of heat that is transferred from outside versus from inside; thus, it affects the change of droplet temperature. The definition of Biot number is

\[
Bi = \frac{hL}{k_d} = \frac{r_{\text{cond}}}{r_{\text{conv}}} \tag{A.12}
\]

where \( h \) is the heat transfer coefficient of the gas and \( k_d \) is the thermal conductivity of the droplet. \( L \) is the characteristic length (taken as the droplet radius).
Estimates of the Biot number of two different droplet sizes at 20 μm and 1 mm are shown in Tables A2 and A3 respectively. The Ranz and Marshal correlations (Ranz and Marshal, 1952 I, II) are used to calculate Nu and, hence, the convective heat transfer coefficient.

![Energy Sink](image)

**Figure A2(a)** Thermal resistance circuit illustrates the hypothesis that the evaporation energy is sucked from both inside and outside of the liquid droplet.

Although Biot number can help provide an idea of how thermal resistance ratio would affect heat flow distribution, the actual heat flow ratio ($E_d/E_\infty$) is still undetermined until the temperature gradients are known on either side of the droplet-air interface. In principle, the lumped capacitance approach encounters some ambiguity because the temperatures inside a droplet are assumed uniform and, hence, there is no temperature gradient inside the droplet. This ambiguity can be clarified by assuming the droplet temperature is at the center of the droplet, so the 1-D thermal circuit shown in Fig. A2(a) can be applied. The 1-D approach can also provide a quick estimate of Droplet Response Time and Evaporation Time.

**Table A2 Estimate of Biot number values for different slip velocities of a droplet with d=20 μm**

<table>
<thead>
<tr>
<th>$U_{slip}$ (m/s)</th>
<th>Re</th>
<th>Nu</th>
<th>$h$ (W/m².k)</th>
<th>Bi</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.01</td>
<td>0.01</td>
<td>2.06</td>
<td>2707.9</td>
<td>0.044</td>
</tr>
<tr>
<td>0.1</td>
<td>0.12</td>
<td>2.19</td>
<td>2876.6</td>
<td>0.047</td>
</tr>
<tr>
<td>1</td>
<td>1.23</td>
<td>2.59</td>
<td>3409.9</td>
<td>0.056</td>
</tr>
<tr>
<td>2</td>
<td>2.46</td>
<td>2.84</td>
<td>3732.9</td>
<td>0.061</td>
</tr>
</tbody>
</table>
Table A3 Lumped capacitance calculations for droplet with d=1 mm

<table>
<thead>
<tr>
<th>$U_{slip}$ (m/s)</th>
<th>Re</th>
<th>Nu</th>
<th>$h$ (W/m².k)</th>
<th>Bi</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.01</td>
<td>0.62</td>
<td>2.42</td>
<td>63.63</td>
<td>0.0519</td>
</tr>
<tr>
<td>0.1</td>
<td>6.16</td>
<td>3.33</td>
<td>87.48</td>
<td>0.0713</td>
</tr>
<tr>
<td>1</td>
<td>61.56</td>
<td>6.19</td>
<td>162.89</td>
<td>0.1328</td>
</tr>
<tr>
<td>2</td>
<td>123.12</td>
<td>7.93</td>
<td>208.58</td>
<td>0.1701</td>
</tr>
</tbody>
</table>

**Droplet Hydrodynamic Response Time**

Both droplet drag and heat transfer depend on the slip velocity between the droplet and the air. The elapsed time (response time) of a particle to achieve the main flow velocity and to reach the main flow temperature varies with the droplet size. The equation of motion for a spherical particle in the air is given by

$$ m \frac{dv}{dt} = \frac{1}{2} C_D \pi d^2 \rho_g (u - v) |u - v| $$  \hspace{1cm} (A.13)

where $u$ is the gas velocity and $v$ is the droplet velocity.

Different hydrodynamic response times can be obtained by using different drag models. For example, by using the Stokes drag law, the hydrodynamic response time is obtained as:

$$ \tau_h = \frac{\rho_d d^2}{18 \mu_g} $$  \hspace{1cm} (A.14)

Following the Schiller and Neumann (1935) drag model, the hydrodynamic response time can be obtained as

$$ \tau_h = \frac{\rho_d d^2}{18 \mu_g [1 + 0.15 Re^{0.687}]} $$  \hspace{1cm} (A.15)

Similarly, the thermal response time ($\tau_t$) is defined as the elapsed time required for the droplet to achieve thermal equilibrium with the mean flow bulk temperature. It can be obtained from Eq. A.6, assuming the temperature is uniform throughout the particle with negligible radiation as:

$$ \tau_t = \frac{\rho_d d^2 c_p}{6 Nu K_g} $$  \hspace{1cm} (A.16)
Estimate of different slip velocities and two different droplet sizes (20 μm and 1 mm) is shown in Table A4. The hydrodynamic response time ranges from 0.77 s to 0.95 s, while thermal response time is longer, around 3.06 s; whereas, the CFD calculations show much lower values (0.005 and 0.003, respectively)

<table>
<thead>
<tr>
<th></th>
<th>Stokes law</th>
<th>Schiller &amp; Naumann</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \tau_h )</td>
<td>0.95s</td>
<td>0.77s</td>
</tr>
<tr>
<td>( \tau_t )</td>
<td>3.06s</td>
<td>(for forced convection)</td>
</tr>
</tbody>
</table>

**Table A4 Particle hydrodynamic and thermal response times**

**Droplet Evaporation Time** - Several models have been proposed for calculating droplet evaporation time in the open literature. Zheng et al. (2002) modeled the droplet evaporation time as

\[
t = \frac{R_d d^2}{8D_v \left( \frac{P_a - P_d}{P_a - P_{sat}} \right) M}
\]  

(A.17)

White and Meacock (2004) proposed another droplet evaporation model as

\[
t = \frac{a^2}{8\rho_a D_v \ln\left( \frac{1+\omega_1}{1+\omega_0} \right)}
\]  

(A.18)

where \( \omega_o = 0.02422 \) and \( \omega_1 = 0.02723 \) are the specific humidity (kg/kg dry air) at dry bulb and wet-bulb temperature, respectively. \( P_d = 3.567 \) kPa is the droplet saturation pressure \( P_d = P_{sat} \) (\( T_d=300 \) k), and \( D_v = 2.88E-05 \) m²/s , is the diffusion coefficient. The estimated evaporation times using both models are shown in Table A5.

**Table A5 Evaporation time in the base case**

<table>
<thead>
<tr>
<th>Case</th>
<th>( t ) (Zheng), s</th>
<th>( t ) (White), s</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base Case</td>
<td>0.00254</td>
<td>0.00052</td>
</tr>
</tbody>
</table>
A large difference is seen when the results from 1-D model are compared with that calculated by the CFD simulation in approach 1. It seems that the droplet evaporation time as calculated with Zheng’s model is twice longer than that calculated with White’s model.

Modified droplet evaporation energy equation

Considering the aforementioned effect of Bi, the droplet evaporation energy equation (Eq. A.6) is therefore modified as:

\[ m_p c_p \frac{dT}{dt} = \pi d^2 h(T_{\infty} - T_d) + f(B_i) \frac{dm}{dt} h_{fg} \]  

(A.19)

with \( f(B_i) \) being a function of Bi number and thermal response time. As \( Bi \rightarrow 0 \), the proposed function \( f \rightarrow 1 \), indicating that all the latent heat comes from the droplet. On the other hand, when \( Bi \rightarrow \infty \), \( f \rightarrow 0 \) which means that convection term is dominant and energy is sucked from outside the droplet. The function \( f \) can be suggested to be in the exponential format as \( \exp(-\phi Bi) \).

Where \( \phi \) is a constant, related to the droplet evaporation rate, to be obtained from experiment.

This approach will allow less latent heat to be sucked from the droplet than Eq. (A.6) and will help avoid overcooling of the droplet. The remaining part of the work in Approach 3 will be dedicated to performing a detailed CFD calculation surrounding a single droplet without using the built-in droplet evaporation model and empirical equations.

(III) SINGLE DROPLET CFD SIMULATION WITHOUT USING BUILT-IN EVAPORATION MODEL

In this approach, the evaporation model of FLUENT is turned off, and evaporation is simulated by assigning a small amount of liquid evaporating from a very thin shell layer surrounding the droplet's surface. Then, the corresponding amount of latent heat required for evaporation is assigned as an energy sink in this thin evaporation layer. The evaporated water vapor on the droplet surface will be transported away from the surface. The local flow behavior and heat transfer will be calculated by the governing equations (Eq. A.1-A.4), and no empirical correlations (Eqs. A.7-A.10) will be used. The main objective is to calculate the heat flow distribution around the droplet surface and to determine the fraction of heat flow sucked from the droplet's interior versus from the surroundings. A transient CFD model is setup with a stationary droplet suspending near the inlet of a long 2D duct. The computational domain is constructed using Gambit, and the discretized control volume form of governing equations is solved using
FLUENT. These calculations, compared to the built-in evaporation model, will give us the ability to do the following:

(a) Perform the evaporation process away from the suspected built-in evaporation model,
(b) Solve the flow field outside, as well as inside the droplet. (Unlike the DPM model which deals globally with the droplet parameters, the calculation procedure gives complete details of the flow field and the temperature inside the droplet.),
(c) Gradients inside and outside the droplet can be calculated and the relative amounts of heat ($E_d$ and $E_\infty$) can be estimated.

To perform this task in the FLUENT environment, two features are used: **Fixing of Solution Variables Technique** and **Source Terms**.

**Fixing the Solution Variables Method** - This method gives the ability to deal with the droplet as a single phase liquid continuum fixed in the computational domain. It simply prevents solving the species transport equations in a specific part of the domain, which is the droplet volume in our case. In the present study, the mass fraction of liquid water is fixed as one to simulate a liquid droplet 100% water with the desired diameter. Accordingly, the species transport equation is not solved in the droplet volume to keep it always in the liquid phase although other transport equations (momentum and energy) are still solved to enable studying the flow and temperature fields inside the droplet. Without this treatment, the air will diffuse into the liquid droplet, and the liquid filaments can be seen to diffuse and be convected in the air stream. This treatment requires the creation of three different zones inside the computational domain: (a) the droplet zone with water liquid mass fraction of 1, (b) the shell zone around the droplet with mass source for water vapor and energy sink to simulate liquid evaporation, and (c) the outer domain represents the rest of the flow field. These three distinct cell zones are connected via non-conformal grid interfaces to allow the transport equations to be solved in the whole domain. These zones are schematically shown in Fig. A2(b).

**Source and Sink Terms** - A positive mass source term ($S_m$ and $S_j$ in Eqs A.1 and A.4) is used to inject water vapor in the shell zone accompanied by the subtraction of a corresponding amount of energy via a negative energy source term ($S_h$ in Eq.A.3), as shown in Fig. A2(b). This process
forces the water vapor to be in the saturated state surrounding the liquid surface and is transported away by diffusion and convection. The heat flows will be naturally developed based on the property values and governing equations. It is important to mention that these added mass and energy sources are Volumetric Rate Sources, not cell sources. This means that the units must be (kg/m³.s) and (W/m³) respectively. This is performed by dividing the mass source by the shell volume and the estimated evaporation time calculated using the CFD calculations as shown in Fig. A1(a).

![Diagram showing energy and vapor flow directions at the droplet surface (within the shell zone)](image)

**Figure A2(b) Schematic shows the energy and vapor flow directions at the droplet surface (within the shell zone)**

**Technical Challenges**

1. During the CFD simulation, many technical challenges were encountered. Some of the major issues are described below. The water liquid was noticed to diffuse out of the droplet zone as a result of the diffusion term in the liquid water species transport equation. Of course, during the real droplet evaporation process only water vapor, not liquid water, diffuses outwardly from the droplet surface. Since the real physics does not involve this kind of liquid phase diffusion, it was necessary to stop this numerically induced error. To prevent the unreal diffusion of liquid water molecules, the *liquid water* species transport equation is forced not to be solved outside the droplet domain by fixing the mass fraction of water liquid to be always zero outside the liquid droplet.
2. The same problem happened to the water vapor outside the droplet zone; it started to diffuse into the droplet. This problem was similarly resolved by fixing the mass fraction of water vapor species to be always zero inside the droplet domain.

3. During the earlier stage of computation, the numerical iteration had always diverged without an apparent reason. Many efforts were spent on investigating the typical numerical procedures and issues that would lead to divergence without much success. Finally, it was discovered that the main reason of divergence was attributed to the mesh quality in the droplet-shell-domain interface. This interface problem was intricate and difficult to be identified; however, its effect is devastating. In one of the many different attempts to search for the answer to the divergence problem, the flow was solved in many steps with gradual complications, starting with only a solid droplet without a shell, and ending with the current problem formulation as shown in Fig. A4. It can be concluded that in any kind of problem which contains a non-conformal grid interface, it is necessary to implement very dense grid at the interface between two zones. This very dense grid reduces the irregularities on the matching interface zones and yields correct flux calculations through the interface zone.

4. Earlier results showed a great disparity in values of heat flux on two sides of the droplet interface. This disparity indicates that something is not correct in the computational process. After investigation, the built-in method of calculating the thermal conductivity and density in the commercial code was found to be the culprit. It gave the property values close to those of air and water vapor on both sides of the interface, rather than giving the property values of liquid water in the liquid domain. It was not clear how this would happen in a domain in which 100% of the material was assigned to be water, but this problem was resolved by adopting the mass weighted average method instead of the ideal gas method. When the ideal gas density calculation method was used in the whole domain (for pressure-based solvers), the droplet density was not correctly calculated. It appears that FLUENT used the properties of liquid and water vapor in the calculations for the liquid-only droplet. After investigation, the mass weighted average method was used to calculate density and thermal conductivity. It gave correct results for water density in the droplet domain after assigning the water vapor mass fraction as zero. The mass weighted average method along with the zero mass fraction of water vapor in the droplet zone removes any possibility of
using water vapor properties in the calculations. It simply calculates the properties of the mixture as a simple mass fraction average of the pure species properties. For more details, see (ANSYS, 2009).

**Mesh Generation**

The grid consists of three separate cell zones: one for the droplet with diameter 20 µm, the second for a very thin shell layer around the droplet with a thickness of 0.2 µm, and the third represents the rest of the computational domain. The three separate cell zones are generated and meshed separately in Gambit, the preprocessor of the FLUENT package, and then are combined together in the solver. GAMBIT generates structured (mapped), unstructured (paved), and hybrid meshes. The mesh used for the model is mainly unstructured except inside the thin shell layer wrapping around the droplet surface and near the interfaces between the droplet, the shell, and the rest of the domain. At these interfaces, the mesh takes the form of structured grid called boundary layer (B.L.) mesh which gives a good resolution at these boundaries with a large gradient in the solution variables. Great care must be taken in generating the mesh at these interfaces to prevent any possible mismatching of the non-conformal grid interface. This could be a main source of divergence and errors during calculation. To avoid these problems, the mesh is designed to be greatly clustered around the interfaces to give a better geometrical resolution, as shown in Fig. A4a. An amplified view of Fig. A4a shows details of the interfaces in Fig. A4b.

As mentioned earlier, great care must be taken in generating the mesh at interfaces which require the clustering of the mesh at these places. Accordingly, the mesh size was successively increased to achieve that purpose and produced a well-converged solution. Taking into account that the flow is laminar and two dimensional, the final mesh size of 139,531 cells was considered sufficient to perform the calculations of the base case. A grid sensitivity study was performed. The mesh sensitivity study was done using a mesh adaption technique. Basically, the solution is calculated first with the primary mesh (139,531 cells), and all the solution variables are recorded. Then the mesh is adapted (refined) in two consecutive steps, namely, boundary adaption around the interfaces followed by volume adaption to achieve a smooth volume transition between cells. After the mesh adaption procedure is done, the solution is recalculated on the refined mesh and solution variables values are compared to the primary mesh before any adaption. The whole adaption and comparison process is repeated three times increasing the mesh size to 165K, 201K, and 341K cells, respectively. This adaption yields a final mesh size which is 2.5 times the
original one. The results of the mesh sensitivity study are shown in Table 6 below. All solution variables are compared but only mass fraction of water vapor and static temperature at exit of the domain are shown here before and after the third adaption. It is clear from the third adaption (refinement) that increasing the mesh size 2.5 times only changes the solution variables by less than 3% (2.8%). It is clear that the solution is almost mesh independent.

Table A6 Mesh sensitivity study results

<table>
<thead>
<tr>
<th></th>
<th>139K Cells</th>
<th>341 K Cells</th>
<th>Change</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exit vapor mass fraction (-)</td>
<td>2.2639E-07</td>
<td>2.1990E-07</td>
<td>2.8%</td>
</tr>
<tr>
<td>Exit Static Temperature (K)</td>
<td>308.116</td>
<td>308.065</td>
<td>0.017%</td>
</tr>
</tbody>
</table>

Figure A4 Computational grid (a) Whole domain (b) In the vicinity of the shell zone

Boundary Conditions

Along the inlet boundary, air velocity of 1 m/s is specified at the inlet boundary with zero concentration of water vapor. Atmospheric static pressure is imposed at the exit boundary. Periodic boundary conditions are used on sides of the domain to simulate the external flow around the droplet. Interface boundary conditions are used between the droplet, the shell, and the domain.
Results and Discussions

Base case results

The base case parameters are shown in Table A7 and are selected according to typical droplet evaporation values as in the CFD calculations in Approach I of the study except for the relative humidity, which is not taken into account in one of the runs. The air velocity is taken to be 1 m/s to give a droplet Reynolds number of 2.63, and the temperature is 310 K which gives a Prandtl number of 0.704. For numerical accuracy, the second order upwind scheme is used for space discretization. A time step size is calculated based on the reasonable Courant number stability criterion. The Courant Number can be roughly estimated for 1D case as $u \Delta t / \Delta x$; where $u$ is the air velocity, $\Delta t$ is the timestep size, and $\Delta x$ is the length interval for the computational cell. A Courant number of 0.2 was selected to achieve numerical stability. Accordingly, a time step size of $2.713 \times 10^{-4}$ s was calculated and used for the calculations based on the velocity and mesh size intervals in the base case. To make sure that the solution is time step size independent, a sensitivity study is performed. Two other time step sizes are used to calculate the solution, and the solution results are compared with the original time step size results. Results of the time step size analysis are shown in Table A8. It is clear that the solution is time step size independent because increasing or decreasing the time step size an order of magnitude has a negligible effect on the solution variables.

Table A7 Base case parameters.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Base Case Value (BC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air temperature</td>
<td>310 K</td>
</tr>
<tr>
<td>Inlet Air Velocity</td>
<td>1 m/s</td>
</tr>
<tr>
<td>Inlet RH</td>
<td>0</td>
</tr>
<tr>
<td>Droplet Re</td>
<td>2.63</td>
</tr>
<tr>
<td>Droplet diameter</td>
<td>20 µm</td>
</tr>
<tr>
<td>Droplet temperature</td>
<td>300 K</td>
</tr>
<tr>
<td>Shell Thickness</td>
<td>0.2 µm</td>
</tr>
<tr>
<td>Water Vapor Mass Source</td>
<td>1 % of the droplet Mass</td>
</tr>
</tbody>
</table>
### Table A8 Time step (s) sensitivity study results (change from the base case of 2.713 E-04 s)

<table>
<thead>
<tr>
<th>Change in</th>
<th>exit vapor mass fraction</th>
<th>2.713 E-05</th>
<th>2.713 E-04</th>
<th>2.713 E-03</th>
</tr>
</thead>
<tbody>
<tr>
<td>Change in</td>
<td>exit static temperature</td>
<td>0.11 %</td>
<td>0 %</td>
<td>0.002 %</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.00032</td>
<td>0 %</td>
<td>0.00032 %</td>
</tr>
</tbody>
</table>

#### Static temperature results

The results of static temperature in the base case are shown in Fig. A5. Figure A5(a) displays the temperature in the whole domain and Fig. A5(b) shows an enlarged view of the temperature field near and inside the droplet. The isotherms in Fig. A5(b) take almost concentric circles in the vicinity around the droplet. This isotherm pattern is expected due to the dominance of diffusion effects in this case because of the small Reynolds number of 2.6. This means that the forced convection effects are negligible. Therefore, the heat transfer is due mainly to the natural convection and the heat diffusion. The effect of natural convection can be discerned by the slightly asymmetric isotherm pattern in Fig. A5(a) with a bit more rising isotherm in the upper domain downstream of the droplet.

Figure A5(b) shows the temperature distribution inside the droplet is uniform due to its low thermal resistance. This agrees with the Biot number results obtained in the second approach of the study using the lumped capacitance method.

Finally, the droplet temperature reaches the steady-state temperature about 304 K, which is 4 K higher than its initial temperature. This means that the droplet passed the transient heat-up period by absorbing heat from the surrounding air.

#### Velocity results

Figure A6(a) shows the velocity field inside the droplet. The internal free-convection (flow) is described by the Rayleigh number, which compares buoyancy forces (causing convection) to diffusive and viscous forces (working against it). For our situation, the Rayleigh number can be defined as follows (Shahidzadeh et al., 2006):

\[
Ra = g \left( \frac{\Delta \rho}{\rho} \right) \frac{d^3}{\nu D}
\]  

(A.20)
where g is the gravitational acceleration, \((\Delta \rho / \rho)\) the reduced density difference, \(\nu\) the kinematic viscosity and D the diffusion coefficient. If \((\Delta \rho / \rho)\) is considered to be 1% with \(\nu = 10^{-6} \text{ m}^2/\text{s}\) and \(D = 10^{-5} \text{ m}^2/\text{s}\), \(Ra\) is about 9.81 for 1 mm droplet and \(Ra=0.785\) for 20 \(\mu\)m diameter droplets. Small values of \(Ra\) shows that the internal free-convection (flow) is negligible in case of small droplets, however in large droplets, the free-convection effect becomes noticeable.

Again, thanks to this detailed calculation procedure, the flow field detail inside the droplet is available. There are many induced vortices known in literature as Marangoni Instability, which results from the surface tension gradient. This gradient is caused by the non-uniform temperature distribution at the surface which occurs due to a high evaporation rate in this small size droplet. This surface tension gradient produces a force in which surface fluid is pulled toward regions of higher surface tension. The viscous force then transports momentum into the interior of the droplet and a convective flow results throughout the droplet. This surface-tension-driven instability flow was described by Cloot and Lebon (1990) and was experimentally visualized by Hegseth et al. (1996). As shown in Fig. A6(b), regarding the flow field around and behind the droplet, it is stable with minimum laminar wake structure due to the very small Re number. The boundary layer can be also clearly seen over the droplet surface Fig. A6(b).

![Static temperature contour distribution](image)

Figure A5  Static temperature contour distribution in (a) the whole domain (b) enlarged view near the droplet

261
Water vapor distribution

The result of water vapor mass fraction (concentration) distribution in the base case is shown in Fig. A7. Figure A7(a) displays the water vapor mass fraction in the whole domain. Inlet water vapor mass fraction value is set zero (dry free stream) to allow a clear demonstration of the evaporation process. The pattern of water vapor concentration contour lines is similar to the isotherm pattern shown in Fig. A5, so the underlying physics is similar: the mass diffusion process is dominant near the droplet with slight enhancement due to natural convection. Although the effect of gravity enters the governing equations only via the momentum equation with the Boussinesq approximation, the temperature field affects the water vapor concentration and accordingly its transport indirectly.
Heat flux calculations

The aim at this juncture is to quantify the proportions of heat coming from the droplet and from the main air stream to feed the latent heat needed for evaporation. This is done by calculating the heat flux through the droplet surface and the shell boundary. As shown schematically in Fig. A8, the heat flux normal to the droplet surface is a vector quantity with two components in $x$- and $y$-directions. The heat flux components are calculated according to the following equations:

$$ q''_x = -k \frac{\partial T}{\partial x}, \quad q''_y = -k \frac{\partial T}{\partial y} \quad (A.21) $$

![Figure A8 Sign convention for the heat flux vector](image)

With the temperature gradient calculated as

$$ \frac{\partial T}{\partial n} = \frac{\partial T}{\partial r} = \sqrt{\left(\frac{\partial T}{\partial x}\right)^2 + \left(\frac{\partial T}{\partial y}\right)^2} \quad (A.22) $$

The resultant heat flux can be calculated as

$$ q''_n = -k \frac{\partial T}{\partial n} \quad (A.23) $$

The total amount of energy crosses the droplet surface or the shell surface can be calculated numerically by integrating the heat flux along this surface as

$$ Q = -kA \frac{\partial T}{\partial n} = \sum_i -k_i A_i \frac{\partial T_i}{\partial n} \quad (A.24) $$

The respective heat flux distributions through the droplet surface and the shell outer surface are plotted in Fig. A9 versus the circumferential angle after 3 time steps ($t = 0.000814$ s).
As the temperature of the droplet is lower than the freestream temperature, the temperature gradient (as shown in Fig. A5) is negative in the direction from the flow to the droplet. This means that the heat flux (which is of the opposite sign) is positive from the flow to the droplet. Consequently, the heat is entering the droplet, resulting in an increase of the droplet’s temperature until steady state equilibrium is reached. This temperature change is clear in Fig. A1(c), where the droplet temperature increases first (heat-up period) and then stays constant during evaporation. Also, as the temperature of the droplet increases, the temperature gradient decreases causing the heat flux entering the droplet to reduce until the heat flux becomes zero when steady-state equilibrium is reached. This interprets the results depicted in Fig. A1(c) showing the steady-state equilibrium temperature of the evaporating droplet. Moreover, Fig. A10 clearly shows that the heat flux entering the droplet is greatly reduced after 1000 time steps ($t = 0.0271$ s). Figure A11 shows the temperature vs. time diagram for a point at the droplet center. It is clear that the temperature, which was initially 300 K, increases with time until it reaches the steady-state equilibrium temperature as shown previously in Fig. A1(c). The droplet temperature is always increasing and never decreases. This above description of physics illustrates a different process from the built-in evaporation model centered at Eq. A6, which shows the potential for reducing the droplet temperature if the latent heat term is larger than the heat convection term.

![Figure A9 Heat flux through the droplet surface and the shell outer surface after 3 time steps ($t = 0.000814$ s). AAHF is the area adjusted heat flux crossing over the shell outer interface.](image-url)
Some extra notes can be mentioned about Figs. (A9 – A11). First, the heat flux plotted in Fig. A9 seems to reveal that more energy is entering the droplet zone than the energy entering the shell, which is not the correct interpretation. The true picture can be seen as the heat fluxes crossing over the shell interface is adjusted for its approximately 4% larger surface area than the droplet interface surface area. This Area-Adjusted Heat Flux (AAHF) is shown as the dotted blue curve, which almost coincides with the curve representing the heat flux entering the droplet excluding the local heat flux maxima. Second, the highest heat flux occurs near 30° from the leading edge. The second highest heat flux occurs at the leading edge and the third heat flux peak occurs at around 90° from the leading edge. These local maxima all occur during the earlier transient period. When the time step reaches 0.0271, those secondary local maxima are smoothed out and the heat flux has significantly decreased to about an order magnitude smaller. The
maximum heat flux occurs at the familiar location at the stagnation point as shown in Fig. A10. The temperature at the droplet’s center, Fig. A11, seems to follow a continuously increasing trend, even at 0.016 second. A coarse extrapolation gives a thermal response time of approximately 0.02 seconds to reach 302K. These 0.02 seconds of thermal response time is about an order of magnitude longer than 0.0025 seconds obtained from Eq. (A.11) in Fig. A1(c). This discrepancy is caused by the employment of a constant droplet size, rather than using a diminishing droplet size in the current CFD model.

To show the effect of Reynolds number on the calculations, a larger droplet size (1 mm) is used with the same inlet air velocity of 1 m/s to yield a Reynolds number of $Re = 61.6$. Higher inlet air temperature of 400 K is used in this case to drive a significant temperature gradient through the droplet. The rest of the parameters are kept as in the base case of 20 µm droplets. As the Reynolds number increases, it can be clearly seen in Fig. A12 that the convection effect becomes dominant. Moreover, as the droplet size increases, the temperature distribution inside the droplet becomes more uniform, and Marangoni instability and natural convection start to occur inside the droplet, as shown in Fig. A13. The bigger the droplet size is, the more the conduction resistance is inside the droplet. This leads to an increased temperature gradient inside the droplet (around 5 degrees difference in temperature inside this droplet of 1 mm diameter).

From the calculations of Biot number (0.1328) for this 1mm droplet (shown in Table A3), a large temperature gradient is expected inside the droplet; thus, a lower amount of heat is sucked from the droplet due to its higher thermal resistance. However, unexpectedly, the results obtained earlier in Figs. A5, A9-A13 show a different situation. All these figures show that the heat is transferred from the main flow to the droplet for different conditions of droplet size, and main stream temperatures and the droplet temperature never reduces.

The question now is how to interpret the droplet temperature reduction observed in using the built-in evaporation model in the commercial code. The plausible answer is that when the droplet evaporation rate is extremely high, the amount of energy sucked at the droplet surface is so high to the degree that part of the evaporation energy is sucked from the droplet, resulting in a momentary reduction of droplet temperature. To showcase this high evaporation rate scenario, a number of cases have been conducted with high evaporation rates by implementing high mass sources and negative energy sources at the droplet surface. Unfortunately, none of them gave a converged solution. Divergence was always detected at the beginning of the calculation as a
result of the increased fluxes through the interfaces. That unstable behavior prevented the authors from showcasing that the droplet temperature can ever reduce.

Figure A12 Effect of increased Reynolds number: static temperature contour distribution in (a) the whole domain (b) near the droplet (1 mm diameter droplet, Re = 61.6).

Figure A13 Static temperature contour inside the droplet (1 mm dia. and Re_d = 61.6).
CONCLUSIONS

This study is motivated by the observation of the droplet temperature’s temporarily dropping too low during evaporation process when a built-in evaporation model is used in a commercial CFD code. Three approaches have been made: (a) using the build-in evaporation model with empirical equations, (b) using a zero-dimensional analysis, and (3) using a commercial CFD to directly calculate the local heat transfer around a droplet without using the built-in empirical equations. (Note: This is not the direct numerical analysis or DNS). The objective of this work is to understand the process and identify the problem of the built-in evaporation models available in most commercial codes.

The result of the zero-dimensional analysis indicates that the droplet temperature always increases. The direct CFD result also draws the same conclusion that the droplet temperature never drops during the evaporation process.

A modification of the droplet temperature equation is proposed with a factor added to account for the effect of Biot number, which would reduce the latent heat to be sucked from the liquid droplet. The direct CFD method adequately captures the flow structure and Marangoni motion inside the droplet and provides information of heat flux surrounding the droplet.
VITA

Reda Ragab was born on November 1st, 1978 in Sharkia, Egypt to Boshraa Khairie and Mohammed Gad Ragab. He received his high school education in Safor Secondary School, Dyarb Negim, Sharkia, Egypt and graduated on May, 1995. In September 1995, he was admitted to Mataria Faculty of Engineering, Helwan University, Cairo, Egypt. A year after, he transferred to Zagazig University, Zagazig, Sharkia, Egypt. In May 2000, he received his Bachelor of Science Degree in Mechanical Power Engineering. Following his graduation, and after finishing his military service in the Egyptian army, he was appointed as a tenure demonstrator “Moeed” in the Mechanical Power Engineering department at Zagazig University doing research and teaching undergraduate classes.

In March 2003, he was admitted to the Graduate School of Zagazig University since March 2003 to May 2008 and received his degree of Masters of Science in Engineering in May 2008. Following his graduation he was promoted to the position of tenure Lecturer Assistant “Modares Mosaed”.

In August 2009, he was admitted to the Graduate School of the University of New Orleans, and was employed as a graduate assistant by the Department of Mechanical Engineering since August 2008 to December 2013. He will finish his PhD in Mechanical Engineering on December 2013.